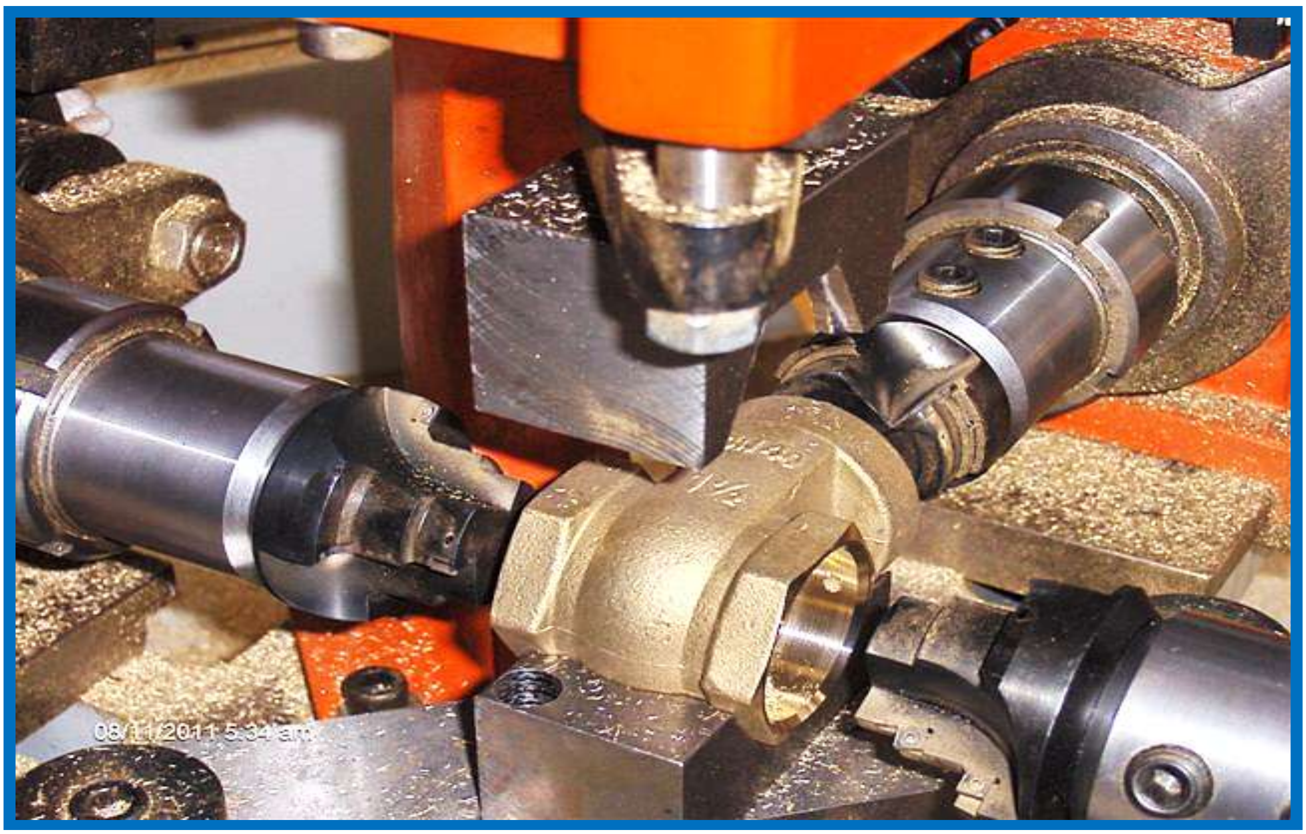
PDHonline Course M 497 (6 PDH)

Conventional Machining Technology Fundamentals

Instructor: Jurandir Primo, PE



2013

PDH Online | PDH Center

5272 Meadow Estates Drive
Fairfax, VA 22030-6658
Phone: 703-988-0088

www.PDHonline.com
www.PDHcenter.org

An Approved Continuing Education Provider

CONVENTIONAL MACHINING TECHNOLOGY – FUNDAMENTALS

OBS.: This is a professional didactic manual. It´s highly recommended downloading and printing the course content for your study, before answering the quiz questions.

Introduction:

Machining is any of various processes in which a piece of raw material is cut into a desired final shape and size by a controlled material-removal process. The precise meaning of the term "machining" has evolved over the past two centuries as technology has advanced.

The "traditional" machining processes are; turning, boring, drilling, milling, sawing, broaching, shaping, planing, reaming and tapping. In these "traditional" or "conventional" machining processes, the machine tools are designated as, lathes, milling machines, planers, broaching machines, etc., used with sharp cutting tools to remove materials to achieve a desired geometry.

Since the advent of new technologies such as, CNC center machines, electrical discharge machining, electrochemical machining, electron beam machining, photochemical machining, ultrasonic machining, etc., conventional machining is used to differentiate from these modern technologies. The term "machining" without an up-to-date qualification, usually implies the traditional machining processes.

Machining is a part of the manufacture of many metal products, but it can also be used on materials such as wood, plastic, ceramic, and composites. A person who specializes in machining is called a "machinist" or a machine operator, and a room, building, or company where machining is done is called "machine shop". Machining can be a business, a hobby, or both. The modern machining nowadays is carried out by Computer Numerical Control (CNC), in which computers are used to control the movement and operation of the cutting machines. The traditional machine tools commonly used for general machining purposes, are:

- Lathes;
- Drilling Machines;
- Milling Machines;
- Planning Machines;
- Shaping Machines;
- Slotting Machines;
- Boring Machines;
- Hobbing Machines;
- Gear Shaping Machines;
- Broaching Machines;
- Grinding Machines.

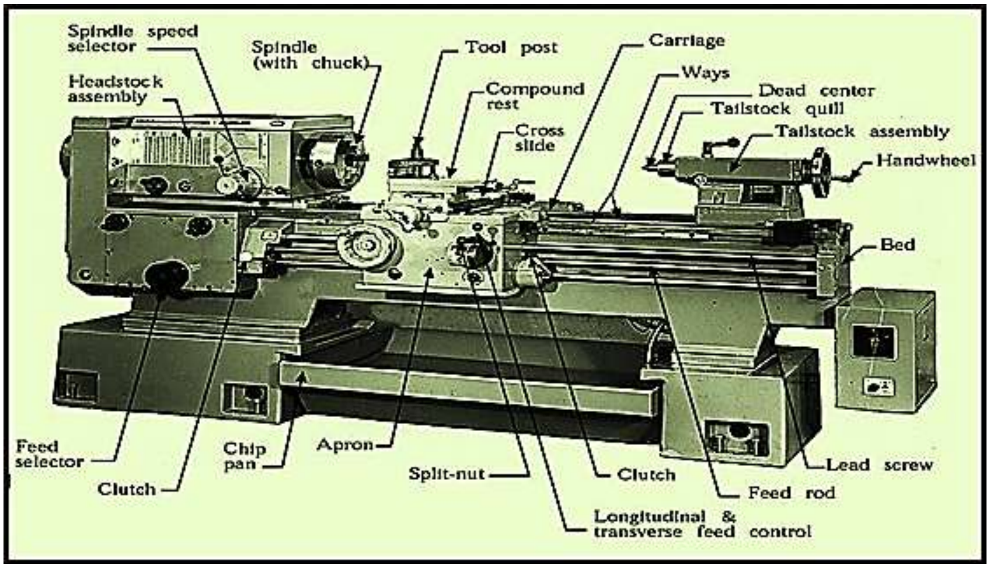
Lathes:

A lathe is a machine tool with a spindle that rotates the workpiece on its axis to perform various operations, such as cutting, sanding, knurling, drilling, formation, facing, turning, with tools that are applied to the workpiece to create an object, which has symmetry about an axis of rotation.

Lathes are used in woodturning, metalworking, metal spinning, thermal spraying/parts and glass-working. Lathes can be used to shape pottery, the best-known design being the potter's wheel. Most suitably equipped metalworking lathes can also be used to produce most solids of revolution, plane surfaces and screw threads or helices.

Metalworking Lathes:

In a metalworking lathe, the metal is removed from the workpiece using a hardened cutting tool, which is usually fixed to a solid moveable mounting, either a tool-post or a turret, which is then moved against the workpiece using hand wheels and/or computer controlled motors. These (cutting) tools come in a wide range of sizes and shapes depending upon their application. Some common styles are diamond, round, square and triangular.



The tool-post is operated by lead-screws that can accurately position the tool in a variety of planes. The tool-post may be driven manually or automatically to produce the roughing and finishing cuts required turning the workpiece to the desired shape and dimensions, or for cutting threads, worm gears, etc. Cutting fluid may also be pumped to the cutting site to provide cooling, lubrication and clearing of swarfs from the workpiece. Some lathes may be operated under control of a computer for mass production of parts (see CNC - "Computer Numerical Control").

Manually controlled metalworking lathes are commonly provided with a variable ratio gear train to drive the main lead-screw. This enables different thread pitches to be cut. On some older lathes or more affordable new lathes, the gear trains are changed by swapping gears with various numbers of teeth on or off the shafts, while more modern or expensive manually controlled lathes have a quick change box to provide commonly used ratios by the operation of a lever. CNC lathes use computers and servomechanisms to regulate the rates of movement.

Classification of Lathes:

Lathes are very versatile of wide use and are classified according to several aspects:

(a) According to configuration:

• Horizontal - Most common for ergonomic conveniences
• Vertical - Occupies less floor space, only some large lathes are of this type.

(b) According to purpose of use:

• General purpose - Very versatile where almost all possible types of operations are carried out on wide ranges of sizes, shapes and materials of jobs, example: center lathes;
• Single purpose - Only one (occasionally two) type of operation is done on limited ranges of size and material of jobs, example – facing lathe, roll turning lathe, etc.;
• Special purpose - Where a definite number and type of operations are done repeatedly over long time on a specific type of blank, example: gear blank machining lathe, etc.

(c) According to size or capacity:

Small (low duty) - In such light duty lathes (up to 1.1 kW), only small and medium size, generally soft and easily not so hard materials are machined;
Medium (medium duty). These lathes of power nearly up to 11 kW are most commonly used;

Large (heavy duty);
Mini or micro lathe. These are tiny table-top lathes used for extremely small size jobs and precision work; example: Swiss type automatic lathe.

(d) According to degree of automation:

Non-automatic - Almost all the handling operations are done manually, example: center lathes;
Semi-automatic - Nearly half of the handling operations are done automatically and rest manually, example: capstan lathe, turret lathe, copying lathe relieving lathe, etc.;
Automatic - Almost all the handling operations are done automatically, example – single spindle automat (automatic lathe), Swiss type automatic lathe, etc.

(e) According to type of automation:

Fixed automation - Common: example – single spindle automatic, Swiss type automatic lathe, etc.
Flexible automation – Machine Centers: example CNC lathe, turning centers, etc.

(f) According to configuration of the jobs being handled:

Bar type - Slender rod like jobs being held in collets
Chucking type - Disc type jobs being held in chucks
Housing type - Odd shape jobs, being held in face plate

(g) According to precision:

- Ordinary
- Precision (lathes) - for high accuracy and finish and are relatively more expensive.

(h) According to number of spindles:

Single spindle - Common
Multispindle (2, 4, 6 or 8 spindles) - used for fast and mass production of small size and simple shaped jobs.

(i) Common lathe machining operations:

Facing;
Centering;
Rough and finish turning;
Chamfering, shouldering, grooving, recessing, etc.
Axial drilling and reaming by holding the cutting tool in the tailstock barrel;

Taper turning by:
– Offsetting the tailstock;
– Swiveling the compound slide;
– Using form tool with taper over short length;
– Using taper turning attachment if available;
– Combining longitudinal feed and cross feed, if feasible.

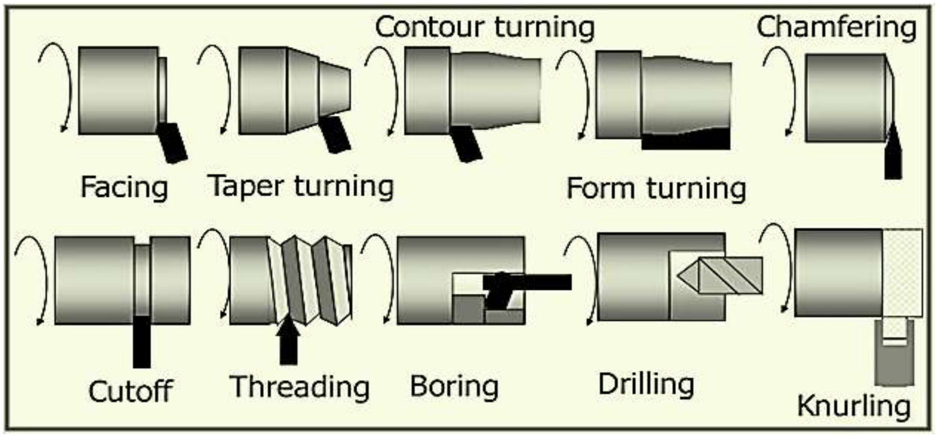
– Boring (internal turning); straight and taper;
– Forming - external and internal;
– Cutting helical threads - external and internal;
– Cutoff;
– Knurling.

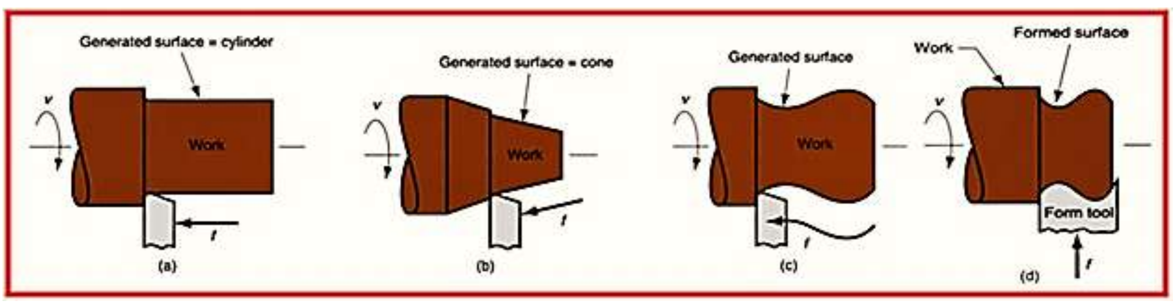
Some more operations are also occasionally done, if desired, in center lathes by mounting suitable attachments available in the market, such as:

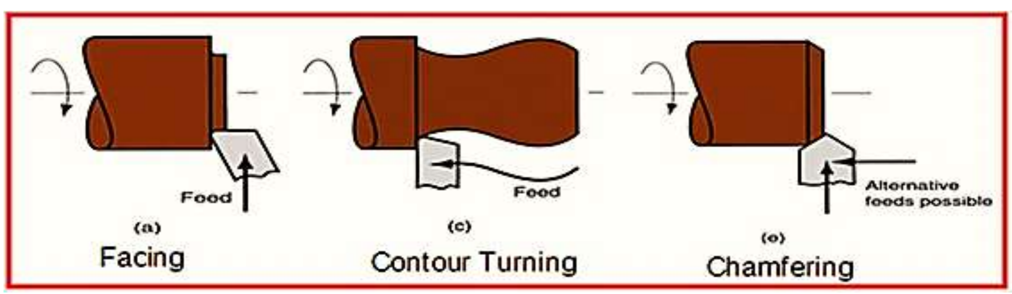
Grinding both external and internal by mounting a grinding attachment on the saddle;
Copying (profiles) by using hydraulic copying attachment;
Machining long and large threads for lead-screws, power-screws, worms, etc., by using thread milling attachments.

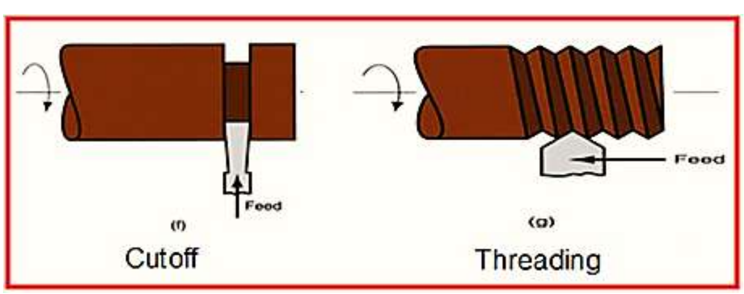
Turning Operations:

A single point cutting tool removes material from a rotating workpiece to generate a cylindrical shape.



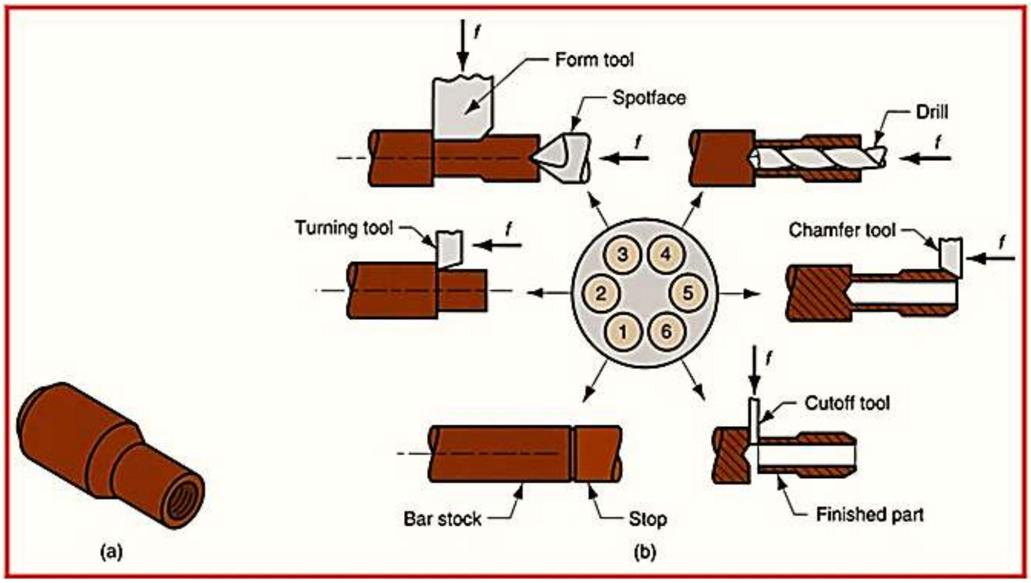






Below is shown a workpiece produced on a six-spindles automatic bar machine and (b) the sequence of operations to produce the machining job:

(1) Feeding stock to a lathe;
(2) Turning the main diameter;
(3) Forming the second diameter and spotting face;
(4) Drilling;
(5) Chamfering;
(6) Cutting off.



Semiautomatic and Automatic Lathes:

Automation is incorporated in a machine tool or machining system as a whole for higher productivity with consistent quality aiming meeting the large requirements and overall economy. Such automation enables quick and accurate auxiliary motions, i.e., handling operations like tool – work mounting, bar feeding, tool indexing etc., with minimum human intervention but with the help of special or additional mechanism and control systems. These systems may be mechanical, electro-mechanical, hydraulic or electronic type or their combination. It is already mentioned that according to degree of automation machine tools are classified as:

• Non automatic where most of the handling operations irrespective of processing operations, are done manually, like center lathes, etc.
• Semiautomatic and Automatic where all the handling or auxiliary operations as well as the processing operations are carried out automatically.

General purpose machine tools may have both fixed automation and flexible automation where the latter one is characterized by computer Numerical Control (CNC). Amongst the machine tools, lathes are most versatile and widely used. The conventional general purpose automated lathes can be classified as:

a) Semiautomatic Lathes:

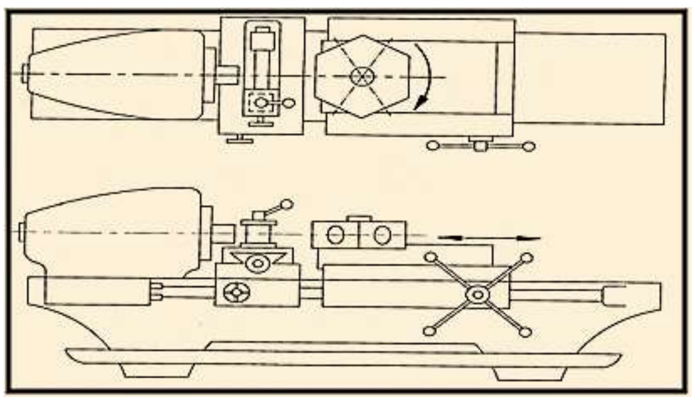
• Capstan lathe (ram type turret lathe);
• Turret lathe;
• Multiple spindle turret lathes;
• Copying (hydraulic) lathe.

b) Automatic Lathes:

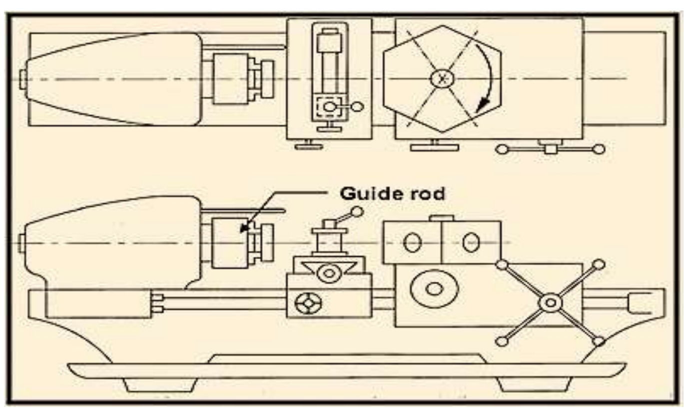
• Automatic cutting off lathe;
• Single spindle automatic lathe;
• Swiss type automatic lathe;
• Multiple spindle automatic lathes.

Capstan lathes and turret lathes are **semiautomatic types** and very similar in construction, operation and application. Below, shows the basic configuration of a **capstan lathe** and the **turret lathe**. In contrast to center lathes, capstan and turret lathes:

• are semiautomatic, possess an axially movable indexable turret (mostly hexagonal) in place of tailstock;
• holds large number of cutting tools, up to four in indexable tool post on the front slide, one in the rear slide and up to six in the turret (if hexagonal) as indicated in the schematic diagrams;
• are more productive for quick engagement and overlapped functioning of the tools in addition to faster mounting and feeding of the job and rapid speed change;
• enable repetitive production of same job requiring less involvement, effort and attention of the operator for pre-setting of work–speed and feed rate and length of travel of the cutting tools;
• relatively costlier, suitable and economically viable for batch production or small lot production.



Capstan lathe



Turret lathe

Obs.: Ram type turret lathes, or capstan lathes are usually single spindle and horizontal axis type. Turret lathes are also mostly single spindle and horizontal type but it may be also:

• Vertical type
• Multispindle type

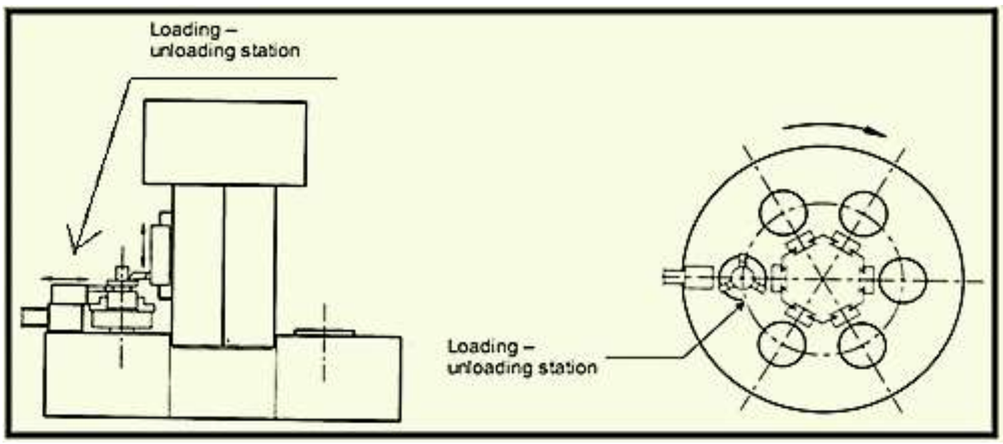
Some more productive turret lathes are provided with preemptive drive, which enables on-line presetting and engaging the next work-speed, and thus help in reducing the cycle time.

Turret Lathes: Turret lathes are mostly horizontal axis single spindle type.

Δ chucking type;
Δ relatively large size;
Δ requiring limited number of machining operations;
Δ lesser floor space occupied;
Δ easy loading and unloading of blanks and finished jobs;
Δ relieving the spindles of bending loads due to job – weight;
Δ number of spindle – four to eight.

The multiple spindle vertical turret lathes are characterized by large lot or mass production of jobs of generally. Vertical turret lathes are of **three categories**:

Parallel Process: The spindle carrier remains stationary. Only the cutting tools moves radially and axially. Identical pieces (say six) are simultaneously mounted and machined in the chucks, parallel at all stations, each one having same set of axially and / or radially moving cutting tools.



Progressively Process: The spindle carrier with the blanks fitted in the chucks on the rotating spindle is indexed at regular interval by a Geneva mechanism. At each station the job undergoes a few preset machining works by the axially and / or radially fed cutting tools. The blank getting all the different machining operations progressively at the different workstations are unloaded at a particular station where the finished job is replaced by another fresh blank. This type of lathes is suitable for jobs requiring large number of operations.

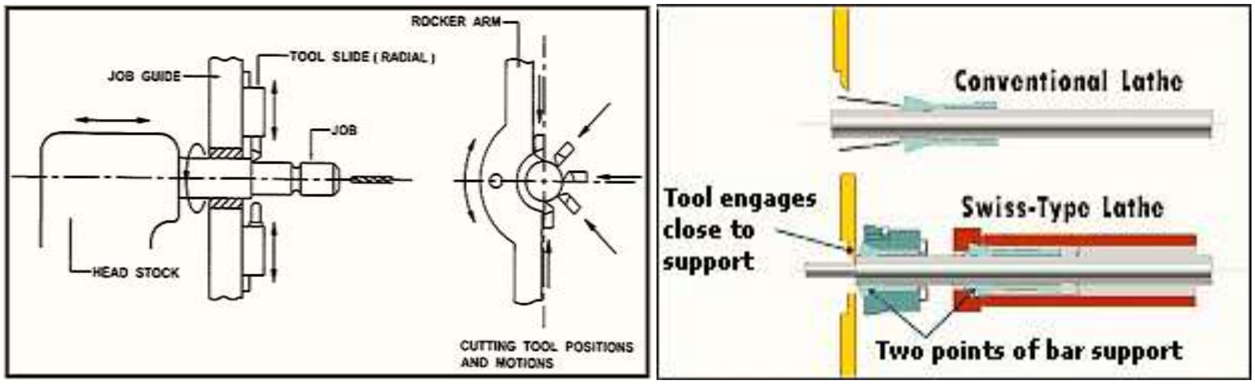
Continuously Working: Like in parallel processing type, here also each job is finished in the respective station where it was loaded. The set of cutting tools, mostly fed only axially along a face of the ram continuously work on the same blank throughout its one cycle of rotation along with the spindle carrier. The tool ram having same tool sets on its faces also rotate simultaneously along with the spindle carrier, which after each rotation halts for a while for unloading the finished job and loading a fresh blank at a particular location. Such system is also suitable for jobs requiring simple machining operations.

Hydraulic Copying (Tracer Controlled) Lathes: Parts having steps, tapers and / or curved profiles, as typically shown below, are conveniently and economically produced in batch or lot in semi-automatically operated tracer controlled hydraulic copying lathe. The movement of the stylus along the template provided with the same desired job-profile) is hydraulically transmitted to the cutting tool tip, which replicates the template profile.

c) Types of Automatic Lathes:

Automatic lathes are essentially used for large lot or mass production of small rod type of jobs. Automatic lathes are also classified into some distinguished categories based on constructional features, operational characteristics, number of spindles and applications as follows:

• **Single Spindle Lathes:** Automatic cutting off lathes; Automatic (screw cutting) lathe; Swiss type automatic lathe, Multispindle automatic lathes:



Swiss type automatic lathe

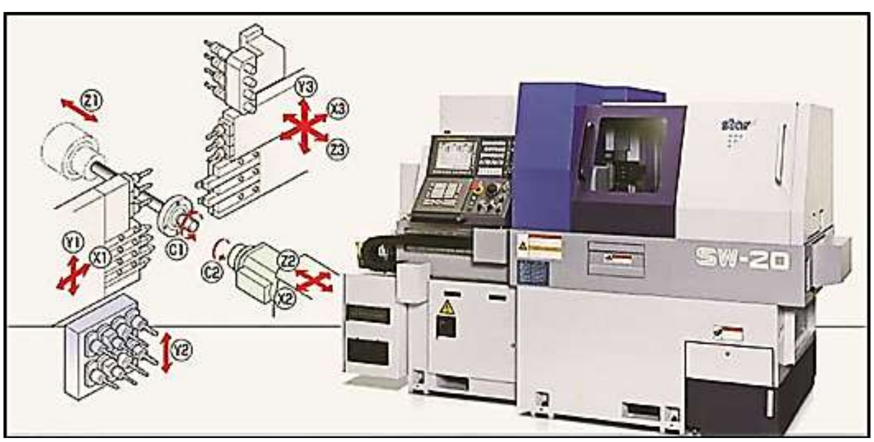
• **Swiss Type Automatic Lathes:** The characteristics and applications of these single spindle automatic lathes are:

- ✓ **Multispindle Automatic Lathes:** Increase the rate of production of jobs usually of smaller size and simpler geometry. Multispindle automatic lathes have four to eight parallel spindles and are preferably used.

- ✓ The multispindle automatic lathes or multiple spindle automats may be of parallel action or progressively working type. Machining of the inner and outer races in mass production of ball bearings is for instance, machined in multispindle automatic lathes. The characteristics are:

- Horizontal (for working on long bar stocks);
- Work mostly on long bar type or tubular blanks.

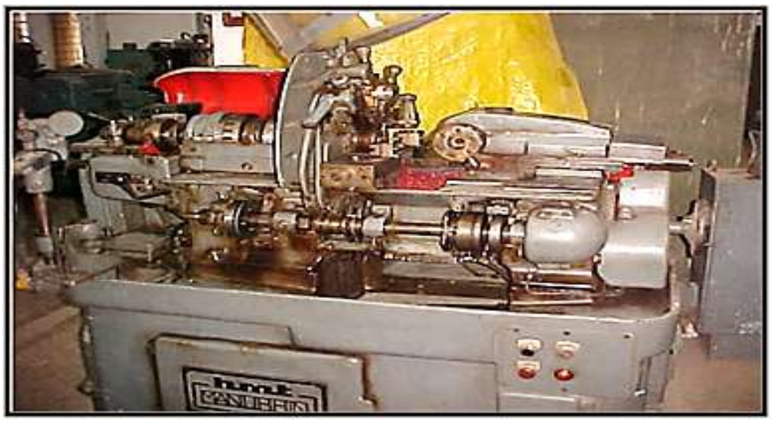
- **Mass Production Lathes:** Thin slender rod or tubular jobs, like components of small clocks and wristwatches, by precision machining. The dimensional accuracy and surface finish are almost as good as provided by grinding. Job size (approximately): Diameter range – 2 to 12 mm; Length range – 3 to 30 mm.



- ✓ **Automatic Cutting Off Lathes:** These simple but automatic lathes are used for producing short work pieces of simple form by using few cross feeding tools. In addition to parting some simple operations like short turning, facing, chamfering etc. are also done.

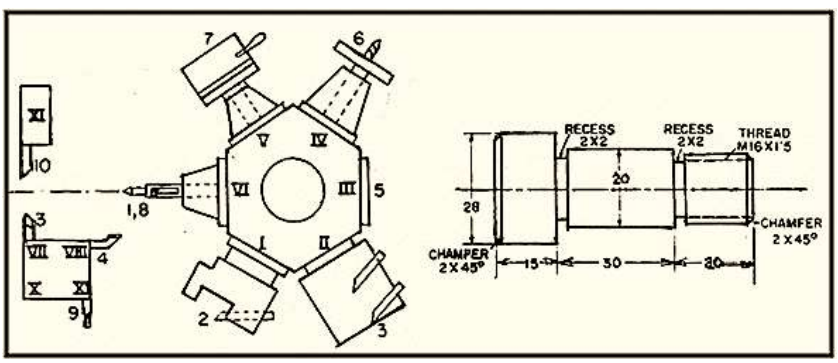
- ✓ **Single Spindle Automatic Lathes:** The general purpose single spindle automatic lathes are used for quantity or mass production (by machining) of high quality fasteners; bolts, screws, studs etc., bushings, pins, shafts, rollers, handles and similar small metallic parts from long bars or tubes of regular section and also often from separate small blanks. Unlike the semiautomatic lathes, single spindle automats are:

• Single spindle automatic lathes are essentially used for larger volume of production, i.e., large lot production and mass production;
• Used always for producing jobs of rod, tubular or ring type and of relatively smaller size;
• Run fully automatically, including bar feeding and tool indexing, and continuously over a long duration repeating the same machining cycle for each product;
• Provided with up to five radial tool slides, which are moved by cams mounted on a camshaft;
• Relatively smaller size and power but have higher spindle speeds.



Single spindle automatic lathe

Tool layout: A typical tool layout for a particular job is schematically shown below.



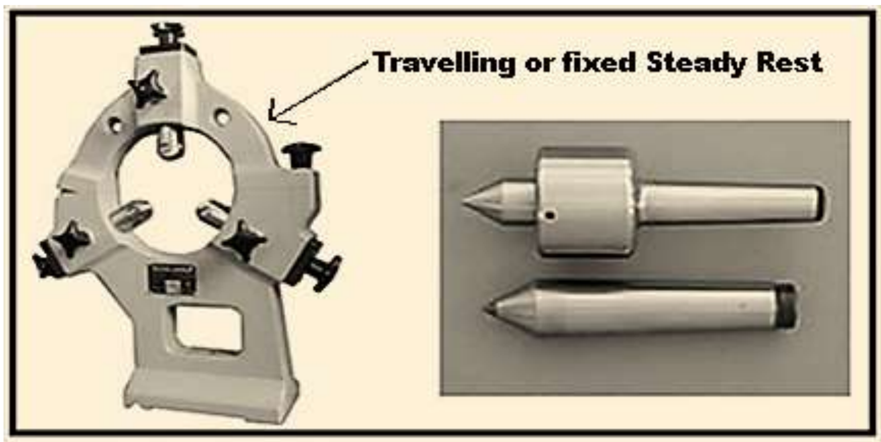
Tool layout for a typical job in a single automatic lathe.

Lathe Accessories:

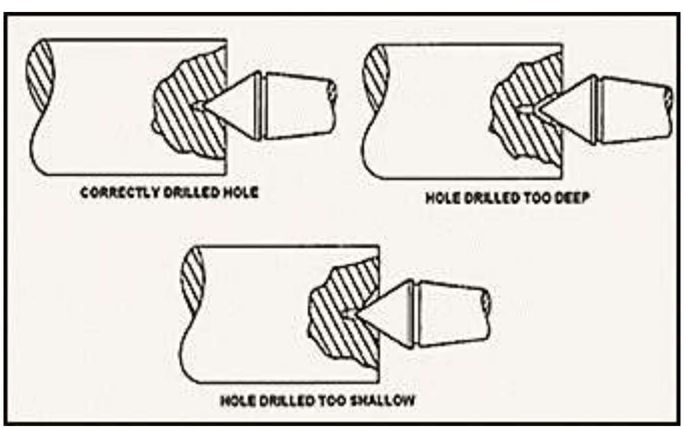
Workpieces: May be mounted on a mandrel or circular work clamped in a three-or four-jaw chuck. For irregular shaped workpieces, it is usual to use a four jaw (independent moving jaws) chuck. These holding devices mount directly to the lathe headstock spindle. In precision work or repetition work, cylindrical workpieces are usually held in a collet inserted into the spindle and secured either by a drawbar, or by a collet-closing cap on the spindle.

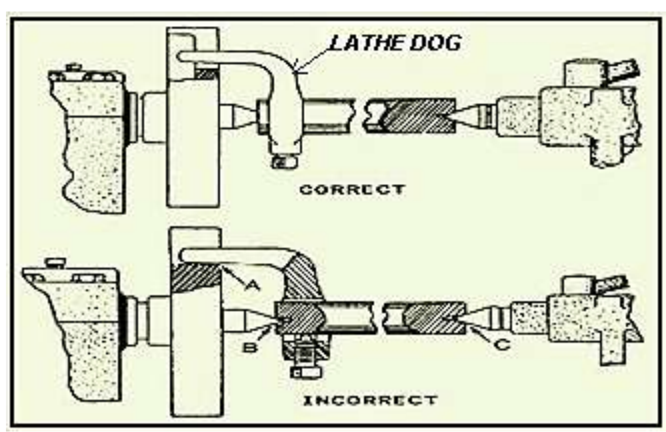
Collets: Are holding devices that form a collar around the machining tool to be held, exerting a strong clamping force when it is tightened, usually by means of a tapered outer collar, but may be used to hold also a workpiece. Whereas for most repetition work purposes, the dead length variety is preferred, this must ensure that the position of the tool or workpiece does not move as the collet is tightened.

Dead and Live Centers: Are used in the headstock spindle as the work rotates with the center. Because the center is soft it can be placed before use. The included angle is 60°. Traditionally, a hard dead center is used together with suitable lubricant in the tailstock to support the workpiece. In modern practice, the dead center is frequently replaced by a live center, as it turns freely with the workpiece — usually on ball bearings — reducing the frictional heat, especially important at high speeds. When facing a long length of material, it must be supported at both ends. This can be achieved by the use of a traveling or fixed steady. A half center has a flat surface machined across a broad section of half of its diameter at the pointed end. A small section of the tip of the dead center is retained to ensure concentricity.

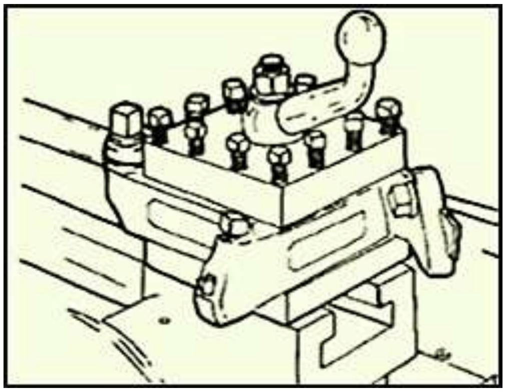
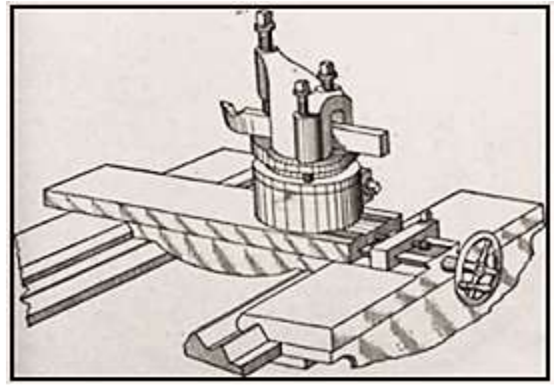


Mounting a Work Between Centers: A correctly drilled and countersunk hole, has a uniform 60° taper and a clearance at the bottom for the point of the lathe center. The figure below illustrates correctly and incorrectly drilled center holes. Drilling and countersinking of center holes can be done on a drilling machine or on the lathe itself. The end of the workpiece must be machined flat to keep the center drill from running off center.

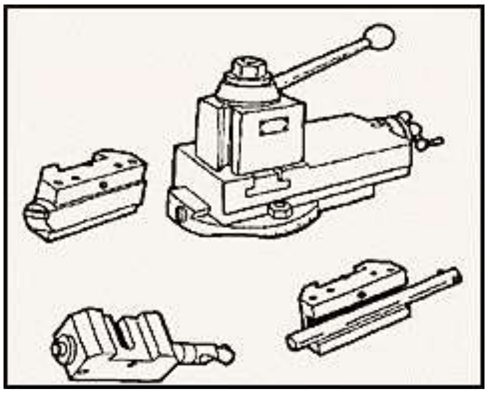
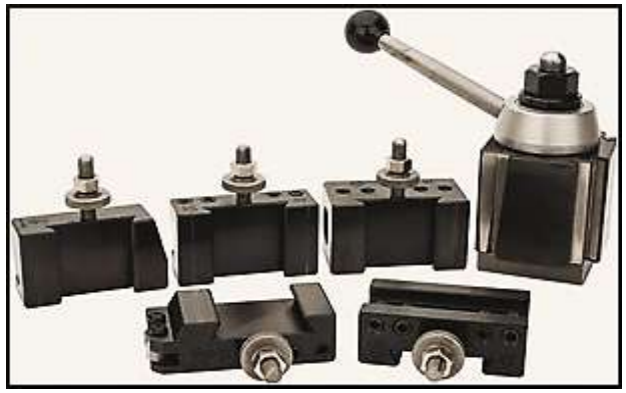




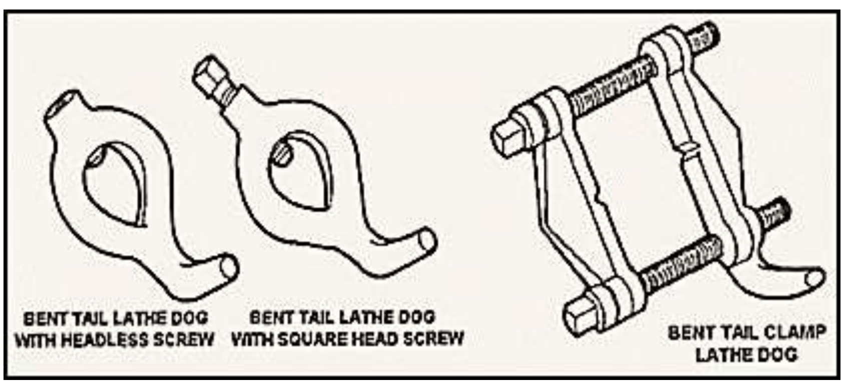
Turret Tool Posts: Other tool holders to fit the standard round tool post include straight, left, and right parting tool holders, knurling tool holders, boring bar tool holders, and specially formed thread cutting tool holders. The turret tool post is a swiveling block that can hold many different tool bits or tool holders, and each cutting tool can quickly be swiveled into cutting position and clamped into place using a quick clamping handle. The turret tool post is used mainly for high-speed production operations.

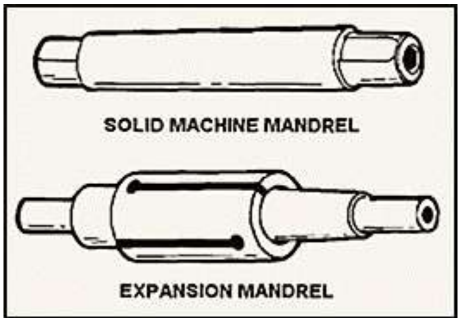
Quick-change Tool System: Consists of a quick-change tool post with a complete set of matching dovetailed tool holders that can be quickly changed as different lathe operations become necessary. This system has a quick-release knob on the top of the tool post that allows tool changes in less than 5 seconds, valuable for production shops.

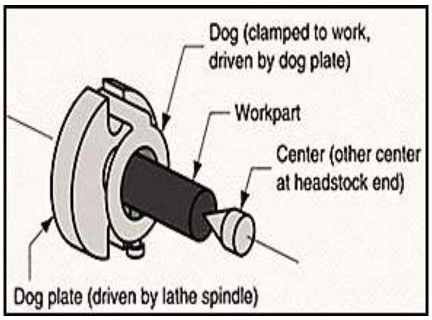
Bent-Tail Dogs: the tail fits into a slot of the driving faceplate. When straight-tail dogs are used, the tail bears against a stud projecting from the faceplate. The bent-tail lathe dog with headless setscrew is considered safer than the dog with the square head screw because the headless setscrew reduces the danger of the dog catching in the operator's clothing and causing an accident. The bent-tail clamp lathe dog is used primarily for rectangular workplaces.

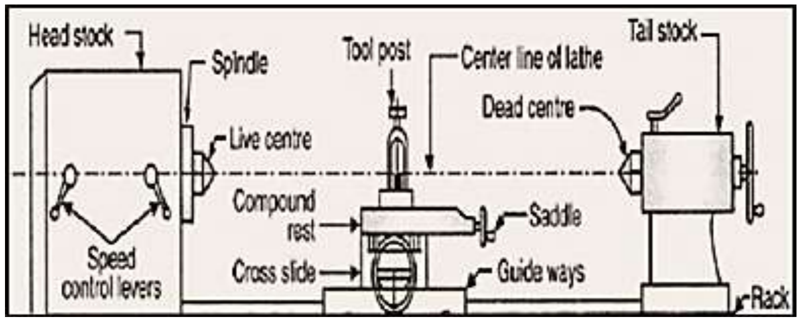


Mandrels: are tapered axle pressed into the bore of the workpiece to support it between centers. A mandrel should not be confused with an arbor, which is a similar device but used for holding tools rather than workplaces. To prevent damage to the work, the mandrel should always be oiled before being forced into the hole. When turning work on a mandrel, feed toward the large end which should be nearest the headstock of the lathe.

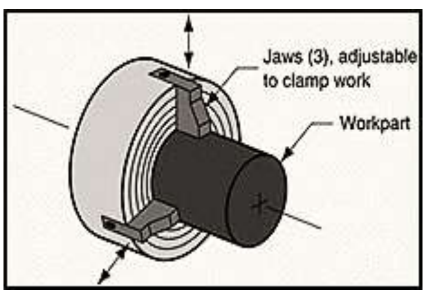


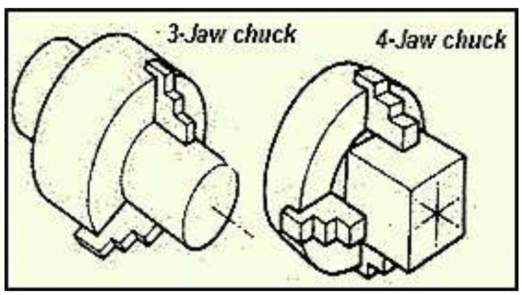
The centers are live centers or dead centers. A live center revolves with the work and does not need to be lubricated and hardened. A dead center does not revolve with the work and must be hardened and heavily lubricated when holding work. Live and dead centers commonly come in matched sets, with the hardened dead center marked with a groove near the conical end point.

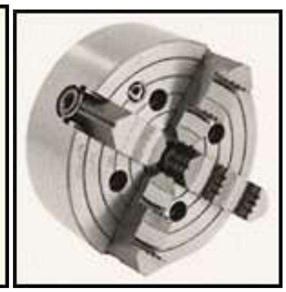




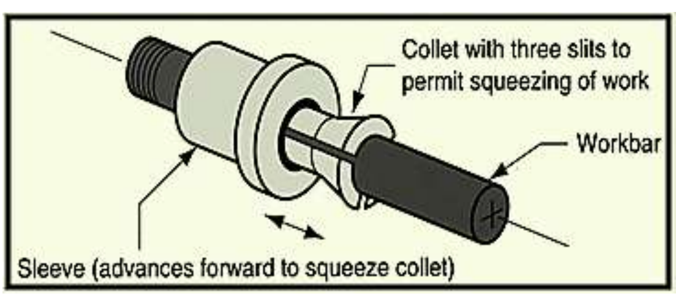
Chucks: The universal scroll chuck is used to hold and automatically center round or hexagonal workplaces. Having only three jaws, the chuck cannot be used effectively to hold square, octagonal, or irregular shapes. The independent chuck, generally has four jaws which are adjusted individually on the chuck face by means of adjusting screws.



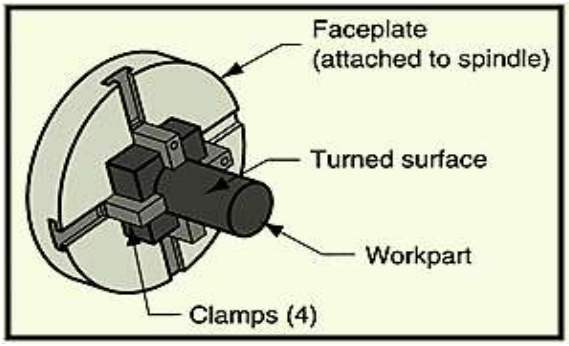




Collets: The collet chuck is the most accurate means of holding small workpieces in the lathe. For general purposes, the spring machine collets are limited in capacity to 1 1/8 inch in diameter. The collet attachment consists of a collet sleeve, a drawbar, and a hand wheel or hand lever to move the drawbar. The spring machine collet and collet attachment together form the collet chuck, as shown below.

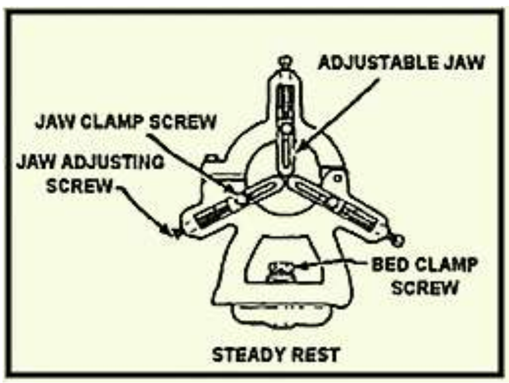


Face Plate for Non-cylindrical Workparts: A lathe faceplate is a flat, round plate that threads to the headstock spindle of the lathe. The faceplate is used for irregularly shaped workplaces that cannot be successfully held by chucks or mounted between centers. The workpiece is either attached to the faceplate using angle plates or brackets or bolted directly to the plate.

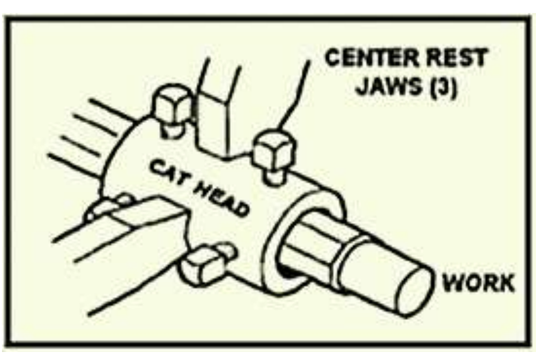


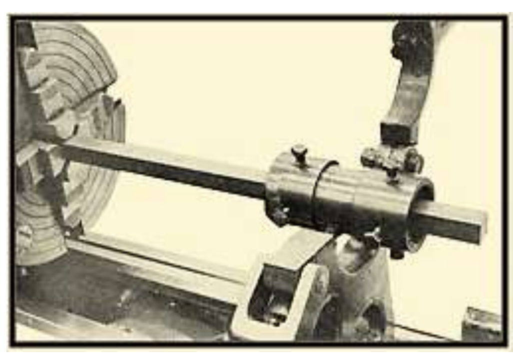
Rests and Micrometer Supports: Workpieces often need extra support, especially long, thin workplaces that tend to spring away from the tool bit. Other common lathe attachments are the steady rest with cathead, the follower rest and the lathe micrometer stop.

- ✓ **Steady Rest:** The steady rest, also called a center rest, is used to support long workplaces for turning and boring operations. It is also used for internal threading operations where the workpiece projects a considerable distance from the chuck or faceplate. The steady rest is clamped to the lathe bed at the desired location and supports the workpiece within three adjustable jaws.

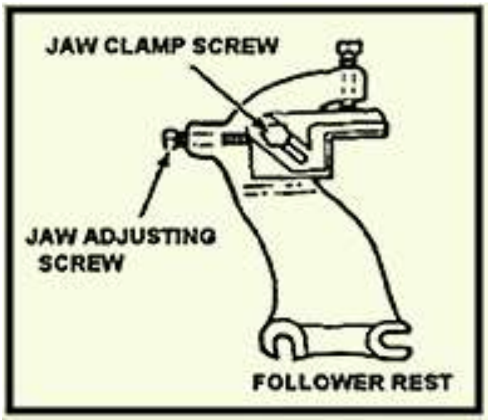



- ✓ **Cathead:** Has a bearing surface, a hole through which the work extends, and adjusting screws. The adjusting screws fasten the cathead to the work. They are also used to align the bearing surface so that it is concentric to the work axis. A dial indicator must be used to set up the cathead to be concentric and accurate.

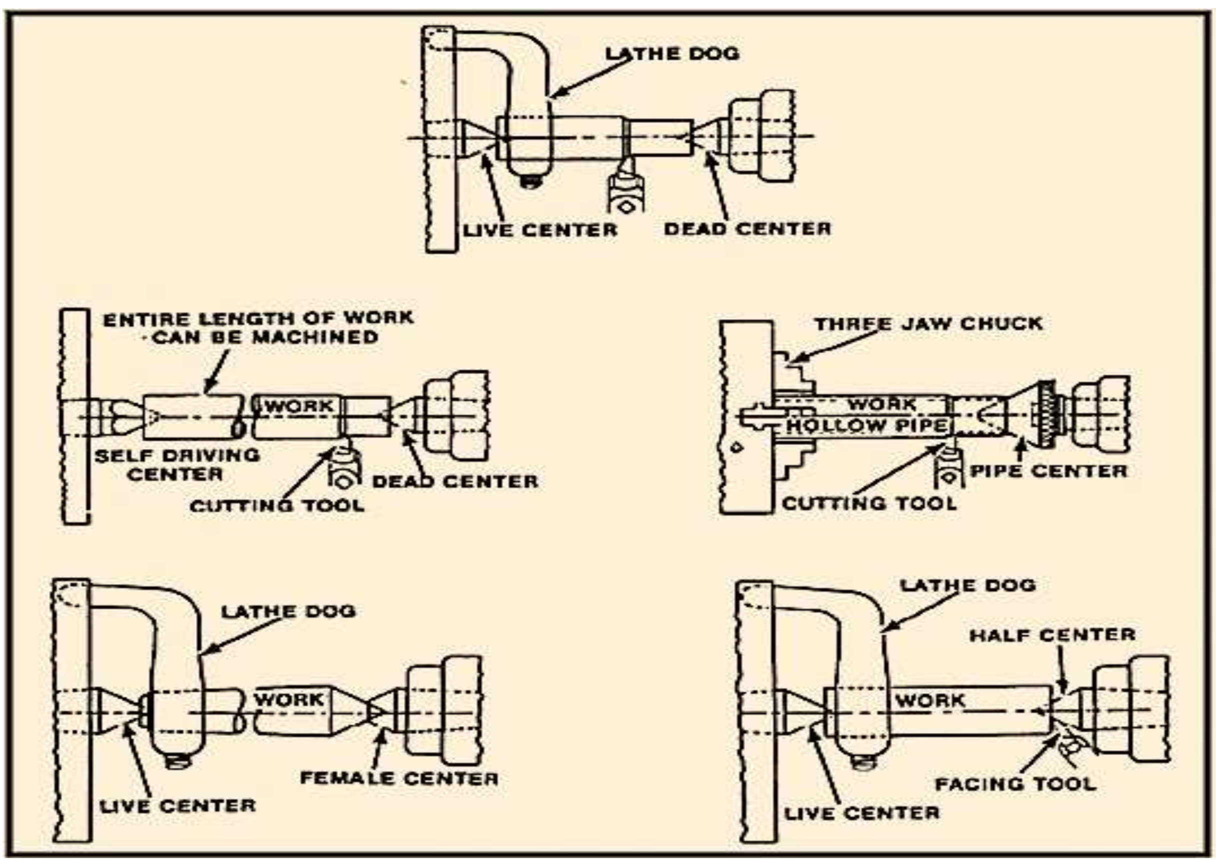




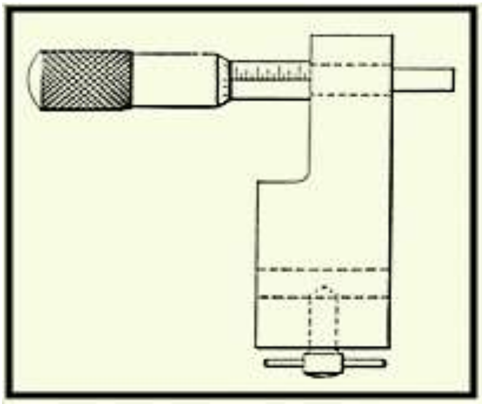
- **Follower Rest:** The follower rest has one or two jaws that bear against the workpiece. The rest is fastened to the lathe carriage so that it will follow the tool bit and bear upon the portion of the workpiece that has just been turned. The cut must first be started and continued for a short longitudinal distance before the follower rest may be applied.



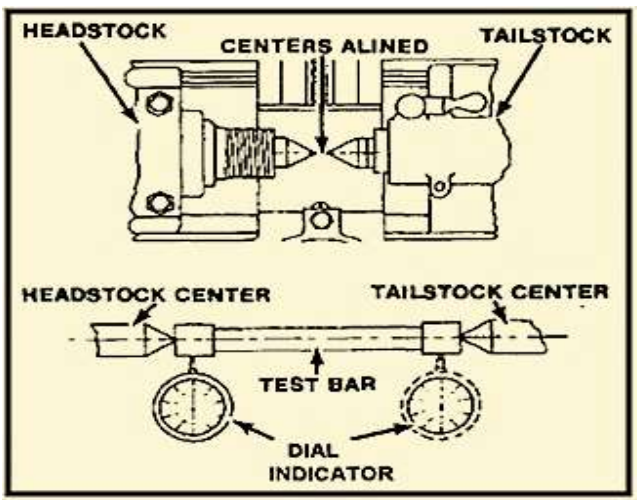
- **Lathe Dog:** The lathe dog tail must move freely in the slot of the faceplate and not bind. A few drops of oil mixed with white lead should be applied to the center before the workpiece is set up. The tailstock should be adjusted so that the tailstock center fits firmly into the center hole of the workpiece. The lathe should be stopped at intervals and additional oil and white lead mixture applied to the dead center to prevent overheating harm to the center and the workpiece

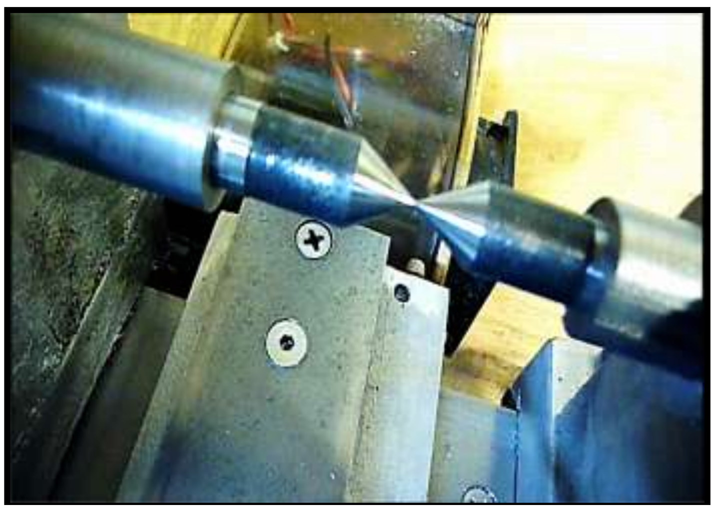


- ✓ **Micrometer Carriage Stops:** The micrometer carriage stop, is used to accurately position the lathe carriage. The micrometer stop is designed- so the carriage can be moved into position against the retractable spindle of the stop and locked into place. A micrometer gage on the stop enables carriage movement of as little as **0.001 inch**. This tool is very useful when facing work to length, turning a shoulder, or cutting an accurate groove.




- ✓ **Alignment of Centers:** The most accurate method of checking alignment of centers is by mounting the workpiece between centers and taking light cuts at both ends without changing the carriage adjustments. Measure each end of this cut with calipers or a micrometer. If the tailstock end is greater in diameter than the headstock end, the tailstock is moved toward the operator.





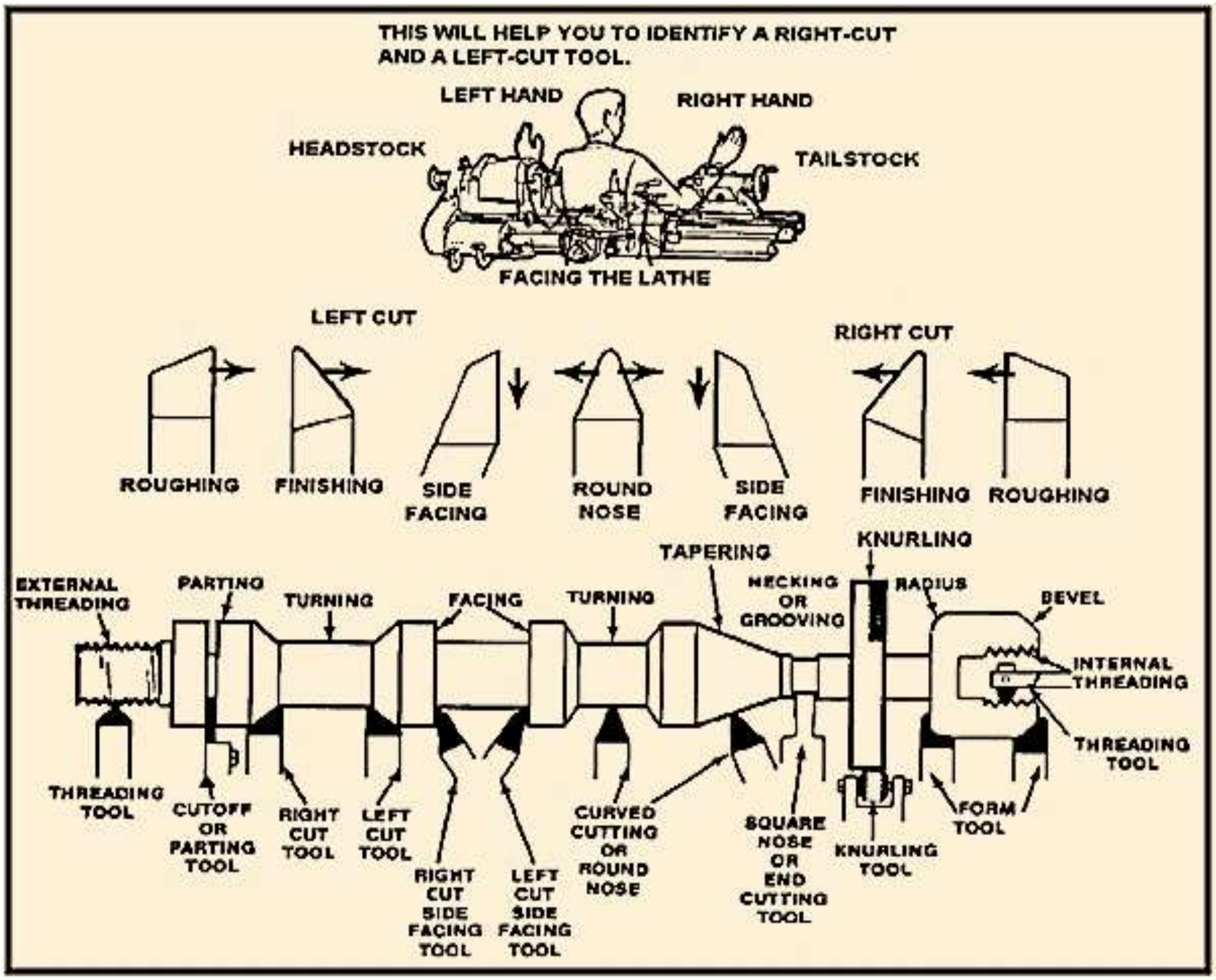
Lathe Cutting Tools:

The lathe cutting tools or tool bits must be made of the correct material and ground to the correct angles to machine a workpiece efficiently. The most common tool bit is the general all-purpose bit made of high-speed steel and are generally inexpensive, easy to grind on a bench or pedestal grinder, take lots of abuse and wear, and are strong enough for all-around repair and fabrication.

High-speed Steel Tool Bits: Made with hard carbon or alloy steel and should bear the high heat generated during cutting and not changed after cooling. These tool bits **are used for** turning, facing, boring and other lathe operations.

Tool bits made from special materials such as carbides, ceramics, diamonds, cast alloys are able to machine workplaces at very high speeds but are brittle and **expensive for normal lathe work**. High-speed steel tool bits are available in many shapes and sizes to accommodate any lathe operation.

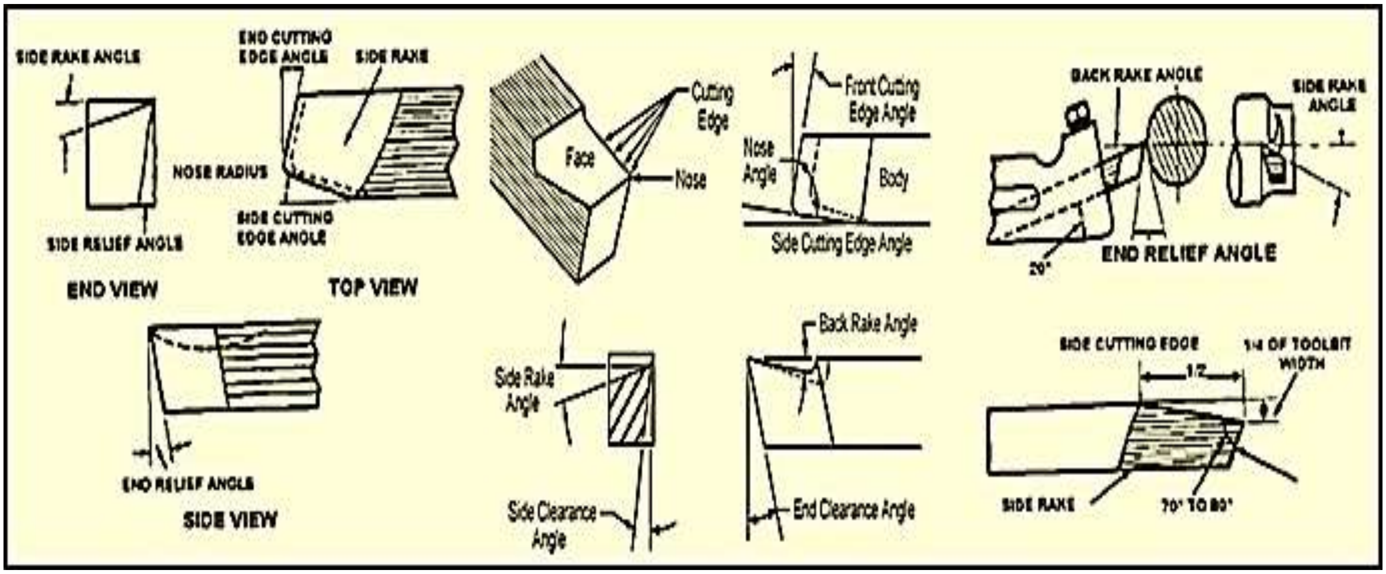
Shapes of Tool Bits: A right-hand turning tool bit is shaped to be fed from right to left, intended for facing on right-hand side shoulders and the right end of a workpiece. The cutting edge is on the left side of the tool bit and the face slopes down away from the cutting edge. The left side and end of the tool bit are ground with sufficient clearance to permit the cutting edge to bear upon the workpiece without the heel rubbing on the work. The right-hand turning tool bit is ideal for taking light roughing cuts as well as general all- around machining.

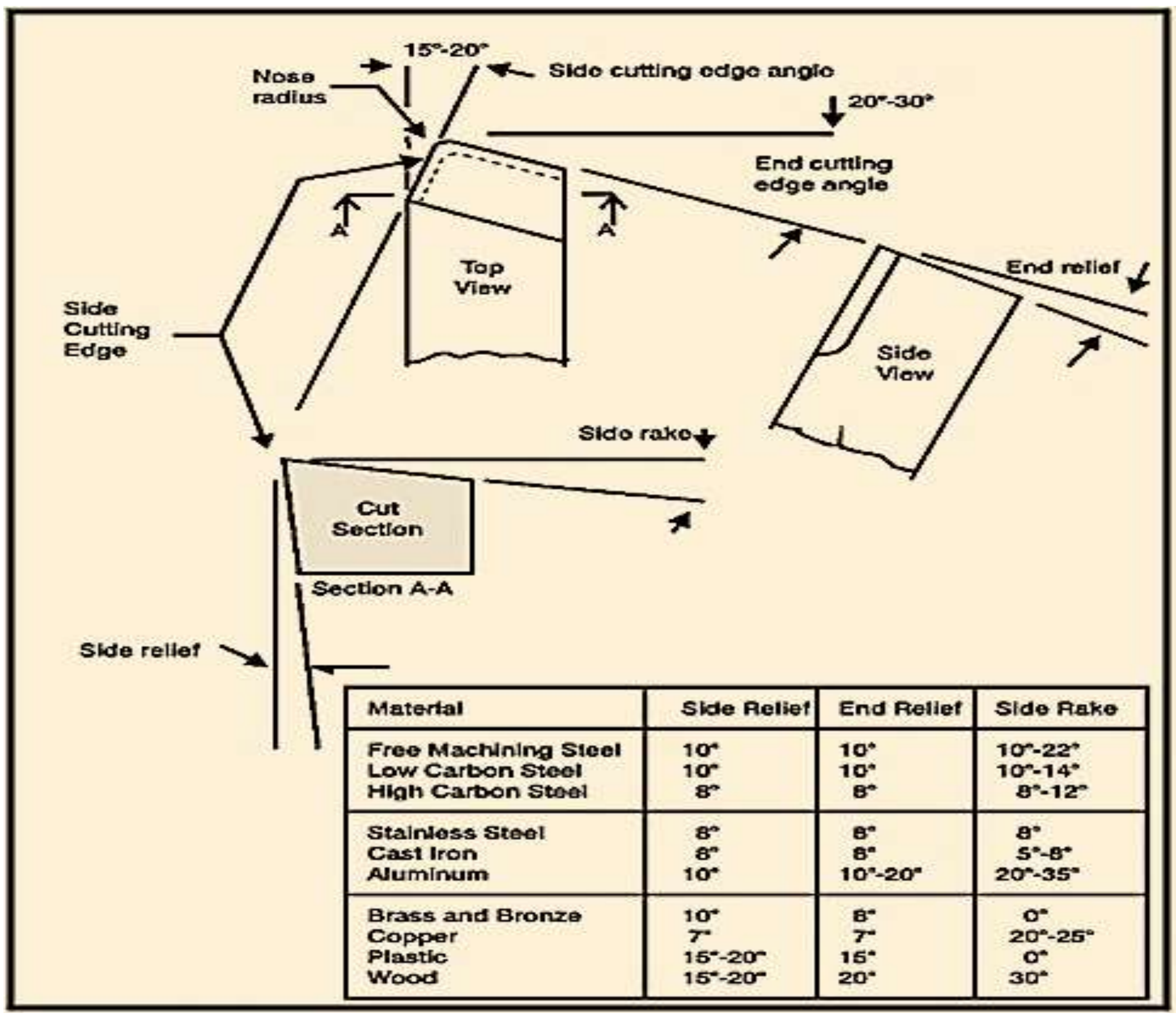


Single Point Tool Bits: Can be one end of a high-speed steel tool bit, one edge of a carbide/ceramic cutting tool or an insert. Basically, a single point cutter bit is a tool that has only one cutting action proceeding at a time:

- The shank is the main body of the tool bit.
- The nose is the part of the tool bit which is shaped to a point and forms the corner between the side cutting edge and the end cutting edge, or the rounded end of the tool bit.
- The face is the top surface of the tool bit upon which the chips slide as they separate from the work piece.
- The side or flank of the tool bit is the surface just below and adjacent to the cuffing edge.
- The cutting edge is the part of the tool bit that actually cuts into the workpiece, located behind the nose and adjacent to the side and face.
- The base is the bottom surface of the tool bit, which usually is ground flat during tool bit manufacturing.
- The end of the tool bit is the near-vertical surface which, with the side of the bit, forms the profile of the bit. The end is the trailing surface of the tool bit when cuffing.
- The heel is the portion of the tool bit base immediately below and supporting the face.

Tools Operation: The machine operator should know the various terms applied to the single point tool bit to properly identify and grind different tool bits, as shown below:

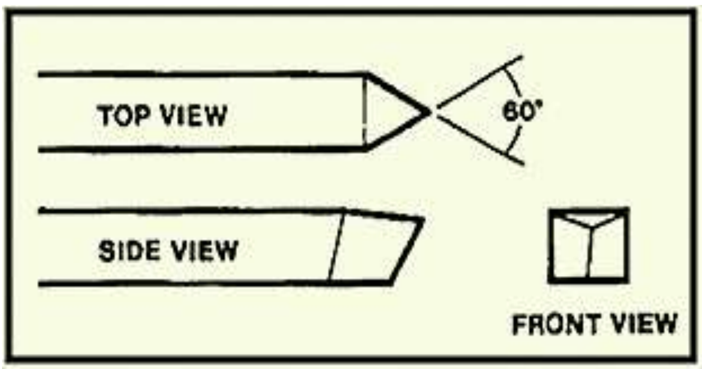


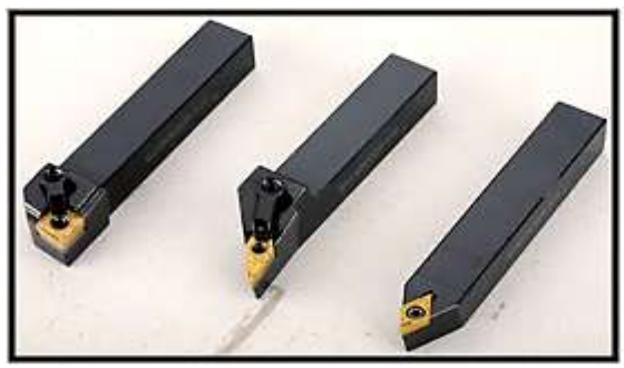


Material	Side Relief	End Relief	Side Rake
Free Machining Steel	10°	10°	10°-22°
Low Carbon Steel	10°	10°	10°-14°
High Carbon Steel	8°	8°	8°-12°
Stainless Steel	8°	8°	8°
Cast Iron	8°	8°	5°-8°
Aluminum	10°	10°-20°	20°-35°
Brass and Bronze	10°	8°	0°
Copper	7°	7°	20°-25°
Plastic	15°-20°	15°	0°
Wood	15°-20°	20°	30°

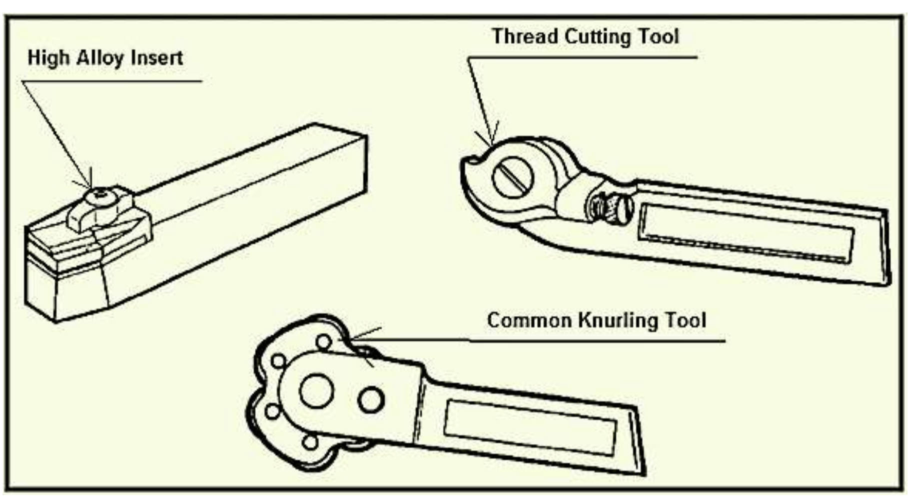
Parting Tool Bits: Are also known as cutoff tool bits. This tool bit has the principal cutting edge at the squared end of the bit that is advanced at a right angle into the workpiece. Both sides should have sufficient clearance to prevent binding and should be ground slightly narrower at the back than at the cutting edge. Besides being used for parting operations, this tool bit can be used to machine square corners and grooves.

Thread-Cutting Tool Bits: Are ground to cut the type and style of desired threads. Side and front clearances must be ground, plus the special point shape for the type of thread desired. Thread-cutting tool bits can be ground for standard 60° thread forms or for square, Acme, or special threads.

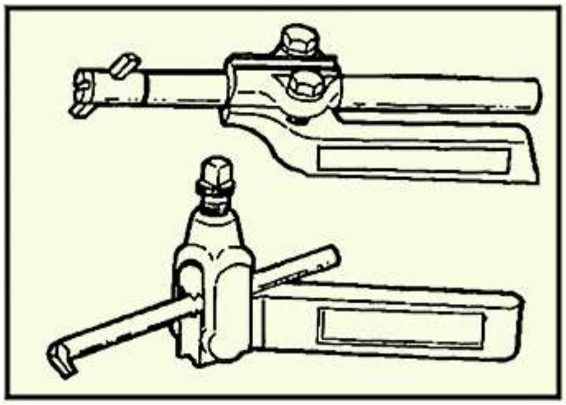


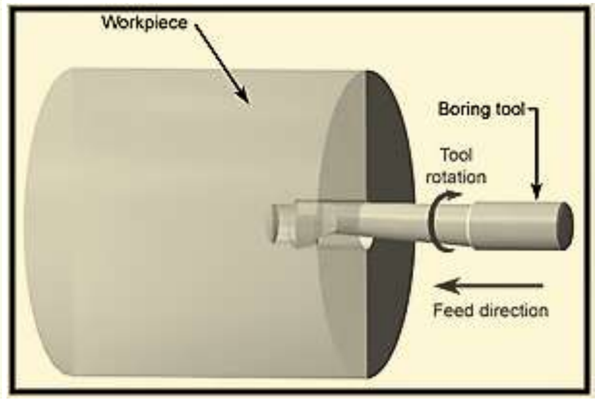


Special Types of Cutting Tools: Are shaped tool bits, as special lathe operations and heavy production work require special types of **cutting tools inserts**, such as, **tungsten carbide, tantalum carbide, titanium carbide, ceramic, oxide, and diamond-tipped tool bits**. These cutting tool inserts are commonly used in high-speed production work when heavy cuts are necessary and where exceptionally hard and tough materials are encountered. Carbide and ceramic inserts can be square, triangular, round, or other shapes. The inserts are designed to be indexed or rotated as each cutting edge gets dull and then discarded. Cutting tool inserts are not intended for reuse after sharpening.



Boring Tool Bits: Are similar to thread-cutting tool bits, but with more end clearance angle to prevent the heel of the tool bit from rubbing against the surface of the bored hole. The boring tool bit is usually clamped to a boring tool holder, but it can be a one-piece unit . The boring tool bit and tool holder clamp into the lathe tool post.





Turning & Single-Point Cutting Tools: Nearly all turning processes use single point cutting tools, this is, tools that cut with only a single edge in contact with the work. Most turning is done with coated indexable carbide inserts, but the tool material may also be high-speed steel, brazed carbide, ceramic, cubic boron nitride, or polycrystalline diamond. When turning with inserts, 75 percent of turning operations use just a few basic tool geometries, built into the tool holder itself rather than the actual insert. The geometry of an insert includes:

- Insert's basic shape;
- Relief or clearance angle;
- Insert type;
- Insert's inscribed circle or "IC" size;
- Insert's nose radius;
- Insert's chip breaker design.

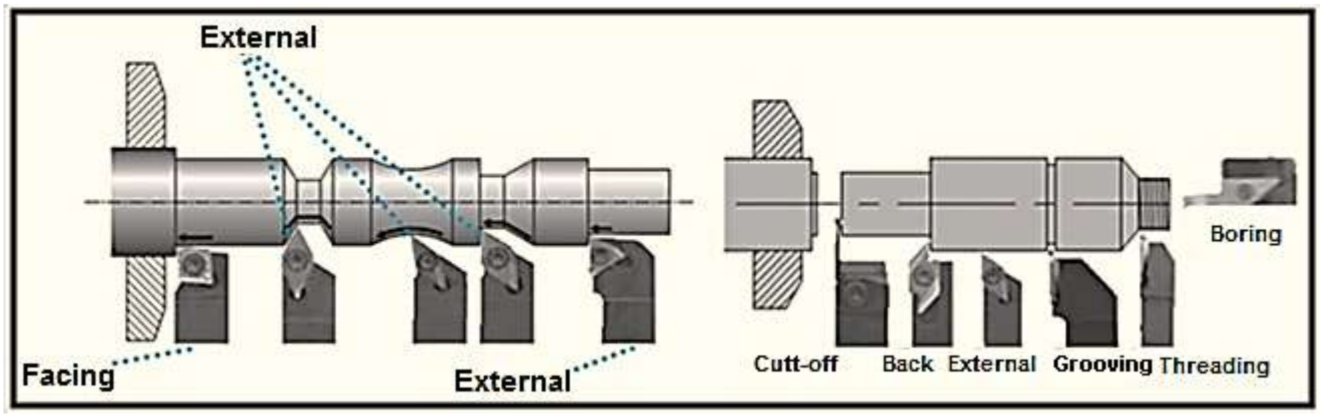
In turning, insert shape selection is based on the trade-off between strength and versatility. For example, larger point angles are stronger, such as round inserts for contouring and square inserts for roughing and finishing. The smaller angles **(35° and 55°)** are the most versatile for intricate work, but turning inserts may be molded or ground to their working shape. The molded types are more economical and have wide application. Ground inserts are needed for maximum accuracy and to produce well defined or sharp contours.

Tool Holders and Inserts: Have been designed to provide optimum performance for large production in different applications and usually over a broad area. The type of operation and, to some extent, size of workpiece determines the selection of tool holding system. Roughing operations on large workpieces make considerably different demands to that of finishing of small components. The selection of the clamping system should be taken from manufacturers tables. It is impossible to pin-point every type of application, especially as the systems overlap at some stage, however, a general purpose tool holder is commonly indicated for each machining application.

Cutting Tool Geometries: Several angles are important when introducing the cutting tool's edge into a rotating workpiece. These angles include:

- Angle of inclination;
- Rake angle;
- Effective rake angle;
- Lead or entry angle;
- Tool nose radius.

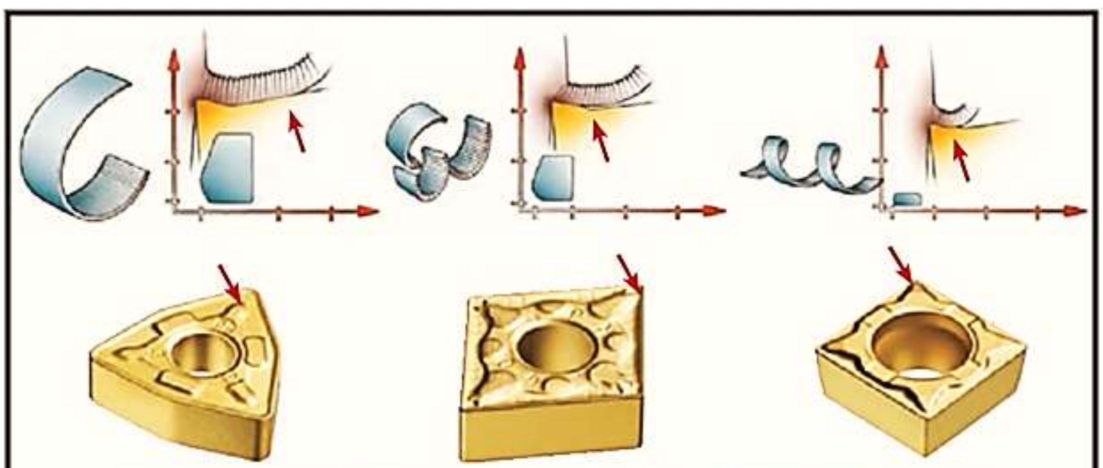
The angle of inclination when viewed from the side or front is the angle of the insert seat or pocket in the tool holder, from front to back. This inclination can be positive, negative, or neutral. The cutting tool's rake angle is the angle between the cutting edge and the cut itself. It may also be positive, negative, or neutral as shown below:



In turning, the chip breaking is critical to efficient work processing and good finishing qualities. Proper chip breaking results from balancing the depth of the cut and the geometry of the tool. Many inserts have chip breaker grooves molded into them. The four basic chip styles generated in turning are:

- Small **"6"** and **"9"** chips;
- Helical or spiral chips;
- Long, stringy chips;
- Corrugated chips.

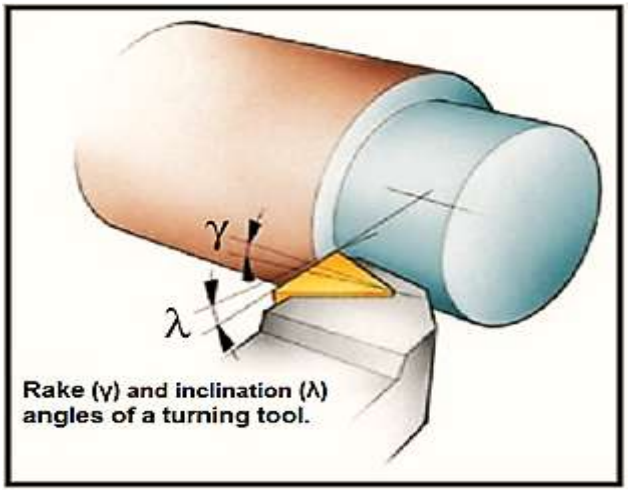
Note: The first type, shaped like the numerals **"6" or "9"**, represents the **ideal chip**. The other types indicate the need for speed and feed adjustments, or selection of a different chip breaker design.


Chipbreakers, as part of the insert geometry, are designed to work at different feeds/depths of cut areas.

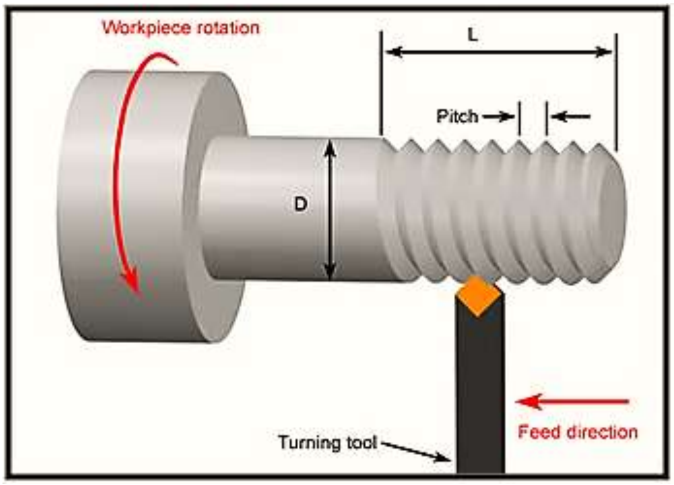
Guidelines: Two times the feed rate = same surface finishing; same feed rate = Twice as good a surface finishing.

Cutting Edge Geometry: A **negative** insert has a wedge angle of 90 degrees seen in a cross-section of the basic shape of the cutting edge. A **positive** insert has an angle of less than 90 degrees. The negative insert has to be inclined negatively in the tool holder, to provide a clearance angle tangential to the workpiece while the positive insert has this clearance built-in.

The **inclination angle (λ)** is a measure of at what angle the insert is mounted in the toolholder. The **rake angle (γ)** is a measure of the edge in relation to the cut although it is often expressed through a flat insert. The rake angle of the insert itself is **usually positive** and varies along the cutting edge, from the nose radius along the straight cutting edge. A flat insert has a **rake angle of zero degrees**. The actual cutting function of the rake angle also varies along the face of the insert, back from the cutting edge, until the chip breaking function takes over the chip formation.

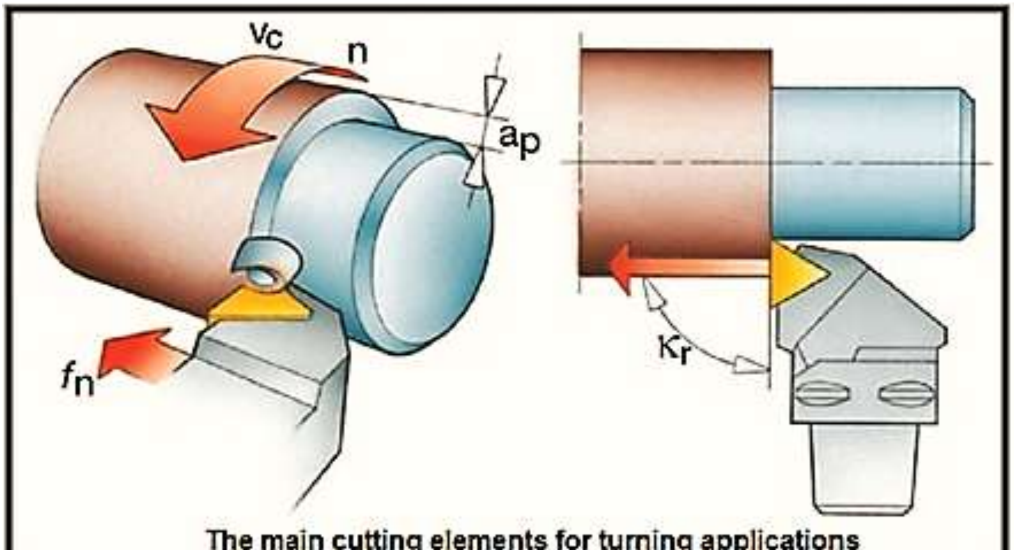


Rake (γ) and inclination (λ) angles of a turning tool.



Cutting Data: The workpiece rotates in the lathe, with a certain spindle speed, at a certain number of **revolutions per minute (n)**. In relation to the diameter of the workpiece, at the point it is being machined, this will give rise to a **cutting speed (Vc)**, (rotational or peripheral speed) in **ft/min or m/min**. The **cutting feed (*f*n)** in **ft/rev** or **mm/rev** is the movement of the tool in relation to the revolving workpiece. This is a key value in determining the quality machined and for ensuring that the chip formation is within the scope of the tool geometry. This value influences, not only how thick the chip is, but also how the chip forms against the insert geometry.

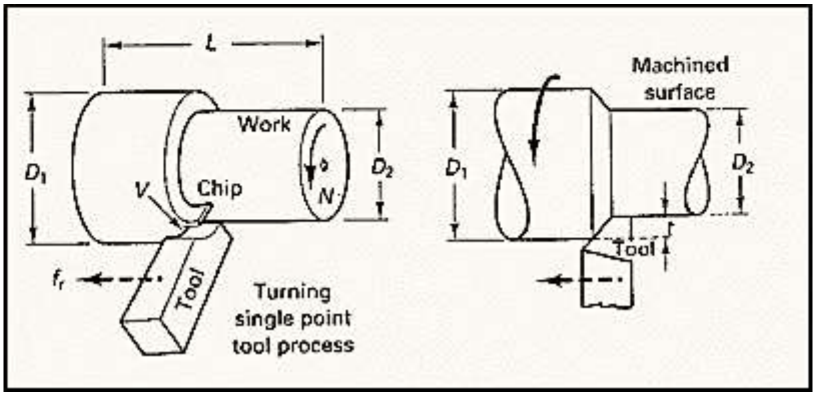
The **cutting depth** in **ft or mm** is the difference between un-cut and cut surface. It is half of the difference between the un-cut and cut diameter of the workpiece. The **cutting depth** is always measured at right angles to the feed direction of the tool. The cutting edge approach to the workpiece is expressed through the **entering angle (κr)** that affects the factors, as the direction of forces involved, the length of cutting edge and the way in which the cutting edge makes contact with the workpiece.

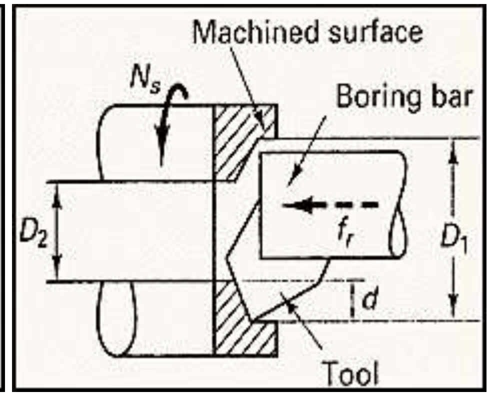


The main cutting elements for turning applications

Lathe Calculations:

Before any operation can be done, a thorough knowledge of the variable factors of **lathe speeds, cutting feeds and depth of cut** must be understood. These factors differ for each lathe operation, and failure to use these factors properly will result in machine failure or work damage. The kind of material being worked, the type of tool bit, the diameter and length of the workpiece, the type of cut desired (roughing or finishing), and the working condition of the lathe will determine which **speed, feed, or depth of cut** is best for any particular operation. The guidelines are general in nature and may need to be changed according to specific applications.





Cutting Operations: The cutting speed of a tool bit is defined as the number of feet of workpiece surface, measured at the circumference that passes the tool bit in one minute. The **feed rate (fR)** is expressed in ft/rev and the **cutting speed**, expressed in FPM (ft/min) and must not be confused with the **spindle speed** (spindle rotation, N) of the lathe which is expressed in RPM. To obtain uniform cutting speed, the lathe spindle must be rotated faster for workplaces of small diameter and slower for workplaces of large diameter.

The proper cutting speed for a given job depends upon the hardness of the material being machined, the material of the tool bit, and how much feed and depth of cut is required. **Cutting speeds** or peripheral velocity (V, Vc) or **surface feet per minute** (SFM) are usually expressed in **ft/min**, measured on the circumference of the work. **Spindle revolutions per minute** (RPM) are determined by using the formula:

1) **Spindle speed, spindle rotation, N (RPM) =**

$$\frac{\textbf{Cutting Speed, Cs (ft/min) x 12}}{\pi \times D}$$

2) **Cutting Speed, V, Vc, Cs, SFM, (ft/min) =**

$$\frac{\pi \times \textbf{D (in) x N (Spindle speed, RPM)}}{12}$$

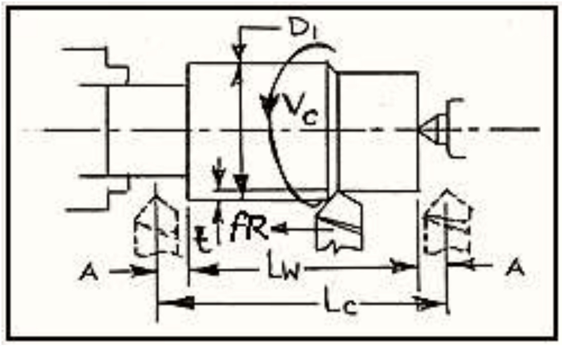
This can be closely approximated as:

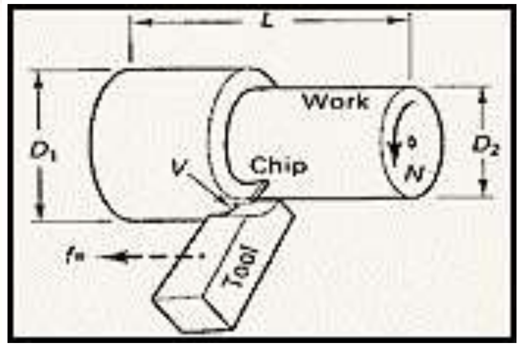
3) **Spindle speed, spindle rotation, N (RPM) =**

$$\frac{Cs \times 4}{D} \quad \text{- (or - } RPM = \frac{CS \times 320}{D}\text{)}$$

Cs (V, Vc, CS or SFM) = **cutting speed** or rated **surface feet per minute** or (ft/min) (in/min) (m/min) (mm/min);
RPM, N = spindle speed in revolutions per minute;
D (D1,or d) = diameter of the work in inches (mm).

Example: To use the formula, simply insert the cutting speed of the metal and the diameter of the workpiece into the formula and you will have the RPM. Turning a **1/2 in** piece of aluminum with a **cutting speed** of **200 ft/min,** would result in the following:





Spindle speed or spindle rotation N (RPM) =

$$\frac{200 \times 4}{1/2''} = 1600 \text{ RPM}$$

Materials machined dry without coolant, require a slower cutting speed. When **carbide-tipped tool bits** are being used, speeds can be increased **two to three times** the speed used for high speed tool bits. (See manufacturers' tables).

Feed rate: Is the term applied to the distance the tool bit advances along the work for each revolution of the lathe spindle, measured in **inches or millimeters per revolution**.

4) **Feed rate, f_R (in/rev) =** $\dfrac{\text{Cutting speed, Cs (in/min)}}{\text{Spindle rotation, N (RPM)}}$

5) **Cutting speed Cs, Vc, (in/min) = Spindle speed, N (RPM) x Cutting feed, f_R (in/rev)**;

6) **Cutting speed, Cs, Vc (ft/min) = π. D. N /12 [**Where: π (3.1416); D (spindle diameter, inches); N (RPM)].

Note: Feed rate (fR) is advance of the tool per spindle rotation (in/rev, ft/rev, m/rev, mm/rev), during machining and cannot be confused with **cutting speed** (Vc, Cs) which refers to surface spindle velocity (ft/min, m/min). This results in most of the pressure of the cut being put on the work-holding device. If the cut must be fed toward the tailstock, use less feeds and less cuts to avoid losing the workpiece.

7) **Cutting time, Ct (min) =** $\dfrac{\text{Lw (cut length) (in)}}{\text{N (RPM)} \times f_R \text{ (in/rev)}} = \dfrac{\text{Lw (in)}}{\text{Cs (in/min)}}$

Or:

Ct = $\dfrac{(L + A)}{N * f_R}$ **– or; Ct =** $\dfrac{Lc \times np}{N \times f_R}$ **(see sketch)** →

Ct = Cutting time per "pass" [min];
Lw = Length cut [in] [mm] – **L**, **Lc** = Lw + 2A [in] [mm];
A = "Allowance" or starting offset, (2~5mm) – (0.062~0.18 in);
f_R = Feedrate per revolution of spindle, [in/rev] – [mm/rev];
np = Number of passes required.

Depth of Cut: Depth of cut is the distance that the tool bit moves into the work measured in **thousandths of an inch or in millimeters**. (Be careful with conversions feet, inches, meters or mm).

Note: General practice is to use a **depth of cut up to 5 times the feed rate**. Then, cutting a stainless steel piece using a **feed per revolution of 0.020 (inch/rev)**, will take a **depth of cut of 0.100 in**. (Take care about conversions).

8) **Depth of Cut, Dc (in) = 5 x f_R (in/rev) =**

Material Removal Rate (MRR):

9) $$MRR = \frac{\text{Volume Removed}}{\text{Cutting time}}\ (in^3/min)\ (mm^3/min) = \frac{\dfrac{\pi \times L \times (D_1^2 - D2^2)}{4 \times L}}{N \times f_R} = 0.7854\,(D_1^2 - D2^2) \times N \times fR.$$

Cutting speeds (Vc, Cs) references for turning and threading with HSS tool bits:

MATERIAL	STRAIGHT TURNING SPEED		THREADING SPEED	
	FEET PER MINUTE	METERS PER MINUTE	FEET PER MINUTE	METERS PER MINUTE
LOW-CARBON STEEL	80-100	24.4-30.5	35-40	10.7-12.2
MEDIUM-CARBON STEEL	60-80	18.3-24.4	25-30	7.6-9.1
HIGH-CARBON STEEL	35-40	10.7-12.2	15-20	4.6-6.1
STAINLESS STEEL	40-50	12.2-15.2	15-20	4.6-6.1
ALUMINUM AND ITS ALLOYS	200-300	61.0-91.4	50-60	15.2-18.3
ORDINARY BRASS AND BRONZE	100-200	30.5-61.0	40-50	12.2-15.2
HIGH-TENSILE BRONZE	40-60	12.2-18.3	20-25	6.1-7.6
CAST IRON	50-80	15.2-24.4	20-25	6.1-7.6
COPPER	60-80	18.3-24.4	20-25	6.1-7.6

NOTE: Speeds for carbide-tipped bits can be 2 to 3 times the speed recommended for high-speed steel

Suggested high-speed steel feed per revolution of spindle (fR) (tool - HSS):

	Rough Cuts		Finish Cuts	
Material	in/rev	mm/rev	in/rev	mm/rev
Carbon steel	0.010–0.020	0.25–0.5	0.003–0.010	0.075–0.25
Tool steel	0.010–0.020	0.25–0.5	0.003–0.010	0.075–0.25
Cast iron	0.015–0.025	0.4–0.65	0.005–0.012	0.125–0.30
Bronze	0.015–0.025	0.4–0.65	0.003–0.010	0.075–0.25
Aluminum	0.015–0.030	0.4–0.75	0.005–0.010	0.125–0.25

Suggested lathe-cutting speed (Vc, Cs) - Using high-speed steel (HSS) Tool bits:

	Rough Cuts		Finish Cuts		Threading	
Material	**ft/min**	**m/min**	**ft/min**	**m/min**	**ft/min**	**m/min**
Carbon steel	90	27	100	30	35	11
Tool steel	70	21	90	27	30	9
Cast iron	60	18	80	24	25	8
Bronze	27	100	30	25	25	8
Aluminum	200	61	300	93	60	18

Suggested cutting speed (Vc, Cs, m/min) & feed per revolution (fR, mm/rev):

Material	Condition hardness (HB)	HSS Speed m/min	HSS Feed (mm/rev) 12mm	25mm	50mm	Carbide Speed m/min	Carbide Feed (mm/rev) 12mm	25mm	50mm
WROUGHT MARTENSITIC									
Free machining grades	160	33	0.08	0.06	0.05	115	0.125	0.10	0.09
Lower C/lower Cr grades	175	26	0.08	0.06	0.05	105	0.125	0.10	0.09
Lower C/lower Cr grades	300	14	0.05	0.04	0.025	55	0.09	0.06	0.05
Higher C/higher Cr grades	240	14	0.08	0.06	0.04	65	0.125	0.10	0.09
Higher C/higher Cr grades	320	12	0.04	0.025	0.02	50	0.08	0.05	0.04
WROUGHT FERRITIC									
Free machining grades	160	33	0.08	0.06	0.05	115	0.125	0.10	0.09
12 - 17% Cr grades	160	26	0.08	0.06	0.05	100	0.125	0.10	0.09
WROUGHT AUSTENITIC									
Free maching grades	160	21	0.08	0.06	0.05	90	0.150	0.10	0.09
Other grades (304, 316, 321 etc)	160	17	0.08	0.06	0.05	60	0.125	0.10	0.08
WROUGHT DUPLEX	230	17	0.08	0.06	0.05	50	0.100	0.08	0.06
CAST PLAIN Cr	210	15	0.04	0.025	0.02	62	0.08	0.06	0.04
CAST AUSTENITIC	170	11	0.04	0.025	0.02	52	0.08	0.06	0.04

m/min = surface metres per minute C = carbon Cr = chromium
Ann = Annealed Q&T = Quenched & tempered N&T = Normalised & tempered

Table 1. Recommended values for precision boring/turning

Workpiece material	Hardness [Bhn]	Cutting speed - High speed steel Vc [feet/min] MIN	MAX	Vc [m/min] MIN	MAX	Cutting speed - Carbide uncoated Vc [feet/min] MIN	MAX	Vc [m/min] MIN	MAX	Feed rate per revolution f [inch] MIN	MAX	f [mm] MIN	MAX
Cast irons	190...320	16	197	5	60	33	492	10	150	0,003	0,020	0,080	0,500
Steel - plain carbon	85...200	49	394	15	120	197	919	60	280	0,003	0,020	0,080	0,500
Steel - alloys	35...50Rc	16	131	5	40	66	492	20	150	0,003	0,020	0,080	0,500
Steel - tool	50...58Rc	16	66	5	20	49	197	15	60	0,003	0,020	0,080	0,500
Steel - stainless	150...450	16	98	5	30	98	394	30	120	0,003	0,020	0,080	0,500
Aluminum alloys	30...150	492	1181	150	360	492	2625	150	800	0,003	0,020	0,080	0,500
Copper alloys	80...100Rb	98	591	30	180	164	1378	50	420	0,003	0,020	0,080	0,500
Nickel alloys	80...360	16	131	5	40	16	394	5	120	0,003	0,020	0,080	0,500
Titanium	250...375	16	98	5	30	33	328	10	100	0,003	0,020	0,080	0,500

Example: Considering the tables above, calculate the average time required to machine a **2.0 in**. diameter, machine-steel shaft **16.0 in**. length, to final **1.85 in**. diameter finish size:

Roughing cut =

$$\text{RPM} = \frac{\text{CS x 4}}{\text{D}} = \frac{90 \text{ x } 4}{2} = 180 - (Cs = 90.table)$$

Roughing feed – feed rate (fR) = 0.020 in/rev (table):

$$\text{Average roughing cut time} = \frac{\text{length of cut}}{\text{feed (fR) x r/min}} = \frac{16}{0.020 \text{ x } 180} = 4.4 \text{ min}$$

Finishing cut:

$$RPM = \frac{100 \text{ x } 4}{1.850} = 216.rpm - (Cs = 100.table)$$

Finishing feed = feed rate (fR) = 0.003 in/rev (table):

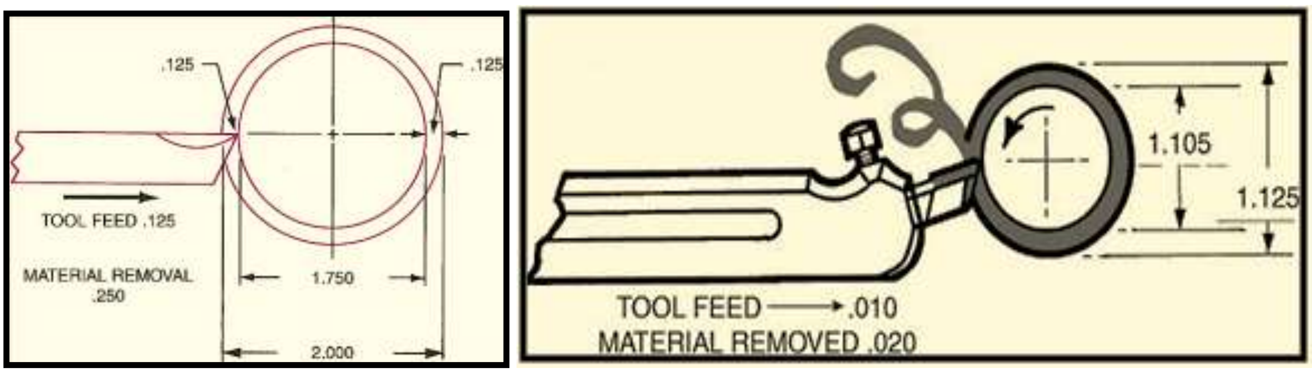
$$\text{Average finishing cut time} = \frac{16}{0.003 \text{ x } 216} = 24.7 \text{ min}$$

Cutting Speeds: Is given in feet per minute (ft/min), in. per minute (in/min) meters per minute (m/min) or (mm/min). Important to use the correct speed for cutting material.

Spindle Rotation (metric):

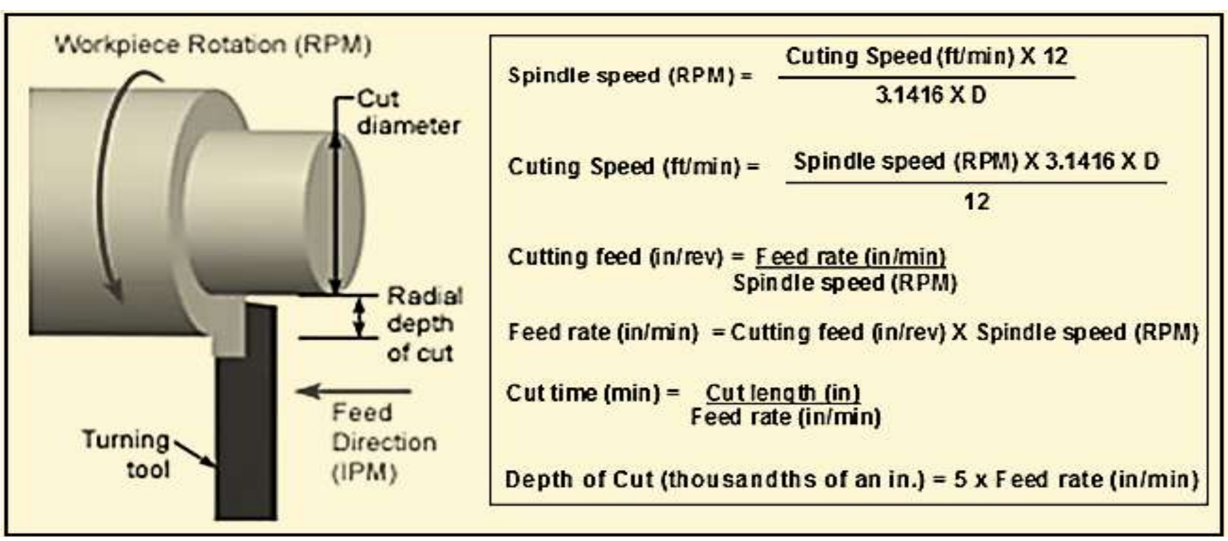
N = $\frac{Cs \times 320}{D}$ **=** Where, D (mm) and Cs (see recommended tables)

Depth of Cut: Depth of chip taken by cutting tool and one-half total amount removed from workpiece in one cut. Only one roughing and one finishing cut, **Roughing** cut should be deep as possible to reduce diameter to within **0.030 to 0.040 inch** (0.75 to 1.0 mm) of size required. **Finishing** cut should not be less than **0.005 inch** (0.125 mm).



Example of depth of cut dimensions.

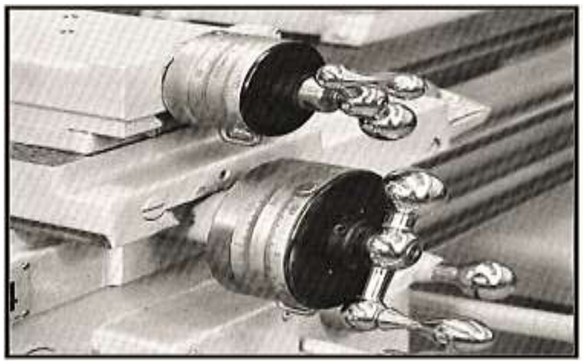
Turning calculations sketch:



Graduated Micrometer Measurements: Used when diameter of work must be turned to accurate size. Sleeves or bushings mounted on compound rest and crossfeed screws.

Measurements:

- Inch system graduated in **0.001 inch;**
- Metric system usually in steps of **0.02 mm.**



Measurements in Imperial System: Circumference of crossfeed and compound rest screw collars divided into **100-125** equal divisions:

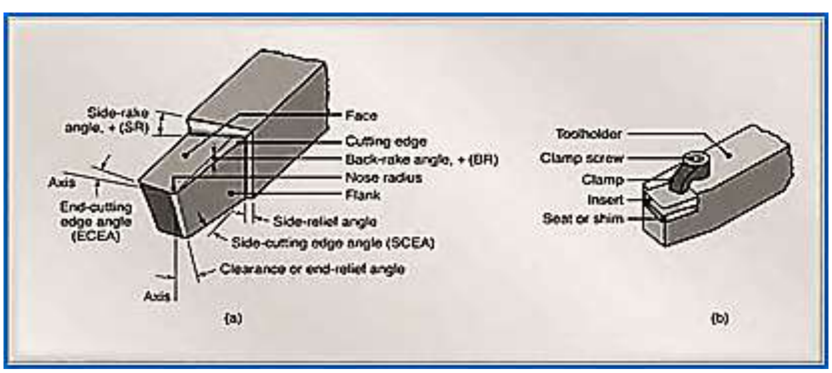
- Each has value of 0.001 inch:
  - Turn crossfeed screw clockwise 10 graduations, cutting tool moved **0.010 inch** toward the workpiece;
  - Lathe revolves, so **0.010** depth of cut taken from entire work circumference reducing diameter **0.020 inch**.

Measurements in Metric System: Circumference of crossfeed and compound rest screw collars divided into **100-125** equal divisions

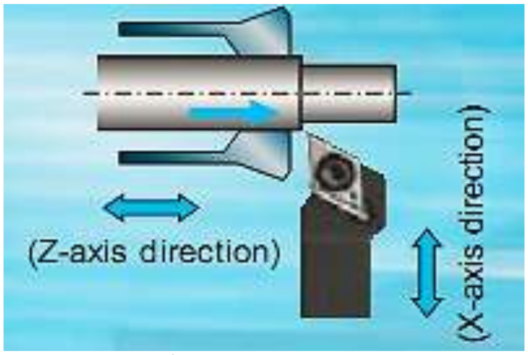
- Each has value of **0.02** mm.
- Turn crossfeed screw clockwise **10** graduations, cutting tool moved **0.2 mm** toward the workpiece;
- Lathe revolves, so **0.2 mm** depth taken from entire work circumference reducing diameter **0.4 mm**.

Tools and Inserts:

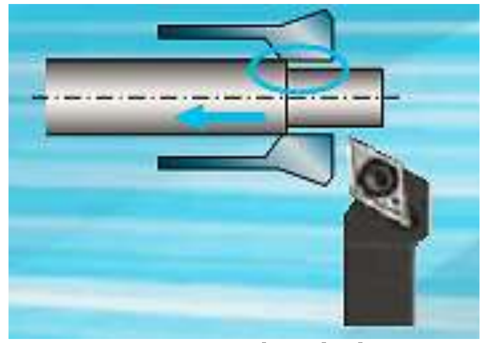
Although these tools traditionally have been produced from ***solid tool-steel bars***, in carbon or alloy steel, these tools have been replaced largely and more productive, with inserts made of carbides and other materials of various shapes and sizes as shown below:



Common holders with inserts:

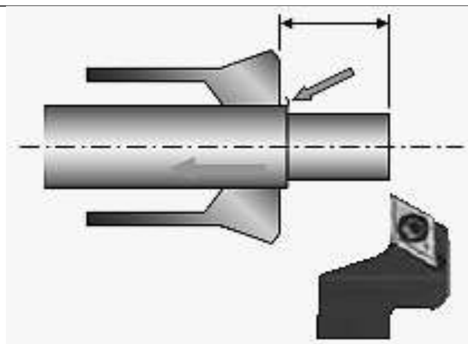


The tool position is fixed and the bar material (workpiece) moves longitudinally.

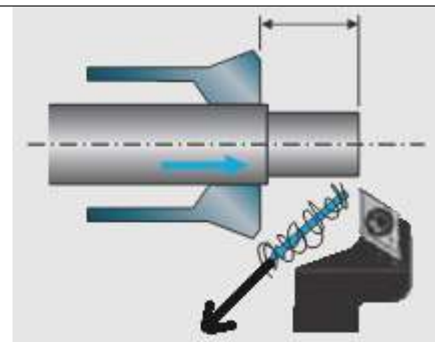


Problem: At step turning with multiple passes, the machined part returns into the guide bush and may cause dimensional variation.

Goose-neck holders with inserts:



Solution 1: The workpiece burr does not contact the guide bush with no breakage.



Solution 2: Large space for chip evacuation. Better and smooth chip control.

Drilling Machines:

Drilling machines or drill presses are one of the most common machines found in the machine shop. A drill press is a machine that turns and advances a rotary tool into a workpiece. The drill press is used primarily for drilling holes, but when used with the proper tooling, it can be used for a number of machining operations.

The most common machining operations performed on a drill press are drilling, reaming, tapping, counterboring, countersinking, and spot facing. There are many different types or configurations of drilling machines, but most drilling machines will fall into four broad categories: upright sensitive, upright, radial, and special purpose.

Classification of Drilling Machines:

Table Top Small Drilling Machines: These small capacity (= 0.5 kW) vertical single spindle drilling machines are mounted on rigid table and manually operated using usually small size (f = 10 mm) drills, as shown in Fig.1.

Pillar Drilling Machines: These drilling machines, shown in Fig.2, usually called pillar drills, are quite similar to the tabletop drilling machines but of little larger size and higher capacity (0.55 ~ 1.1 kW) and are grouted on the floor (foundation). The drill-feed and the worktable movement are also done manually. These low cost drilling machines have tall tubular columns and are generally used for small jobs and light drilling.

Fig.1, Fig. 2 and Fig.3

Box Column Drilling Machines: These type-drilling machines (Fig.3) are much more strong, rigid and powerful than the pillar drills. In column drills, the feed gearbox enables automatic and power feed of the rotating drill at different feed rates as desired. Blanks of various size and shape are rigidly clamped on the bed or table or in the vice fitted on that. Such drilling machines are most widely used and over wide range (light to heavy) work.

Radial Drilling Machines: This usually large drilling machine, shown in Fig.4, possesses a radial arm, which along with the drilling head can swing and move vertically up and down as can be seen below. The radial, vertical and swing movement of the drilling head enables locating the drill spindle at any point within a very large space required by large and odd shaped jobs. There are some more versatile radial drilling machines where the drill spindle can be additionally swiveled and / or tilted.

CNC Column Drilling Machines: These versatile and flexibly automatic drilling machines have box column type rigid structure the worktable movements and spindle rotation are programmed and accomplished by Computer Numerical Control (CNC). These modern sophisticated drilling machines are suitable for piece or batch production of precision jobs, as shown in Fig.5.

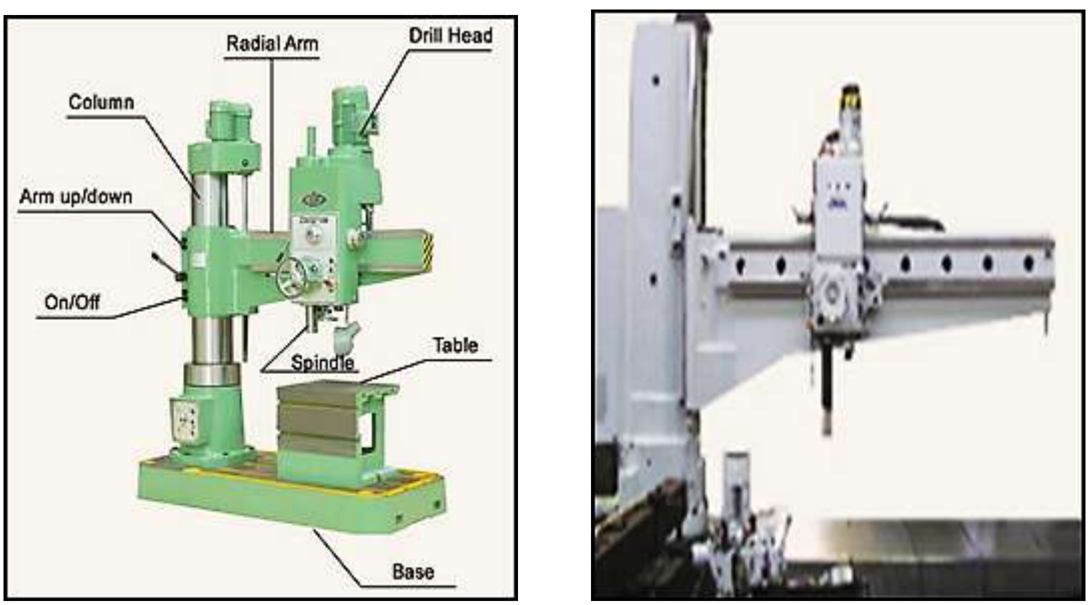


Fig. 4 and Fig. 5

Gang Drilling Machines: Is a single purpose and productive machine which may have 2 to 6 spindles with drills (same or different sizes) in a row, made to produce a number of holes progressively or simultaneously through the jig. Fig. 6 shows a typical gang drilling machine.

Turret Drilling Machines: Are also very productive by having a pentagon or hexagon turret, to bear a number of drills and similar tools, indexed, moving up and down to perform quickly the desired series of operations progressively, as shown below. These drilling machines have structurally rigid columns, and are available with varying degree of automation, both fixed and flexible types, as shown in Fig.7.

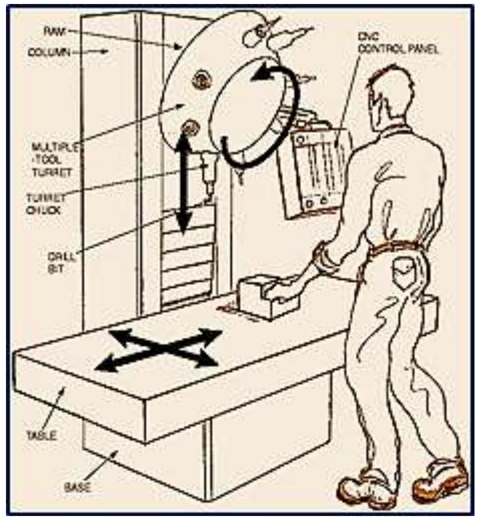


Fig. 6 and Fig. 7

Multispindle Drilling Machines: In these high production machine tools a large number of drills work simultaneously on a blank through a jig specially made for the particular job. (Fig. 8 shows a typical multispindle drilling machine). The entire drilling head works repeatedly using the same jig, for batch or lot of production in a particular job. The rotation of the drills are derived from the main spindle and the central gear through a number of planetary gears in mesh with the central gear) and the corresponding flexible shafts.

Mini Drilling/ Milling Machines: Are capable of machining metal and nonmetallic stock by cutting, drilling, and milling. It can cut circular surfaces, both inside and out, cones, mill planes or grooves, and other cutting functions depending on the tools used. The machine consists of the following main components as shown in Fig. 9.

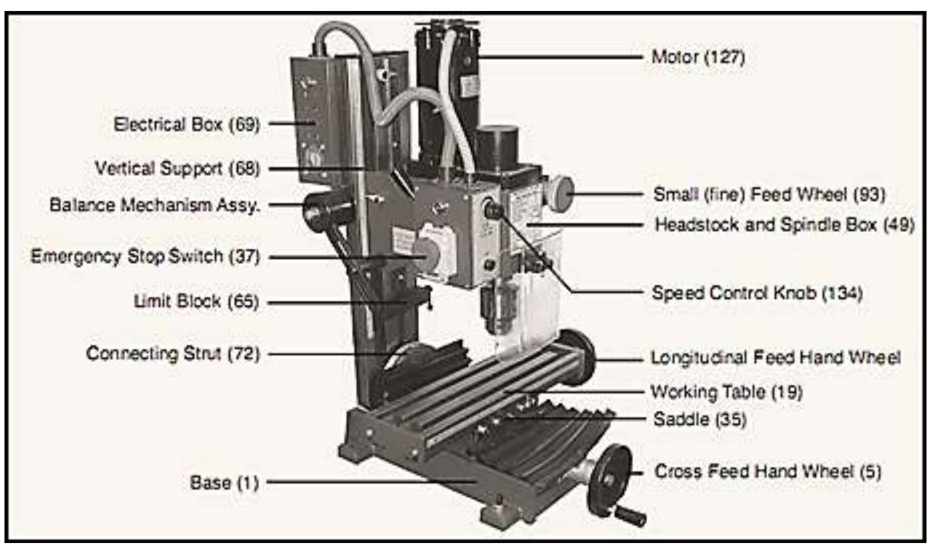


Fig. 8 and Fig. 9

Portable Drilling Machines: Consists of a drive and gearbox, four rigid pillars and a mounting base. The spindle incorporates a standard #5 Morse taper to run tooling such as twist drills and trepanning cutters. The drive motors are easily interchanged, along with the availability of special tooling adapters allowing for the quick and simple selection of the correct tooling and speeds. The spigot sleeve also allows optional drill bushes to be fitted to centralize the drill when starting on radial or uneven faces, as shown in Fig. 10.

Micro Drilling Machines: This type of tiny drilling machine of height within around 200 mm (8 inches) is placed or clamped on a table, as shown in Fig. 11, and operated manually for drilling small holes of around 1 to 3 mm (1/32" to 1/8") diameter in small workpieces.

Hand Drills: The hand drill is a portable drilling device, which is mostly held in hand and used at the locations where holes have to be drilled, as shown in Fig. 12. The small and reasonably light hand drills are run by a high-speed electric motor. In fire hazardous areas the drill is often rotated by compressed air.



Fig. 10, Fig. 11 and Fig. 12.

Pneumatic Drilling Machines: These machine types have two permanent magnetic bases, two pneumatic motors, and #5 Morse taper spindles, make this machine one of the most powerful portable magnetic drills on the market, for very special applications easily assembled, as shown in Fig. 13 and 14. The pneumatic pick is suitable for drilling soft ores in mining, digging poles, feet-holes, or crashing concrete and clay in construction work or working in the mechanical industry, as shown in Fig. 15.


Fig. 13, Fig. 14 and Fig. 15

High-Pressure Drilling Machines: Are used to make connections to pipelines, tanks and plant piping to make 1/2" through 4" completion plugs in tapping nipples to permit recovery of tapping valves in preparation for plugging machine applications, without shutdown. These drilling machines may be manual or power-driven which under high pressure, as shown in Fig. 16.

Hydraulic Drilling Machines: Is perfectly suitable for mass production and second operation drilling single or multi holes after machining in CNC lathes and machining centers. Hydraulic drilling machines (Fig. 17) increases production with consistency and almost 70% reducing the labor cost. Small and big drill bits can be drilled in a shorter time, driven by a hydraulic motor.

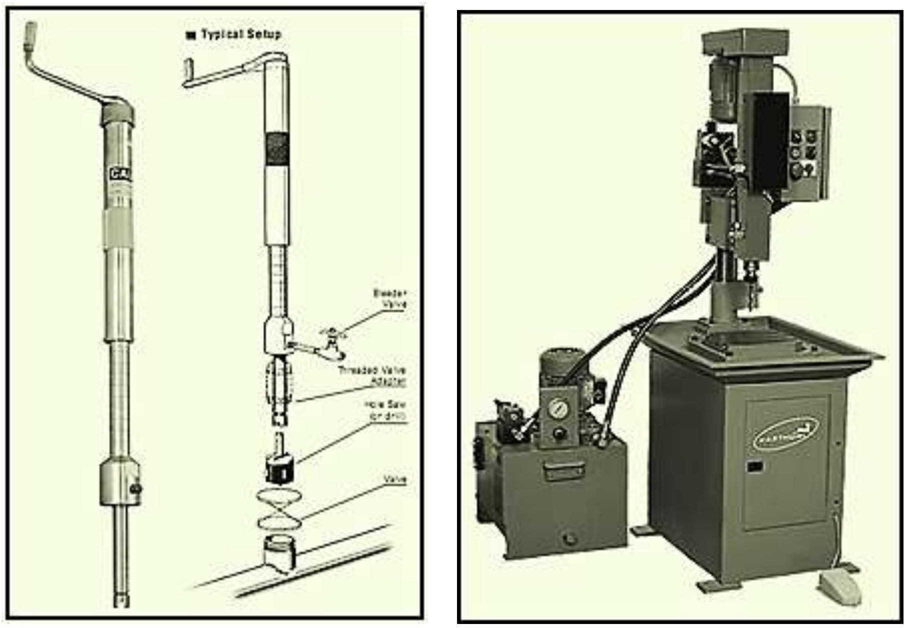


Fig. 16 and Fig. 17

Application of Drilling Machines:

Drilling machines of different capacity and configuration are basically used for originating cylindrical holes and occasionally for enlarging the existing holes to full or partial depth. But different types of drills are suitably used for various applications depending upon work material, tool material, depth and diameter of the holes. The general-purpose drills may be classified as:

• **According to material:**
∆ High-speed steel – most common;
∆ Cemented carbides;
- Without or with coating;
- In the form of brazed, clamped or solid.

• **According to size:**
∆ Large twist drills of diameter around 40 mm (1 1/8");
∆ Micro drills of diameter 25 to 500 µm (0.001 to 0.002 in);
∆ Medium range (most widely used) diameter ranges between 3 mm to 25 mm (1/8" to 1").

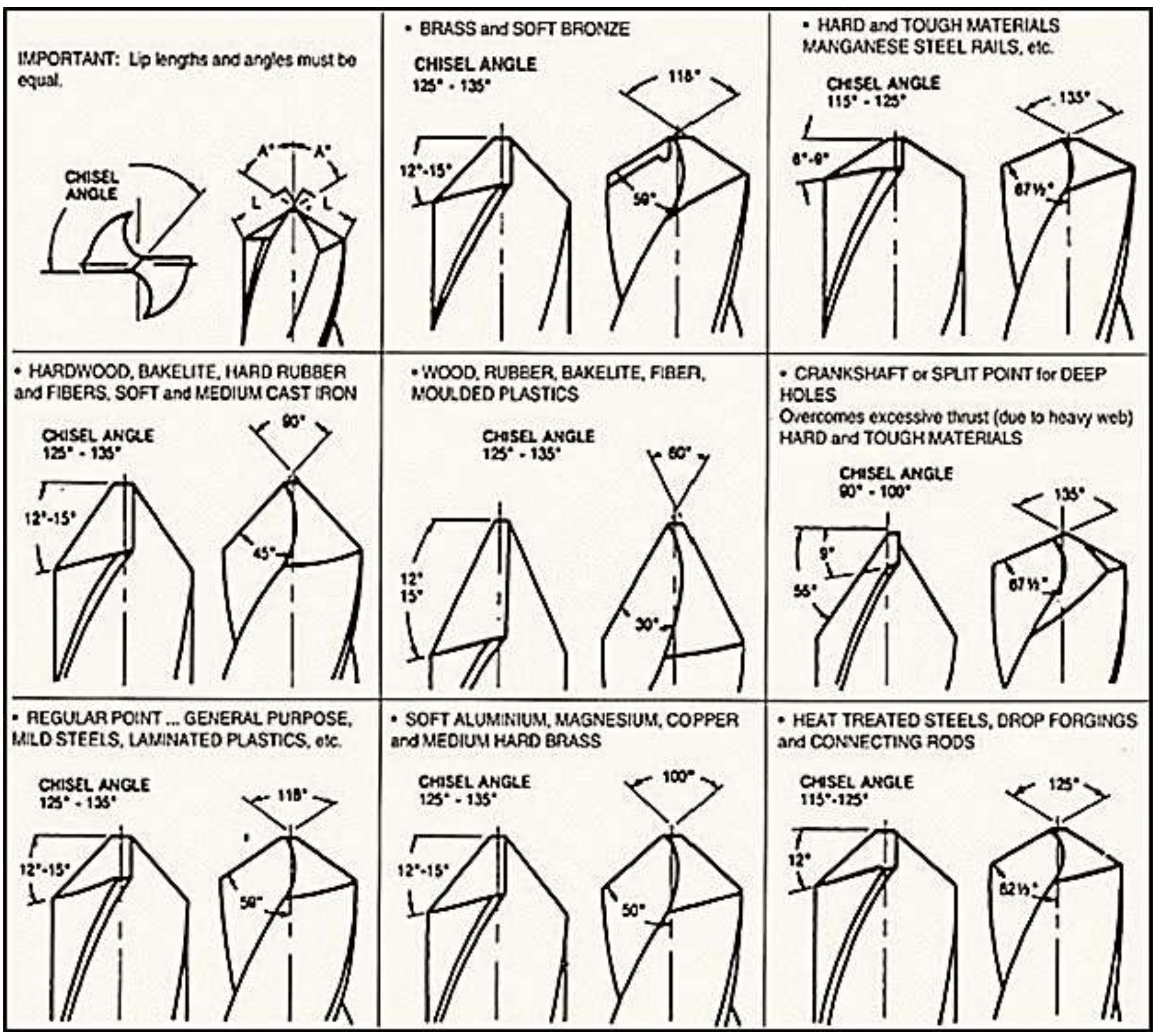
• **According to number of flutes:**
∆ Two fluted – most common;
∆ Single flute – e.g., gun drill (robust);
∆ Three or four flutes – called slot drill;

• **According to helix angle of the flutes:**
∆ Usual – 20° to 35° – most common;
∆ Large helix: 45° to 60° suitable for deep holes and softer work materials;
∆ Small helix: for harder / stronger materials;
∆ Zero helix: spade drills for high production drilling micro-drilling and hard work materials.

• **According to length – to – diameter ratio:**
∆ Deep hole drill; e.g. crank shaft drill, gun drill, etc.
∆ General type: L/f - Ø= 6 to 10 mm (1/4” to 3/8”)
∆ Small length: e.g., center drill.

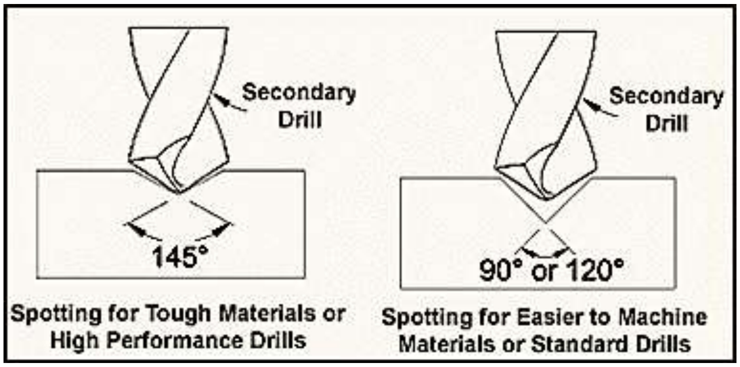
• **According to shank:**
∆ Straight shank – small size drill being held in drill chuck;
∆ Taper shank – medium to large size drills fitted into the spindle nose directly or through taper sockets.

• **According to specific applications:**
∆ Centre drills - for small axial hole with 60° taper end to accommodate lathe center for support;
∆ Step drill and sub land drill - for small holes with two or three steps.

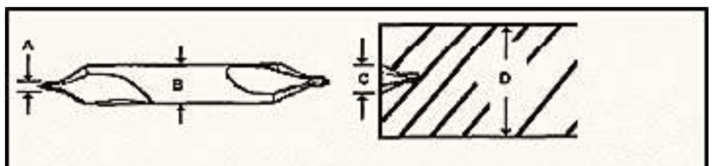


Types of drills:

- **Spot Drills:** The purpose of spot drilling is to drill a hole that will act as a guide for drilling the final hole. The hole is only drilled part way into the workpiece because it is only used to guide the beginning of the next drilling process.

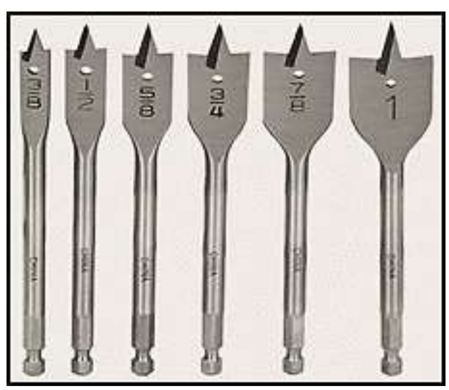


- **Center Drills:** The purpose of center drilling is to drill a hole that will act as a center of rotation for possible following operations. Center drilling is generally performed using a drill with a special shape, known as a center drill.

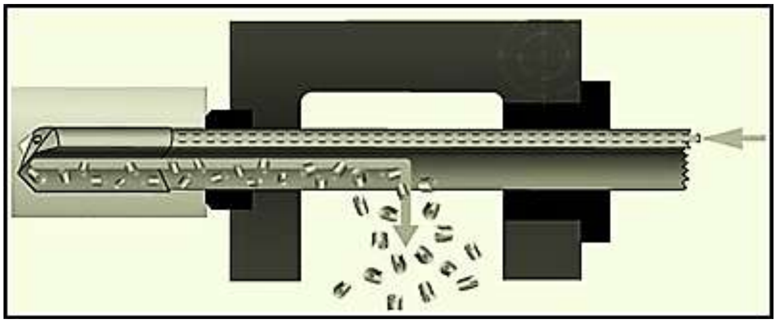


SIZE	A	B	C	D
1	3/64"	.125	3\32	3/16" TO 5/16"
2	5/64"	.1875	9/64	3/8" TO 1"
3	7/64"	.250	3/16	1 1/4" TO 2"
4	1/8"	.3125	15/64	2 1/4" TO 4"
5	3/16"	.4375	21/64	4" TO 6"

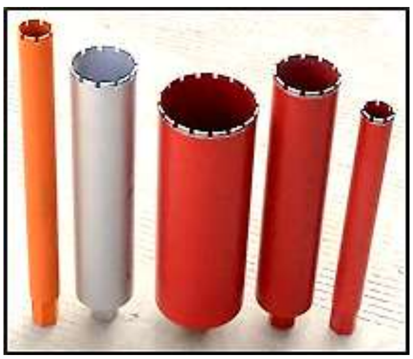
- **Spade Drills:** Are used for rough boring in wood. They tend to cause splintering when they emerge from the workpiece. They are flat, with a centering point and two cutters. The cutters are often equipped with spurs in an attempt to ensure a cleaner hole. With their small shank diameters relative to their boring diameters, spade bit shanks often have flats forged or ground into them to prevent slipping in drill chucks.



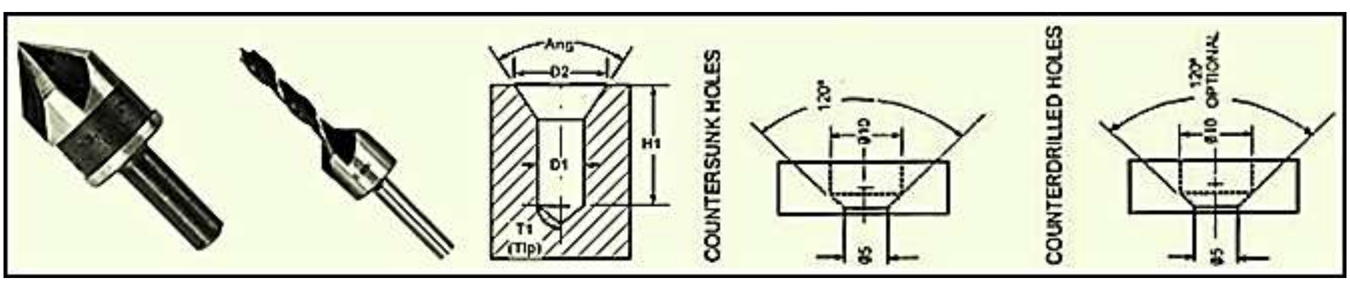
- **Gun Drills:** Originally developed to drill out gun barrels and smaller diameter deep holes. The key feature of gun drilling is that the bits are self-centering, for deep accurate holes. The bits use a rotary motion similar to a twist drill; however, the bits are designed with bearing pads that slide along the surface of the hole keeping the drill bit on center. Gun drilling is usually done at high speeds and low feed rates.



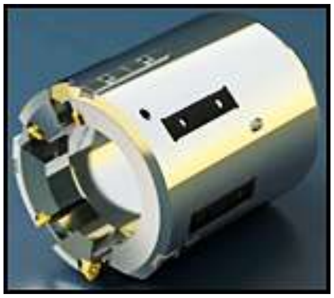
- **Core Drills:** A bit used to enlarge an existing hole, is called a core drill bit. The name comes from a casting or a stamped (punched) for drilling out a *foundry core*. These core drill bits are similar in appearance to reamers as they have no cutting point or means of starting a hole. They have 3 or 4 flutes which enhance the finish of the hole and ensure the bit cuts evenly. Core drill bits differ from reamers in the amount of material they are intended to remove.



- **Countersink Drill:** is a conical hole cut into a manufactured object. A common use is to allow the head of a bolt or screw, with a shape exactly matching the countersunk hole, to sit flush with or below the surface of the surrounding material. A countersink may also be used to remove the burr left from a drilling or tapping operation.

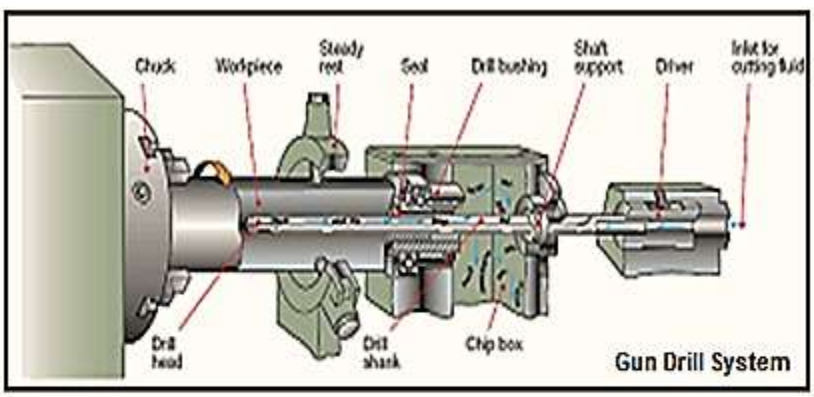


- **Trepanning Tools:** Is commonly used for creating larger diameter holes (up to 915 mm (36.0 in) where a standard drill bit is not feasible or economical. Trepanning removes the desired diameter by cutting out a solid disk similar to the workings of a drafting compass. Trepanning is performed on flat products such as sheet metal, granite (curling stone), plates, or structural members like I-beams.

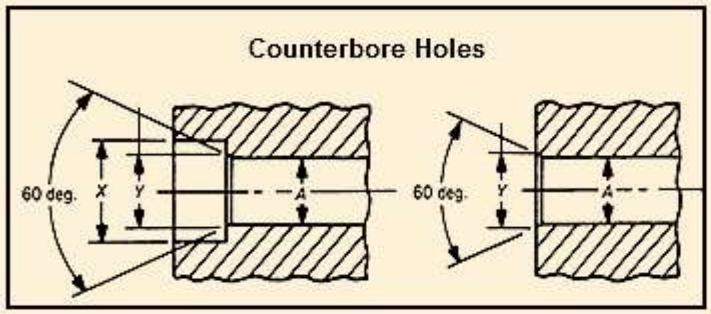


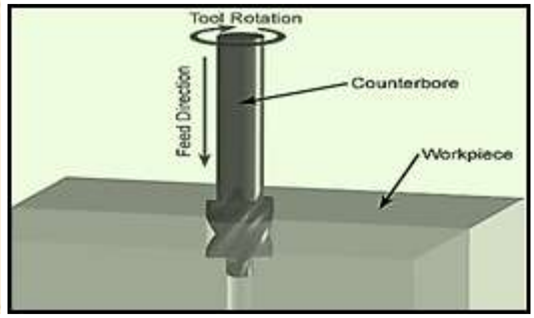

- **Microdrilling:** Refers to the drilling of holes less than 0.5 mm (0.020 in). Drilling of holes at this small diameter presents greater problems since coolant fed drills cannot be used and high spindle speeds are required. High spindle speeds that exceed 10,000 RPM also require the use of balanced tool holders.

- **Deep Hole Drilling:** Is defined as a hole depth greater than five times the diameter of the hole. Deep hole drilling is generally achievable with a few tooling methods, usually gun drilling or BTA drilling. This technology consists in fractionating chips by a small controlled axial vibration of the drill. Therefore, the flutes of the drill easily remove the small chips. There are three different systems for Deep Hole Drilling: The Gun Drill System; The Single Tube System; The Ejector System.

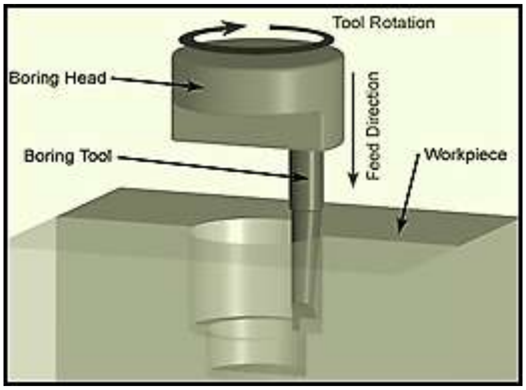


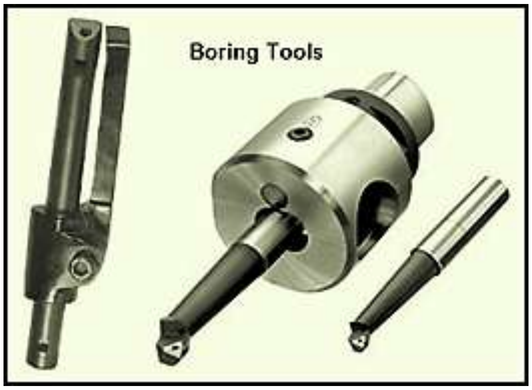
- **Counterbore:** Is a cylindrical flat-bottomed hole that enlarges another coaxial hole, or the tool used to create that feature. A counterbore hole is typically used when a fastener, such as a socket head cap screw, is required to sit flush with or below the level of a workpieces surface. A counterbore is a *flat-bottomed* enlargement of a smaller coaxial hole; a countersink is a *conical* enlargement of a hole.





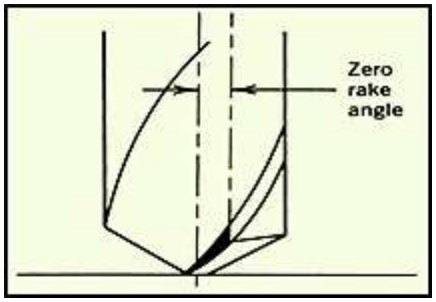
- **Boring:** Is the process of enlarging a hole that has already been drilled (or cast), by means of a single-point cutting tool, for example, as in boring a gun barrel or an engine cylinder. Boring is used to achieve greater accuracy of the diameter of a hole, and can be used to cut a tapered hole. Boring can be viewed as the internal-diameter counterpart to turning, which cuts external diameters.

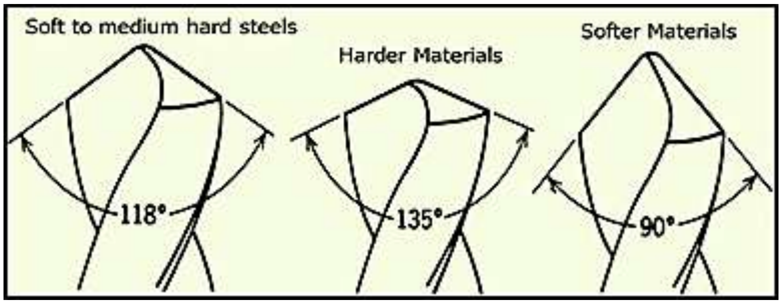




Drill Sizes and Geometry:

- Twist drills may be purchased "off the shelf" in diameters from 0.0135" to 2 ¼ inches.
- Sizes are in fractions, numbers and letters, and metric.
- Standard Twist drills may have straight or tapered shanks.
- Modification of the drill point when drilling brass. (When a straight fluted twist drill is not available.)





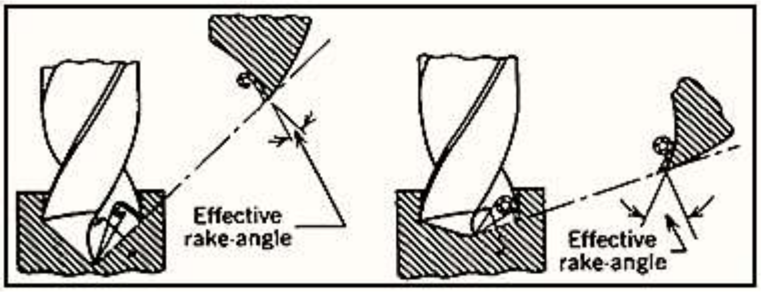
Drill Point Angles:

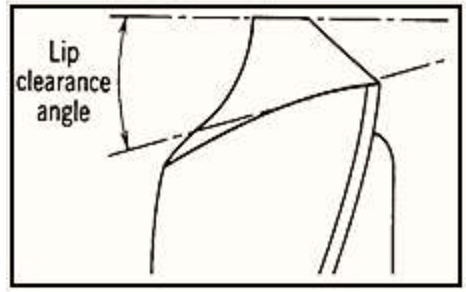
The drill point angle may be modified to cut a variety of materials.

- Use 90º to 140º for aluminum alloys.
- Use 70º to 80º for magnesium alloys.
- Use 118º to 135º for high strength steels.

Changing the drill point angle and Lip relief (clearance) angles:

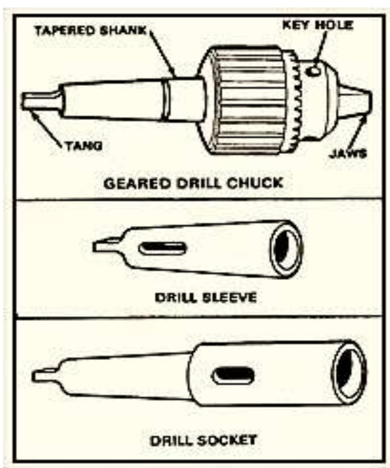
- The rake angle is influenced by the drill point angle.
- Lip relief angles are normally 10º to 15º;
- Lip relief for high strength or tougher steels is between 7º to 12º (Backing for cutting edge);
- Lip relief for softer or free machining materials may be between 12º to 18º;

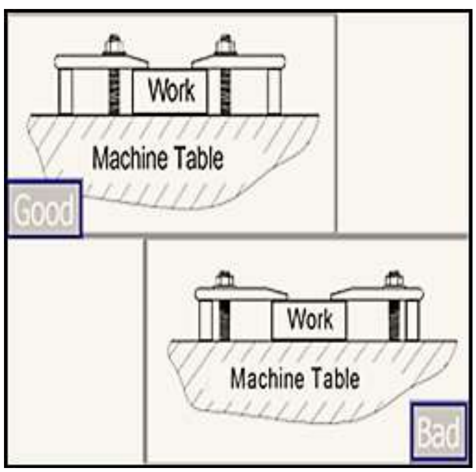


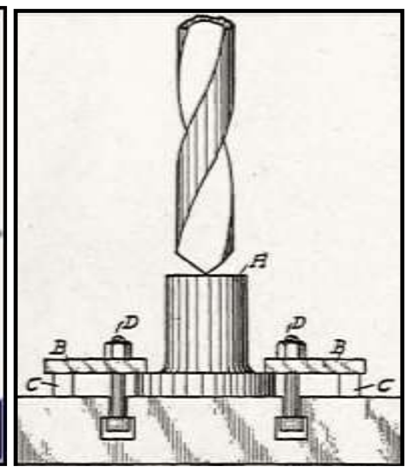


Drill Holding & Clamping of Workpieces:

- Morse taper drill sleeves (Use a drift device for removal);
- Jacob's Chuck (danger due flying chuck keys);
- Holding the Workpiece;
  - Using a vise;
  - Clamping directly to the table, using a drill jig.







Clamping work to a machine table

Drill Bits Geometry and Nomenclature:

Drill bits are cutting tools used to make cylindrical holes of circular cross-sections and are held in a tool called "drill", which rotates and provides torque and axial force, creating holes of many sizes and characteristics. Drill bits are supplied in many sizes for many uses. Specialized bits are also available for non-cylindrical-shaped holes.

- The ***shank*** is the part of the drill bit grasped by the chuck of a drill. The cutting edges of the drill bit are at one end, and the shank is at the other.
- Drill bits come in standard sizes. A comprehensive drill bit and tap size chart lists metric and imperial sized drill bits alongside the required screw tap sizes. The concept of *drill bit* or *bit* is used throughout to refer to a bit for use in a drilling machine, and *drill* refers always to a drilling machine.

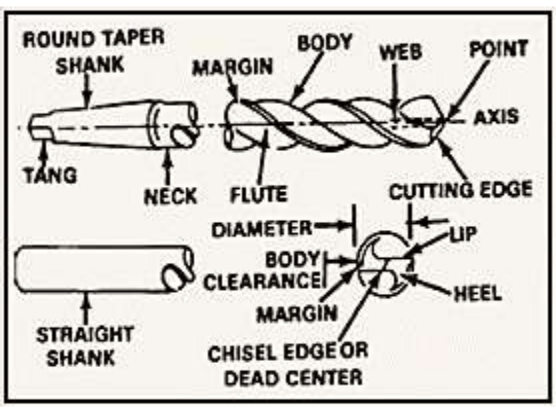
Geometry and Nomenclature:

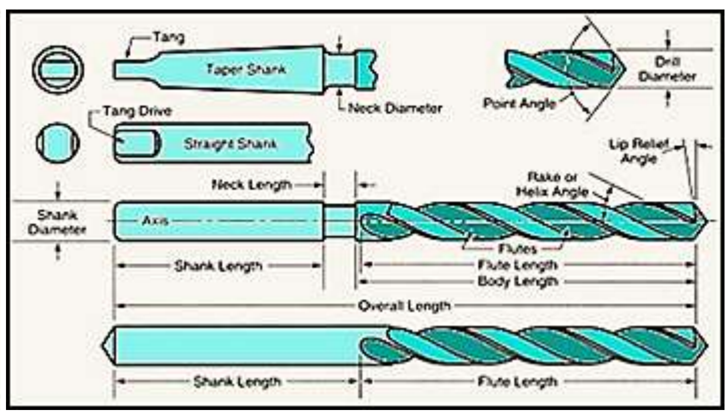
- The **spiral** (or rate of twist) in the drill bit controls the rate of chip removal. A fast spiral drill bit is used in high feed rate applications under low spindle speeds, where removal of a large volume of swarf is required. Low spiral drill bits are used in cutting applications where high cutting speeds are traditionally used, and where the material has a tendency to gall on the bit or otherwise clog the hole, such as aluminum or copper.

- The **point angle**, or the angle formed at the tip of the bit, is determined by the material the bit will be operating in. Harder materials require a larger point angle, and softer materials require a sharper angle. The correct point angle for the hardness of the material controls wandering, chatter, hole shape, wear rate, and other characteristics.

- The **lip angle** determines the amount of support provided to the cutting edge. A greater lip angle will cause the bit to cut more aggressively under the same amount of point pressure as a bit with a smaller lip angle. Both conditions can cause binding, wear, and eventual catastrophic failure of the tool. The proper amount of lip clearance is determined by the point angle. A very acute point angle has more web surface area presented to the work at any one time, requiring an aggressive lip angle, where a flat bit is extremely sensitive to small changes in lip angle due to the small surface area supporting the cutting edges.

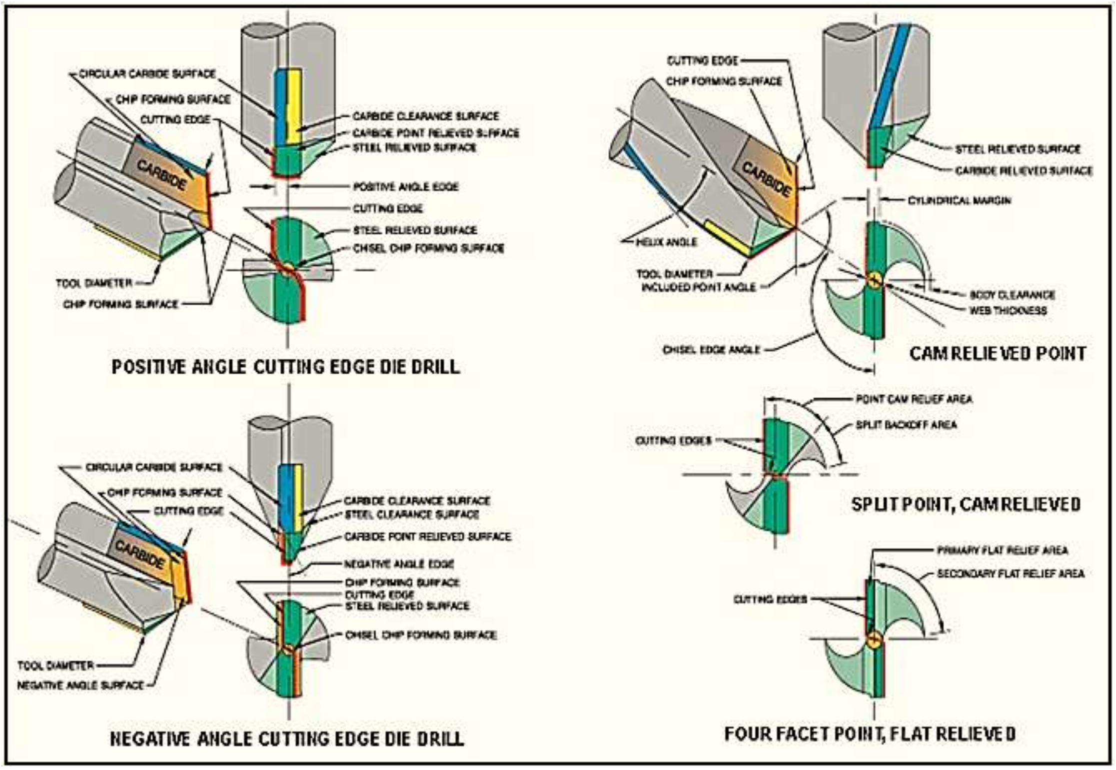
- The **length** of a bit determines how long a hole can be drilled, the stiffness of the bit and accuracy of the resultant hole. Twist drill bits are available in standard lengths, referred to as Stub-length or Screw-Machine-length (short), the extremely common Jobber-length (medium), and Taper-length or Long-Series (long).
    - The diameter-to-length ratio of the drill bit is usually between 1:1 and 1:10. Much higher ratios are possible (e.g., "aircraft-length" twist bits,), but the higher the ratio, the greater the technical challenge of producing good work. The best geometry to use depends upon the properties of the material being drilled. The following table lists geometries recommended for some commonly drilled materials.





Tool geometry			
Workpiece material	**Point angle**	**Helix angle**	**Lip relief angle**
Aluminum	90 to 135°	32 to 48°	12 to 26°
Brass	90 to 118°	0 to 20°	12 to 26°
Cast iron	90 to 118°	24 to 32°	7 to 20°
Mild steel	118 to 135°	24 to 32°	7 to 24°
Stainless steel	118 to 135°	24 to 32°	7 to 24°
Plastics	60 to 90°	0 to 20°	12 to 26°

Types of Drills with Inserts:



Drill Materials:

Many hard materials, such as carbides, are much more brittle than steel, and are far more subject to breaking, particularly if the drill is not held at a very constant angle to the workpiece, e.g. when hand-held.

- **Soft Low Carbon Steel Bits:** Are inexpensive, but do not hold an edge well and require frequent sharpening. Even working with hardwoods rather than softwoods can noticeably shorten their lifespan.

- **High Carbon Steel Bits:** Are more durable than low-carbon steel bits due to the properties conferred by hardening and tempering the material. If they are overheated (e.g., by frictional heating while drilling) they lose their temper, resulting in a soft cutting edge. These bits can be used on wood or metal.

- **High Speed Steel (HSS):** Is a form of tool steel; HSS bits are hard, and much more resistant to heat than high carbon steel. They can be used to drill metal, hardwood, and most other materials at greater cutting speeds than carbon steel bits, and have largely replaced carbon steels.

- **Cobalt Steel Alloys:** Are variations on high speed steel which contain more cobalt. They hold their hardness at much higher temperatures, and are used to drill stainless steel and other hard materials. The main disadvantage of cobalt steels is that they are more brittle than standard HSS.

- **Tungsten Carbide:** Are extremely hard, and can drill virtually all materials while holding an edge longer than other bits. The material is expensive and much more brittle than steels; consequently they are mainly used for drill bit tips, small pieces of hard material fixed or brazed onto the tip of a bit made of less hard metal. It is becoming common in job shops to use solid carbide bits. In some industries, most notably PCB manufacturing, requiring many holes with diameters less than 1 mm, carbide bits are used.

- **Polycrystalline Diamond (PCD):** Is among the hardest of all tool materials and is therefore extremely resistant to wear. It consists of a layer of diamond particles, typically about 0.5 mm (0.019") thick, bonded as a sintered mass to a tungsten carbide support. Bits are fabricated using this material by either brazing small segments to the tip of the tool to form the cutting edges, or by sintering PCD into a vein in the tungsten carbide "nib".

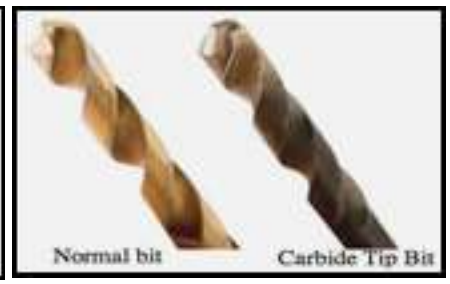
- **Carbide Nib:** Can be brazed to a carbide shaft; it can then be ground to complex geometries that would otherwise cause braze failure in the smaller "segments". PCD bits are typically used in the automotive, aerospace, and other industries to drill abrasive aluminum alloys, carbon fiber reinforced plastics, and other abrasive materials, and in applications where machine downtime to replace or sharpen worn bits is exceptionally costly.

Types of Drill Bits:

Twist Drill Bits: Was **invented by Steven A. Morse** of East Bridgewater, Massachusetts in 1861.The original method of manufacture was to cut two grooves in opposite sides of a round bar, then to twist the bar (the tool name) to produce the helical flutes. Nowadays, the drill bit is usually made by rotating the bar while moving it past a grinding wheel to cut the flutes in the same manner as cutting helical gears.


Common twist drill bit


Twist drill bit with cutting edges



The twist drill bit is the type produced in largest quantity today. It has a cutting point at the tip of a cylindrical shaft with helical flutes. The flutes act as an **Archimedean screw**. Twist drill bits range in diameter from 0.002 to 3.5 in (0.051 to 89 mm) and can be as long as 25.5 in (650 mm). The geometry and sharpening of the cutting edges is crucial to the performance of the bit. Small bits that become blunt are often discarded because sharpening them correctly is difficult and they are inexpensive. For larger bits, special grinding jigs are available. A **special tool grinder** is available for **sharpening** or **reshaping cutting surfaces** in order to optimize the bit for a particular material.

Drill Bit Geometry: The **spiral** (or rate of twist) in the drill bit controls the rate of chip removal. Low spiral (low twist rate or "elongated flute") drill bits are used in cutting applications where high cutting speeds are traditionally used, where the material has a tendency to gall or otherwise clog the hole, such as aluminum or copper. The **point angle**, or the angle formed at the tip of the bit, is determined by the material the bit will be operating in. Harder materials require a larger point angle, and softer materials require a sharper angle.

The **lip angle** determines the amount of support provided to the cutting edge. A greater lip angle will cause the bit to cut more aggressively under the same amount of point pressure as a bit with a smaller lip angle. The **length** of a bit determines how long a hole can be drilled, and also determines the stiffness of the bit and accuracy of the resultant hole. Twist drill bits are available in standard lengths, referred to as Stub-length or Screw-Machine-length (short), the extremely common Jobber-length (medium), and Taper-length or Long-Series (long).

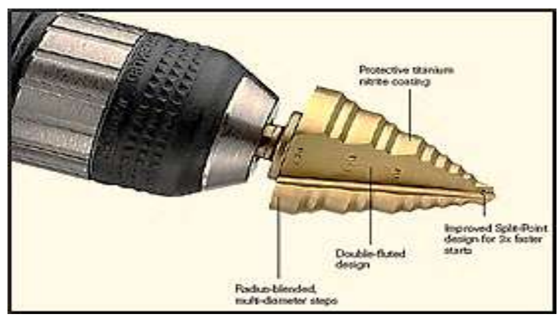
The most common **twist drill bit** has a point angle of **118 degrees**, acceptable for use in wood, metal, plastic, and most other materials, although it does not perform as well as using the optimum angle for each material. In most materials it does not tend to wander or dig in. A more aggressive angle, such as **90 degrees**, is suited for very soft plastics and other materials; it would wear rapidly in hard materials. Such a bit is generally self-starting and can cut very quickly. A shallower angle, such as **150 degrees**, is suited for drilling steels and other tougher materials.

No point Angle Drill Bits: Are used in situations where a blind, flat-bottomed hole is required. These bits are very sensitive to changes in lip angle, and fast cutting may suffer premature wear. *Long series* drill bits are long twist drill bits. However, they are not the best tool for routinely drilling deep holes, as they require frequent withdrawal to clear the flutes of swarf and to prevent breakage of the bit. Instead, gun drill bits are preferred for deep hole drilling.


Twist drill bit with Morse taper shank and 11/32" (8 mm) drill bits - long-series Morse or plain Morse

Step Drill Bits: Is a drill bit that has the tip ground down to a different diameter. The transition between this ground diameter and the original diameter is either straight, to form a counterbore, or angled, to form a countersink. The advantage to this style is that both diameters have the same flute characteristics, which keeps the bit from clogging when drilling in softer materials, such as aluminum; in contrast, a drill bit with a slip-on collar does not have the same benefit. Most of these bits are custom-made for each application, which makes them more expensive.

Unibits: Is a roughly conical bit with a stair-step profile. The larger-size bits have blunt tips and are used for hole enlarging. One drill bit can drill the entire range of holes necessary on a countertop, speeding up installation of fixtures. The unibit was **invented by Harry C. Oakes,** patented in 1973, sold only by the Unibit Corporation until the patent expired (1980), and later sold by other companies. They are most commonly used on softer materials, such as plywood, particleboard, drywall, acrylic, and laminate, very thin sheet metal, but are ideal for use in electrical work where thin steel, aluminum or plastic boxes and chassis are encountered.

Left-Hand Bits: Are almost always **twist bits** predominantly used in the repetition engine-ring industry on screw machines or drilling heads. Left-handed drill bits allow a machining operation to continue where either the spindle cannot be reversed or the design of the machine makes it more efficient to run left-handed.


Left-hand bit Lip and Spur drill bit

Lip and Spur Drill Bits: The lip and spur drill bit is a variation of the twist drill bit which is optimized for drilling in wood. It is also called the brad point bit or dowelling bit. For metalwork, this is countered by drilling a pilot hole with a spotting drill bit. In wood, the lip and spur drill bit is another solution: The center of the drill bit is given not the straight chisel of the twist drill bit, but a spur with a sharp point and four sharp corners to cut the wood. Lip and spur drill bits are effective in soft plastic. Lip and spur drill bits are ordinarily available in diameters from 3 mm (1/8") to 16 mm (5/8").

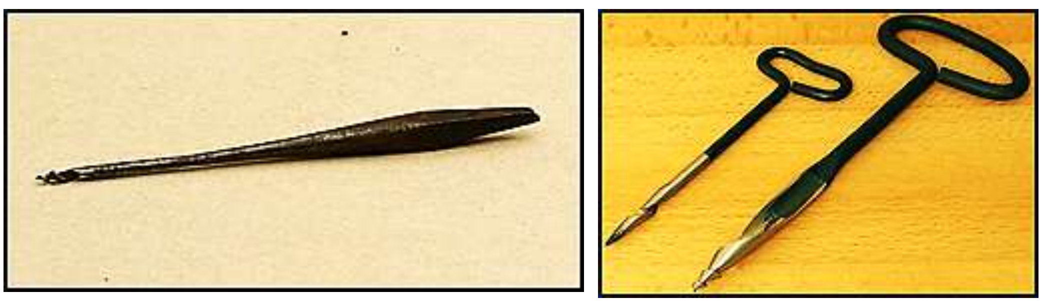
Forstner Bits: Named after their inventor, **Benjamin Forstner**, bore precise, flat-bottomed holes in wood, in any orientation with respect to the wood grain. They can cut on the edge of a block of wood, and can cut overlapping holes, normally used in drill presses or lathes rather than in portable drills. Unlike most other types of drill bits, they are not practical to use as hand tools. Bits are commonly available in sizes from 8 mm (5/16") to 50 mm (2") diameter**.** Saw tooth bits are available up to 100 mm (4") diameter. Originally the Forstner bit was very successful with gunsmiths because of its ability to drill an exceedingly smooth-sided hole.


Auger Bits: The cutting principles of the auger bit are the same as those of the center bit above. The auger adds a long deep spiral flute for effective chip removal. Two styles of auger bit are commonly used in hand braces: the Jennings-pattern bit has a self-feeding screw tip, two spurs and two radial cutting edges. This bit has a double flute starting from the cutting edges, and extending up the shank of the bit, for waste removal. This pattern of bit was developed by Russell Jennings in the mid-19th century.


Irwin or Solid-Center Auger Bits: Is similar, the only difference being that one of the cutting edges has only a "vestigial flute" supporting it, which extends only about 1/2" (12 mm) up the shank before ending. The other flute continues full-length up the shank for waste removal. The Irwin bit may afford greater space for waste removal, greater strength (because the design allows for a center shank of increased size within the flutes, as compared to the Jennings bits), or smaller manufacturing costs. This style of bit was invented in 1884, and the rights sold to Charles Irwin who patented and marketed this pattern.

Irwin Bits

Gimlet Bits: Are self-contained tools for boring small holes in wood by hand. Since about 1850, gimlets have had a variety of cutter designs, but some are still produced with the original version. The gimlet bit is intended to be used in a hand brace for drilling into wood. It is the usual style of bit for use in a brace for holes below about 7 mm (1/4") diameter. The tip of the gimlet bit acts as a tapered screw, to draw the bit into the wood and to begin forcing aside the wood fibers, without necessarily cutting them. The cutting action occurs at the side of the broadest part of the cutter. Most drill bits cut the base of the hole. The gimlet bit cuts the side of the hole.



Gimlet bit for wood, made sometime before 1950.

Hinge Sinker Bits: As many European kitchen cabinets are made from particle board or medium-density fiber-board (MDF) with a laminated plastic veneer these types of pressed wood boards are not very strong, and the screws of the butt hinge extend to pull out. A specialist hinge has been developed which uses the walls of a 30 mm (1-3/16") diameter hole, bored in the particle board, for support, very common and relatively successful construction method.

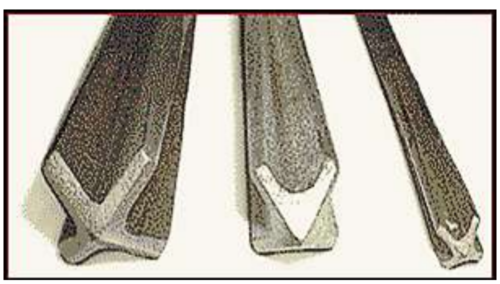
30 mm hinge sinker bit

Wood Bits: Are also known as expansive wood bit, has a small center pilot bit with an adjustable, sliding cutting edge mounted above it, usually containing a single sharp point at the outside, with a set screw to lock the cutter in position. When the cutting edge is centered, the hole drilled will be small, and when the cutting edge is slid outwards, a larger hole is drilled. A ruler or Vernier scale is usually provided to allow precise adjustment of the bit size.


Wood bits meant for use in a brace

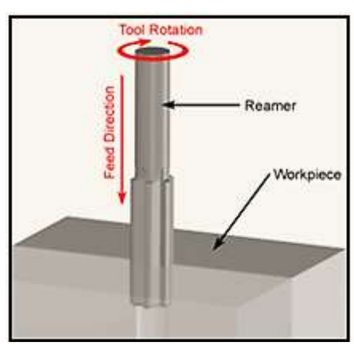
Masonry Drill Bits: Are soft steels machined with a mill rather than ground. An insert of tungsten carbide is brazed into the steel to provide the cutting edges. Masonry bits of the style shown are commonly available in diameters from 5 mm to 40 mm. For larger diameters, core bits are used. Masonry bits up to 1000 mm (39") long can be used with hand-portable power tools, and are very effective for installing wiring and plumbing in existing buildings.


Star Drill Bits: Are similar in appearance and function to a hole punch or chisel, is used as a hand powered drill in conjunction with a hammer to drill into stone and masonry. A star drill bit's cutting edge consists of several blades joined at the center to form a star pattern.




Reamers: Are used for accuracy, after drilling or boring holes and good surface finishing. Different types of **reamers** of standard sizes are available for different applications. The figure below shows some of the different types of reamers commonly utilized.



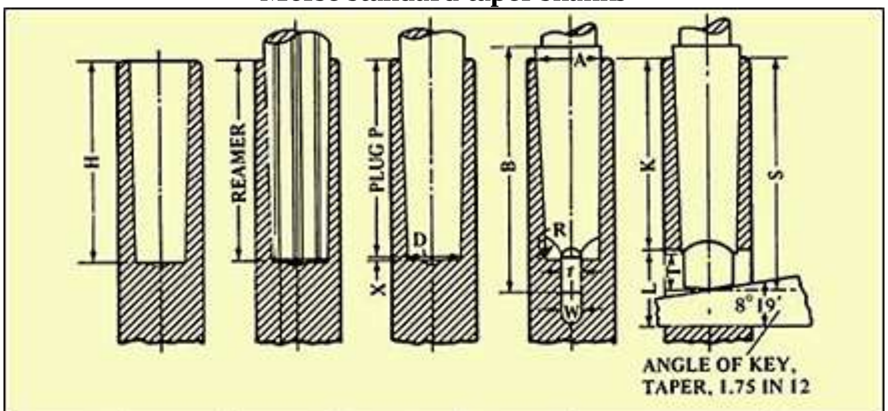


Holding of Drill Bits:

Parallel Drill Shank: Drills and similar tools with **parallel shanks** are held in a drill chuck, and to ensure maximum grip, the chuck should be tightened using the correct size of chuck key. This prevents the drill from spinning during use and chewing up the drill shank.

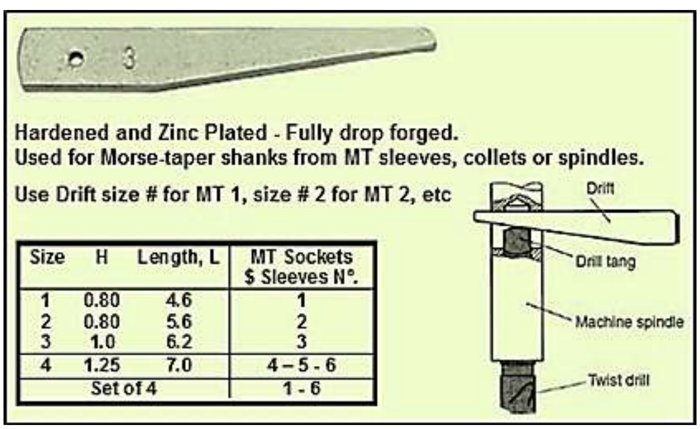
Morse Taper Shank: Is fitted with a Morse-taper shank, which fits into a corresponding Morse taper in the spindle. The numbers 1, 2, 3, 4, 5, and 6 identifies the size of Morse taper from smallest to largest. The included angle of each taper is different but is very small, being in the region of 3 degrees. If the two mating tapered surfaces are clean and in good condition, this shallow taper is sufficient to provide a drive between the two surfaces. At the end of the taper shank, two fiats are machined, leaving a portion known as the tang. Drills are also available with Morse-taper shanks, which fit directly into the spindle without the need for a chuck.

Morse standard taper shanks



No. of Taper	Taper per Foot	Taper per Inch	Small End of Plug *D*	Diameter End of Socket *A*	Shank		Depth of Hole *H*
					Length *B*	Depth *S*	
0	0.62460	0.05205	0.252	0.3561	$2\frac{11}{32}$	$2\frac{7}{32}$	$2\frac{1}{32}$
1	0.59858	0.04988	0.369	0.475	$2\frac{9}{16}$	$2\frac{7}{16}$	$2\frac{5}{32}$
2	0.59941	0.04995	0.572	0.700	$3\frac{1}{8}$	$2\frac{15}{16}$	$2\frac{39}{64}$
3	0.60235	0.05019	0.778	0.938	$3\frac{7}{8}$	$3\frac{11}{16}$	$3\frac{1}{4}$
4	0.62326	0.05193	1.020	1.231	$4\frac{7}{8}$	$4\frac{5}{8}$	$4\frac{1}{8}$

Drill Drift: To remove a shank from the spindle, a taper key known as a drift is used.



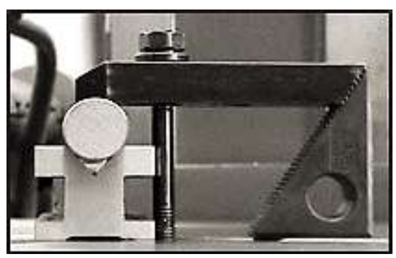
Size	H	Length, L	MT Sockets $ Sleeves N°.
1	0.80	4.6	1
2	0.80	5.6	2
3	1.0	6.2	3
4	1.25	7.0	4 – 5 - 6
Set of 4			1 - 6

Devices for Holding Workpieces:

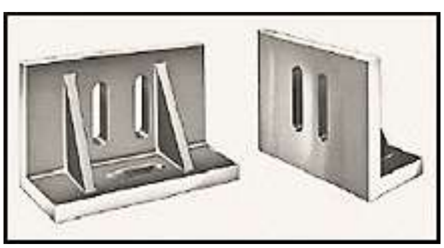
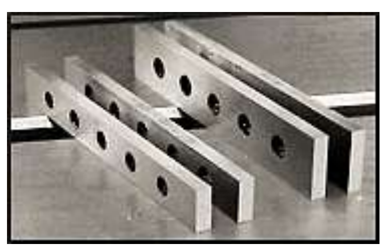
Machine Vises: The drill press vise is by far the most common type of work holding device. Modern drill press vises are capable of holding round stock, flat stock, or any other small parallel-sided parts. Most drill press vises come equipped with V shaped slots for holding round stock and stepped jaws. This will avoid contact between the drill and the vise or also use a magnetic drilling machine base onto the surface of the vise.


Direct Clamping: Work that is too large or has an odd configuration is customarily bolted directly to the table, as shown below. There are a number of accessories that can be used to help you set up parts and a variety of commercially available clamp sets can be used for directly mounting workpieces.


V-Blocks: Hold and support round work for drilling as shown. V-Blocks come in many different sizes.

Angle Plates and Parallels: An angle plate is an L-shaped piece of cast iron or steel that has tapped holes or slots to facilitate the clamping of the workpiece. Parallels are pieces of steel bar stock accurately machined so that the opposing sides are parallel to each other.

Drilling Calculations:

Drill Speed: Is usually measured in terms of the rate at which the outside or periphery of the tool moves in relation to the work being drilled. The common term for this velocity is "surface feet per minute", abbreviated as **sfm**. Every tool manufacturer has a recommended table of **sfm values** for their tools. The peripheral and rotational velocities of the tool are related as shown in the following equation:

1) $V, V_c = \pi * D * N$

V, Vc = recommended peripheral velocity for the tool being used (ft/min), (m/min);
D = diameter of the drill bit (ft) (m);
N = rotational velocity of the tool.

Peripheral Velocity: Is expressed in units of **feet/min** (**or m/min**) and the tool diameter is typically measured in units of inches (or units of mm). The spindle or tool rotation (in **ft/min units)**, ***N*** is found in the following manner:

2) $$N\,[rpm] = \frac{12\,[in/ft] * V_c\,[ft/min]}{\pi * D\,[in]} = \frac{V_c * 4}{D} =$$

V, Vc = Cutting speed (ft/min);
4 = Constant for calculation for RPM, (12 / π) using velocity in ft/min (except metric);
D = Diameter of the drill bit (in inches).

Example: A twist drill bit **1/2-inch (0.500-inch)** is necessary to drill a common aluminum block using a cutting speed (or velocity speed) of **200 ft/min**. The formula would be setup as follows:

$$N = \frac{200 \times 4}{0.500} = \frac{800}{0.500} = 1600\ RPM$$

The metrical peripheral velocity (*V* in **m/min)**, ***N*** is found in the following manner:

3) $$N = \frac{V_c\,(m/min) \times 320}{D\,(mm)}$$ – (See recommended tables for Vc, V)

V, Vc = Recommended peripheral speed (in m/min);
320 = A constant for all metric RPM (1000 / π) calculations (when *Vc* is in m/min);
D = Diameter of the twist drill in millimeters (or mm).

Example: A twist drill **15.0 mm** is necessary to drill a medium-carbon steel, with a recommended peripheral speed of **21.4 meters per minute**. The formula would be set up as follows:

4) $$N = \frac{V_c\,(m/min) \times 320}{D\,(mm)}$$

$$N = \frac{21.4\,(m/min) \times 320}{15\,(mm)} = 456.533\ RPM \text{ or } 457\ RPM$$

Note: The **factors 4 and 320** used in formulae are the approximated results (from **12 dividing π** and **1000 dividing π)**, for D in inches and mm respectively. The recommended use of speed should be closest to the recommended RPM.

Drilling Cutting Speed (Cs): Or surface feet per minute (SFM) is the distance a drill travels into the workpiece during each revolution of the spindle. It is expressed in thousandths of an inch or in millimeters. To calculate the cutting speed, use the following formula:

5) Cs = N * fR

Cs = Cutting speed [in/min] or [mm/min] – see recommended tables;
N = Spindle rotation [rpm] – see recommended tables;
fR= Feed rate or feed per revolution [in/rev] or [mm/rev] – see recommended tables.

Recommended speeds (Vc, V) for Common Materials (HSS):

Material	***Vc* [ft/min]**
Aluminum and its alloys	250
Bronze (high tensile)	100
Cast Iron (soft)	100
Cast Iron (medium hard)	80
Cast Iron (very hard)	20
Magnesium and its alloys	300
High nickel steel	50
Mild carbon steel	100
Carbon Steel	60
Tool steel	40
Forgings	40
Steel alloys (300-400 Brinell)	30
Stainless steel free machining	40
Stainless work hardened	20

Recommended average feed rates for 2 Flute HSS Drills:

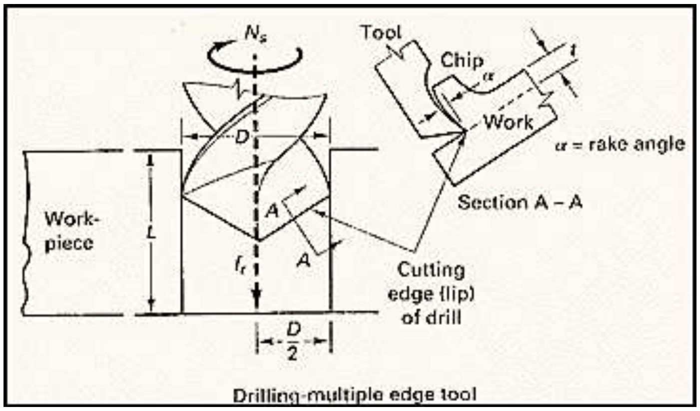
Drill Diameter [in.]	**Feedrate, *fR* [in./rev]**
Under 1/8”	up to 0.002
1/8” to 1/4"	0.002 to 0.004
5/16" to 1/2"	0.005 to 0.008
5/8" to 3/4”	0.009 to 0.011
1” and over	0.012 to 0.020

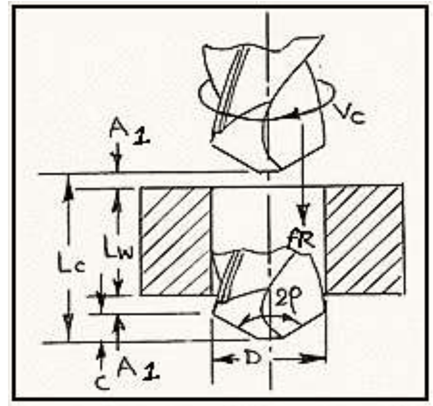
Example: Calculate the RPM, feed rate and cutting speed of a 1/4"- HSS drill bit for a soft cast iron, using a manual milling machine with a peripheral velocity of 100 ft/min. Round the results and scale the feed 50% less.

N [rpm] = 12 * V / (π * D) = *12 * 100 ft/min / (π * 0.25)* ***= 1528 rpm (use 1500 rpm) - 50% less = 750 rpm***
N [rpm] = 4 * V / D) = *4 * 100 ft/min / 0.25* = ***1600 rpm*** – ***50% less = 800 rpm***

From the table above, the minimum recommended feed per revolution for a 1/4” HSS drill bit:

fR = ~ 0.002 in/rev
Cs [in/min] = N [rpm] * f [in/rev]
Cs [in/min] = *1500 rev/min * 0.002 in/rev* = ***3.0 in/min*** – ***50 % less = 1.5 in/min***





6) Cutting Time (min):

$$Ct = \frac{\pi * D * (Lw + 2A_1)}{12 * V * f_R} \text{ – (ft/min) – or – } Ct = \frac{\pi * D * (Lw + 2A_1)}{1000 * V * f_R} \text{ - (m/min)}$$

A_1= approach and over run (0.062 ~ 0.18 in) - (~ 2 to 5 mm)
fr = feedrate (ft/min) - (in/min) [mm/min] – see recommended tables
f_R = feed per revolution [in/rev] or [mm/rev] – see recommended tables
V, Vc = peripheral velocity = π * D * N [ft/min] – [m/min]
L = Lw = Length of hole (in) or (mm) – [*Attention to conversions feet, inches, meters or millimeters].

7) Material Removal Rate (MRR):

$$MRR = \frac{\text{Volume Removed}}{\text{Cutting Time}} \ (ft^3/min)\ (mm^3/min) = \frac{\pi * L * D^2 * fR * N}{4 * L} =$$

Note: Excessive **heat** at the drill point will result in **premature failure**. Then, a reduction in speed and feed is necessary, where coolant cannot be effectively applied. Consequently, **feeds and speeds** should be **reduced up to 50%** when drilling holes **deeper than 3 times** the drill diameters.

Recommended values for drilling – HSS tools:

Workpiece material	Hardness [Bhn]	Cutting material	Cutting speed Vc [feet/min]		Vc [m/min]		Feed rate per revolution f [inch]		f [mm]	
			MIN	MAX	MIN	MAX	MIN	MAX	MIN	MAX
Cast irons	190...320	High speed steel	33	295	10	90	0,002	0,008	0,050	0,200
Steel - plain carbon	85. . .200	High speed steel	49	149	15	45	0,002	0,008	0,050	0,200
Steel - alloys	35...50Rc	High speed steel	16	66	5	20	0,002	0,008	0,050	0,200
Steel - tool	50. . .58Rc	High speed steel	16	66	5	20	0,002	0,008	0,050	0,200
Steel - stainless	150...450	High speed steel	16	33	5	10	0,002	0,008	0,050	0,200
Aluminum alloys	30. . . 150	High speed steel	16	377	5	115	0,002	0,008	0,050	0,200
Copper alloys	80...100Rb	High speed steel	66	230	20	70	0,002	0,008	0,050	0,200
Nickel alloys	80...360	High speed steel	33	66	10	20	0,002	0,008	0,050	0,200
Titanium	250...375	High speed steel	16	49	5	15	0,002	0,008	0,050	0,200

Recommended values for gun drilling – carbide tools:

Workpiece material	Hardness	Cutting speed				Gun Drill Diameters [inch]											
	[Bhn]	Vc [feet/min]		Vc [m/min]		5/64" - 5/32"		5/32" - 1/4"		1/4" - 1/2"		1/2" - 3/4"		3/4" - 1"		1" - 2"	
						Feed - f [inch/revolution]											
		MIN	MAX	MIN	MAX	MIN	MAX	MIN	MAX	MIN	MAX	MIN	MAX	MIN	MAX	MIN	MAX
Cast irons- soft	120-220	250	350	76	107	0,00015	0,00025	0,0003	0,001	0,0015	0,001	0,003	0,002	0,005	0,003	0,007	0,003
Cast irons - hard	220-320	150	200	46	61			0,0003	0,0005								
Ductile iron	140-260	200	300	61	91	0,00015	0,00025	0,0003	0,0005		0,0006		0,001		0,002		0,002
Malleable iron	110-240	250	350	76	107	0,00015	0,00025	0,0003	0,0005		0,0006		0,001		0,002		0,002
Steel - soft	85...200	425	675	130	206	0,00015	0,00025	0,0003	0,0005		0,0006		0,001		0,001		0,002
Steel - Medium	200-325	225	450	69	137												
Steel - Hard	325-450	130	200	40	61												
Stainless Steel-Soft	135-275	250	300	76	91	0,00015	0,00025	0,0003	0,0005		0,0006		0,001		0,001		0,002
Stainless Steel-Hard	275-425	150	225	46	69												
Aluminum alloys-except Die casting			650		198	0,00015	0,00025	0,0003	0,001		0,003		0,005		0,008		0,01
Alum.Die casting			650		198												
Magnesium			650		198												
Brass and Bronze		500	600	152	183	0,00015	0,00025	0,0003	0,0005	0,001	0,003	0,003	0,005	0,005	0,008	0,008	0,01
Copper			350		107						0,001		0,003		0,005		0,008

Common RPM and Feed Rate	Dia.	1/16″	1/8″	3/16″	1/4″	5/16″	3/8″	1/2″	5/8″	3/4″	1″
Cast Iron	**RPM**	6000	3000	2000	1450	1150	975	725	600	500	375
	HP	0.01	0.04	0.11	0.18	0.28	0.38	0.57	0.80	1.10	1.75
	Feed	0.001	0.002	0.004	0.005	0.006	0.007	0.008	0.009	0.010	0.012
Low Carbon Steel	**RPM**	4300	2150	1450	1100	900	750	550	450	360	275
	HP	0.01	0.05	0.12	0.20	0.30	0.35	0.68	1.05	1.50	2.10
	Feed	0.001	0.003	0.005	0.007	0.008	0.009	0.012	0.014	0.016	0.018
Stainless Steel	**RPM**	3200	1550	1025	775	620	510	400	310	360	270
	HP	0.01	0.05	0.12	0.21	0.40	0.48	0.85	1.45	2.05	3.1
	Feed	0.001	0.002	0.003	0.004	0.005	0.005	0.007	0.009	0.010	0.011

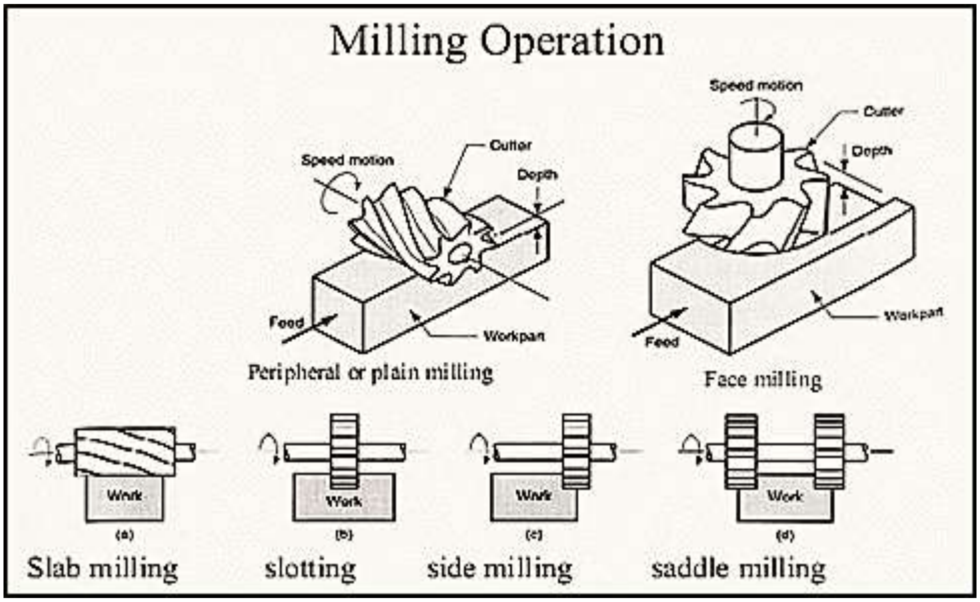
Milling Machines:

Milling machines were first invented and developed by **Eli Whitney** to mass produce interchangeable musket parts. Although crude, these machines assisted man in maintaining accuracy and uniformity while duplicating parts that could not be manufactured with the use of a file. Milling is typically used to produce parts that are **not axially symmetric** and have many features, such as holes, slots, pockets, and even three dimensional surface contours. Another application of milling is the fabrication of tooling for other processes, for example, three-dimensional molds.

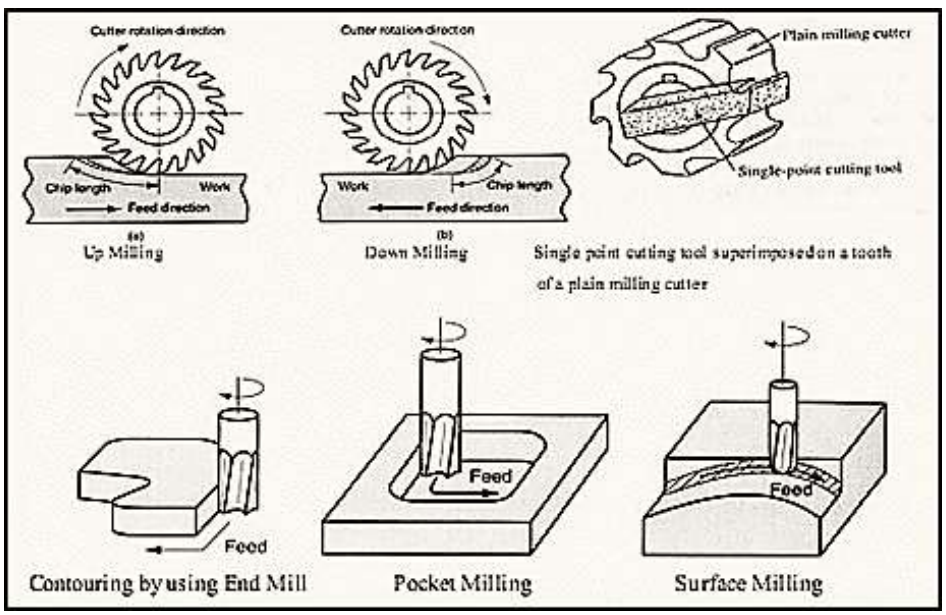
Milling is also commonly used as a **secondary process** to add or refine features on parts that were manufactured using a different process. Due to the high tolerances and surface finishes that milling can offer, it is ideal for adding precision features to a part whose basic shape has already been formed.

Milling Operations:

The milling machine removes metal with a revolving cutting tool called a milling cutter. Milling machines can be used for **boring, slotting, facing, drilling**, **keyways, racks, gears, tapping, reaming and cutting workpieces.**



The basic function of milling machines is to produce flat surfaces, as well as, surfaces of revolution, helical surfaces and contoured surfaces of various configurations. **Up milling** needs stronger holding of the job and **down milling** needs backlash free screw-nut systems for feeding. Milling machines are basically classified as being **horizontal or vertical** to indicate the axis of the milling machine spindle. These machines are also classified as **knee-type**, **ram-type**, manufacturing or **bed-type**, and **planer-type** milling machines. Most machines have self-contained electric drive motors, coolant systems, variable spindle speeds, and power-operated table feeds.



Schematic views of conventional up and down milling

Classification of Milling Machines:

Knee-Type: Characterized by a vertical adjustable worktable resting on a saddle supported by a knee. The knee is a massive casting that rides vertically on the milling machine column and can be clamped rigidly to the column in a position where the milling head and the milling machine spindle are properly adjusted vertically for operation.

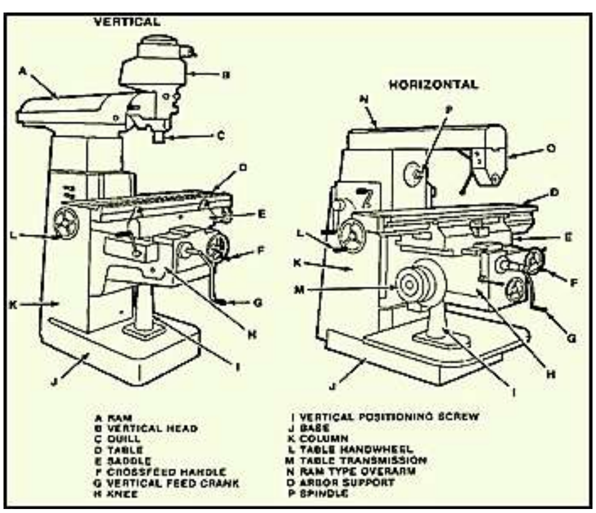
Bed-Type: The worktable is mounted directly on the bed, which replaces the knee, and can move only longitudinally. These machines have high stiffness and are used for high production work. Usually of larger size and capacity, the vertical feed is given to the milling head instead of the knee type bed.

Knee type milling machine

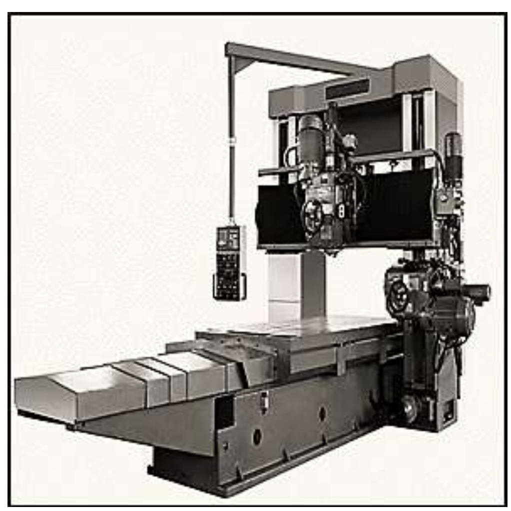
Bed type milling machine

Ram-Type: Has a spindle mounted to a movable housing on the column, permitting positioning the milling cutter forward or rearward in a horizontal plane. Two widely used ram-type milling machines are the floor-mounted universal milling machine and the swivel cutter head ram-type milling machine. The Swivel Cutter Head Ram-type Milling Machine as shown below has a cutter head containing the milling machine spindle attached to the ram.

Planer-Type: These heavy duty large machines, called plane-miller, look like planing machine where the single point tools are replaced by one or a number of milling heads; generally used for machining a number of longitudinal flat surfaces simultaneously, viz., lathe beds, table and bed of planning machine, etc.



Swivel cutter ram-type milling machines

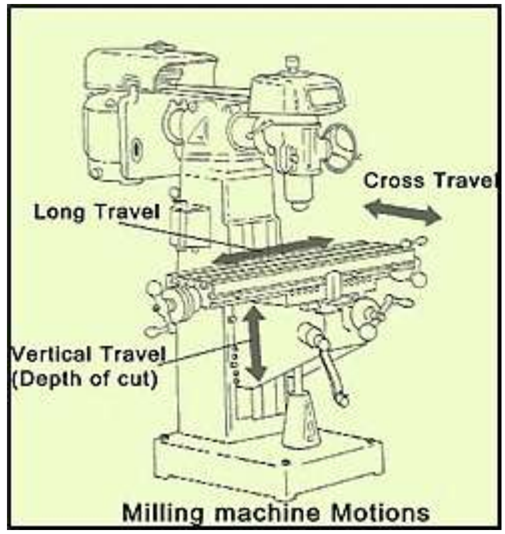


Planer-type milling machines

Rotary Table Type: Such open or closed ended high production milling machines possess one large rotary worktable and one or two vertical spindles as typically shown below. The positions of the milling head is adjusted according to the size and shape of the job. Planer and rotary table type vertical axis milling machines are not automated but provide relatively higher production rate.

Plain Horizontal Knee Type: Is a non-automatic general purpose milling machine of small to medium size, which possesses a single horizontal axis-milling arbor. The worktable can be linearly fed along three axes (X, Y, Z) only. These milling machines are most widely used for piece or batch production of jobs of relatively simpler configuration and geometry. **Horizontal axis (spindle) and swiveling bed type** are very similar to the **plain horizontal arbor knee machines** but possess one additional swiveling motion of the worktable.

Rotary table type milling machine



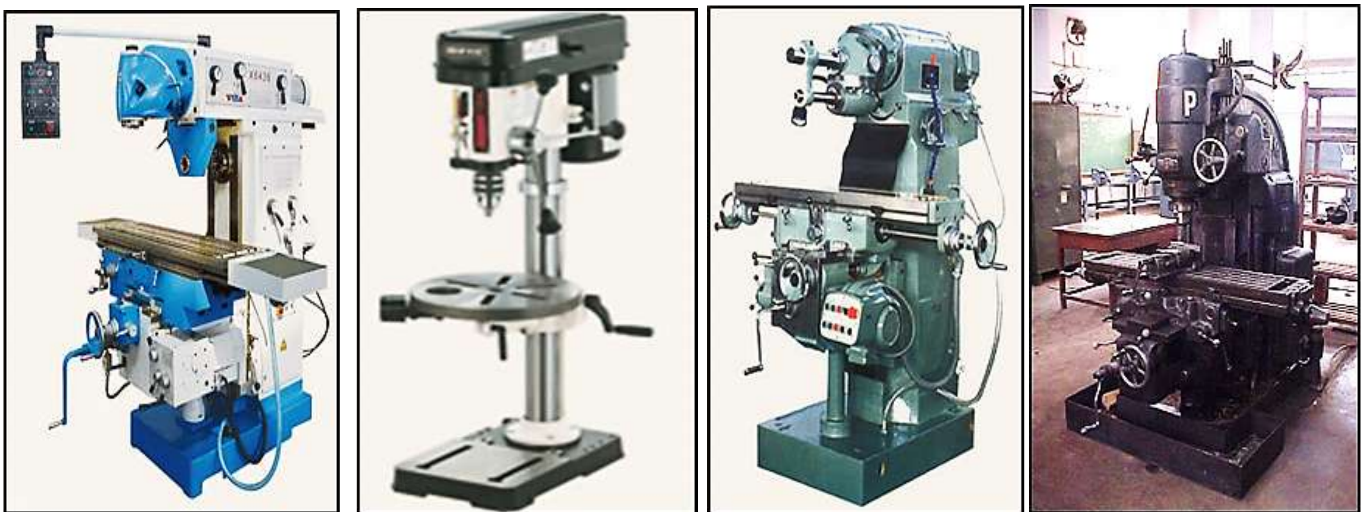
Plain horizontal knee type milling machine

Universal Swivel Head Type: Is a versatile milling machine and not only possess both horizontal milling arbor but also a vertical axis spindle; the latter spindle can be further tilted about one (X) or both the horizontal axes (X and Y), enabling machining jobs of complex shape, as typically shown below. The bench-type plain horizontal is a small version of the floor-mounted plain horizontal milling machine, mounted to a bench or a pedestal instead of directly to the floor. The milling machine spindle is horizontal and fixed in position.

Bench-Type Plain Horizontal: Is a small version of the floor-mounted plain horizontal milling machine, mounted to a bench or a pedestal instead of directly to the floor. The milling machine spindle is horizontal and fixed in position. The worktable is generally not power fed on this size machine. The saddle slides on a dovetail on the knee providing a crossfeed adjustment, moving vertically up or down the column to position the worktable in relation to the spindle.

Floor-Mounted Universal Horizontal: The basic difference between a universal horizontal milling machine and a plain horizontal milling machine is in the adjustment of the worktable, and in the number of attachments and accessories available for performing various special milling operations. The universal horizontal milling machine has a worktable that can swivel on the saddle with respect to the axis of the milling machine spindle, permitting workpieces to be adjusted in relation to the milling cutter.

Vertical Spindle Type: In this machine, typically shown below, the only spindle is vertical and works using an end mill type and face milling cutters. The horizontal spindle of a horizontal milling machine can also be converted to a vertical spindle, clamped to the column and driven from the horizontal spindle. End milling and face milling operations are more easily accomplished with this attachment, because the cutter and the surface being cut are in plain view. The table may or may not have swiveling features.



Universal swivel milling machine, Floor-mounted (bench type), Radial Universal, Vertical spindle type

Hand Milling Machine: Is the simplest form of milling machine where even the table feed is also given manually, as can be seen in figure below.

Tracer Controlled Milling Machine: Typically shown below, are mechanically or hydraulically operated semi-automatic milling machines used for lot production of cams, dies etc. by copying the masterpiece.

Hand milling machine

Tracer controlled milling machine

(a) According to nature of purposes of use: General purpose – used mainly for big or small production lots; Single purpose – thread milling machines, cam milling machines and slitting machine, generally used for batch or lot production; Special purpose – used for lot or mass production, duplicating mills, die sinkers, short thread milling, etc.

(b) According to configuration and motion of the work-holding table/bed: Knee type - typically shown below. In such small and medium duty machines, the table with the job/work travels over the bed (guides) in horizontal (X) and transverse (Y) directions and the bed with the table and job on it moves vertically (Z) up and down.

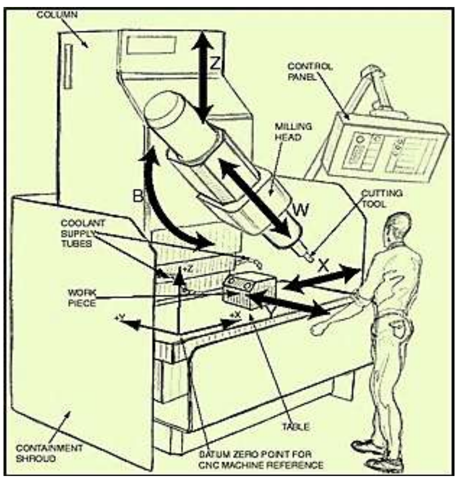
Obs.: Milling machines are mostly general purpose and used for pieces or small lot production. However, like other machine tools, some milling machines are also incorporated with certain type and degree of automation or mechanization to enhance production rate and consistency of product quality. Milling machines for short thread milling may be considered single purpose and automatic being used for mass production of small bolts and screws.

Machine Centers:

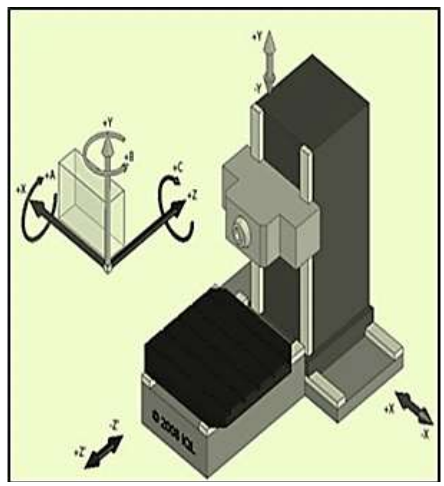
Machine Centers are programmed machines commanded by a computer numerical control (CNC). In modern CNC systems, an end-to-end component design is highly automated using a computer-aided design (designated as CAD) and a computer-aided manufacturing (designated as CAM) programs.

Computer Numerical Controlled (CNC): Flexible automation by developing and using CNC has made a great break through since mid-seventies in the field of machine tools' control. Figure below typically shows a CNC milling machine. The versatility of CNC milling machine has been further enhanced by developing what is called Machining Centre. The advantageous characteristics of CNC machine tools over conventional ones are:

- ✓ Flexibility in automation;
- ✓ Change-over (product) time, effort and cost are much less;
- ✓ Less or no jigs and fixtures are needed;
- ✓ Complex geometry can be easily machined;
- ✓ High product quality and its consistency;
- ✓ Optimum working condition is possible;
- ✓ Lesser breakdown and maintenance requirement.

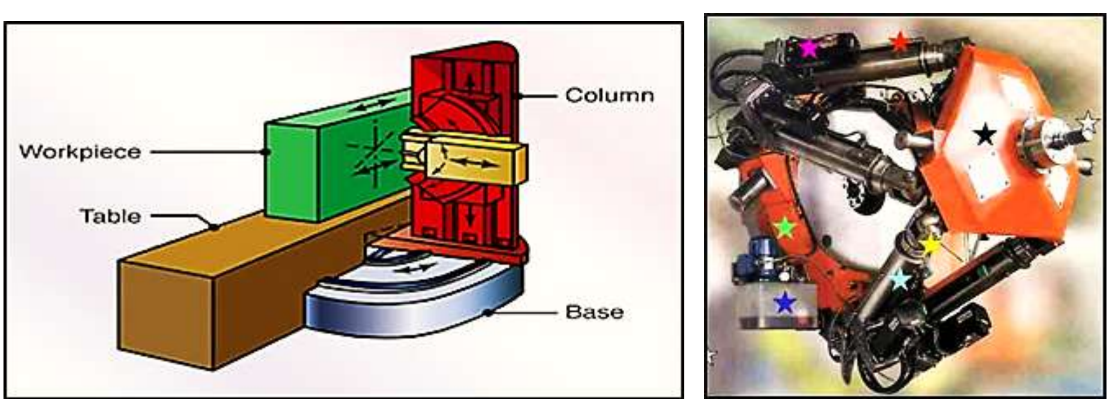


CNC Milling Machine



CNC Machining Centre

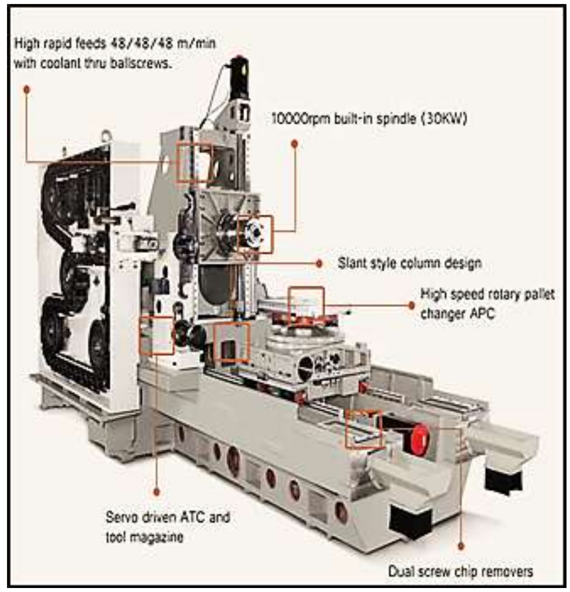
Operation Programs: The programs produce a computer file, which is interpreted to extract the commands needed to operate a particular machine via a postprocessor, and then loaded into the CNC machines for production. Since any particular component require the use of a number of different tools – drilling, turning, milling, sawing, etc., modern machines often combine multiple tools into a single "cell".



Five-Axis Profile Milling Machine

CNC Milling Machines: In many installations, a number of different machines are used with an external controller and human or robotic operators that move the component from machine to machine. The series of steps for high automation needed to produce any part must closely matches the original CAD design.


CNC Milling Machine



Vertical CNC Machine Center

Milling Machine Parts:

(1) Column: The column, including the base, is the main casting which supports all other parts of the machine. An oil reservoir and a pump in the column keep the spindle lubricated. The column rests on a base that contains a coolant reservoir and a pump that can be used when performing any machining operation that requires a coolant.

(2) Knee: The knee is the casting that supports the table and the saddle.. It is supported and can be adjusted by the elevating screw. The lever can be raised or lowered either by hand or power feed. The hand feed is usually used to take the depth of cut or to position the work, and the power feed to move the work during the machining operation.

(3) Saddle and Swivel Table: The saddle slides on a horizontal dovetail, parallel to the axis of the spindle, on the knee. The swivel table (on universal machines) is attached to the saddle and can be swiveled approximately 45°.

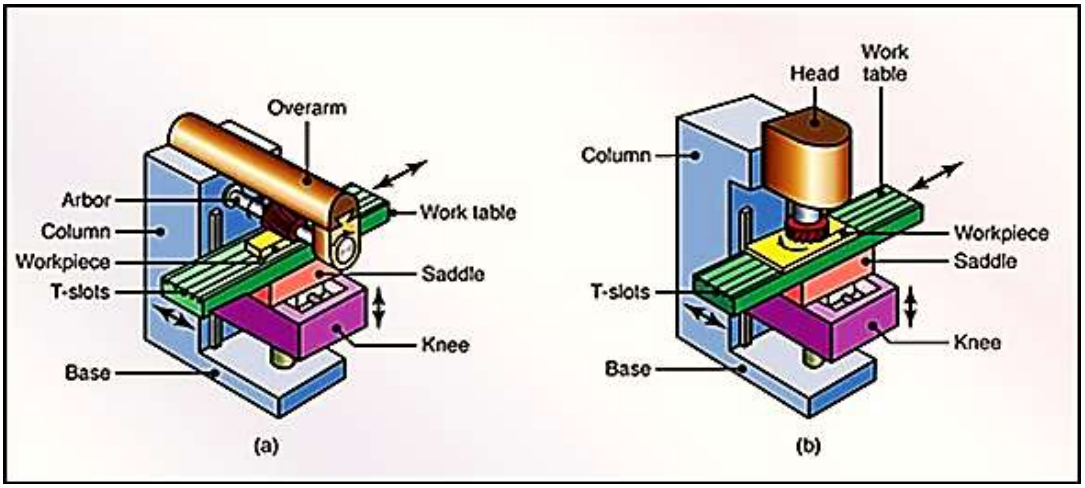
(4) Power Feed Mechanism: The power feed mechanism is contained in the knee and controls the longitudinal, transverse (in and out) and vertical feeds. The desired rate of feed can be obtained on the machine by positioning the feed selection levers as indicated on the feed selection plates.

(5) Table: The worktable is the rectangular casting located on top of the saddle. It contains several T-slots for fastening the work or work holding devices. The longitudinal directional control lever can be positioned to the left, to the right, or in the center.

(6) Spindle: The spindle holds and drives the various cutting tools. It is a shaft, mounted on bearings supported by the column. The spindle is driven by an electric motor through a train of gears, all mounted within the column. Large face mills are sometimes mounted directly to the spindle nose.

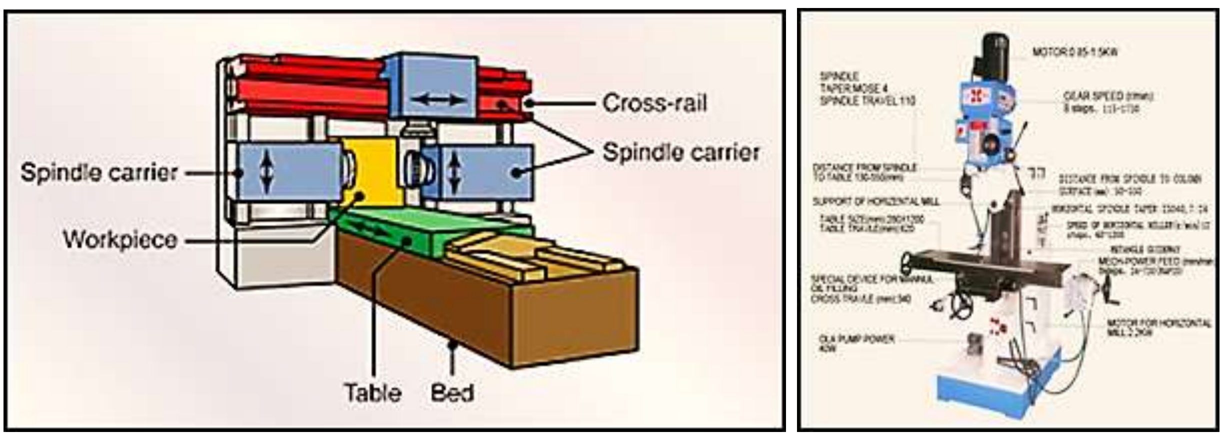
(7) Overarm: The overarm, may be a single casting that slides in the dovetail ways on the top of the column. It may consist of one or two cylindrical bars that slide through the holes in the column. On some milling machines, the coolant supply nozzle is fastened to the overarm. The nozzle can be mounted with a split clamp to the overarm after the arbor support has been placed in position.

(8) Arbor Support: It helps to keep the arbor from springing during cutting operations. Two types of arbor supports are commonly used. One type has a small diameter bearing hole, usually 1-inch maximum in diameter. An arbor support can be clamped anywhere on the overarm. Small arbor supports give additional clearance below the arbor supports when small diameter cutters are being used. Large arbor supports can be positioned at any point on the arbor.



Milling Machine Parts

(9) Size Designation: All milling machines are identified by four basic factors: size, horsepower, model, and type. The size of a milling machine is based on the longitudinal (from left to right) table travel, in inches. Vertical, cross, and longitudinal travel are closely related to the overall capacity. There are six sizes of knee-type milling machines, with each number representing the number of inches of travel. The Standard sizes are, No. 1 — 22 inches; No. 2 — 28 inches; No. 3 — 34 inches; No. 4 — 42 inches; No. 5 — 50 inches; No. 6 — 60 inches.



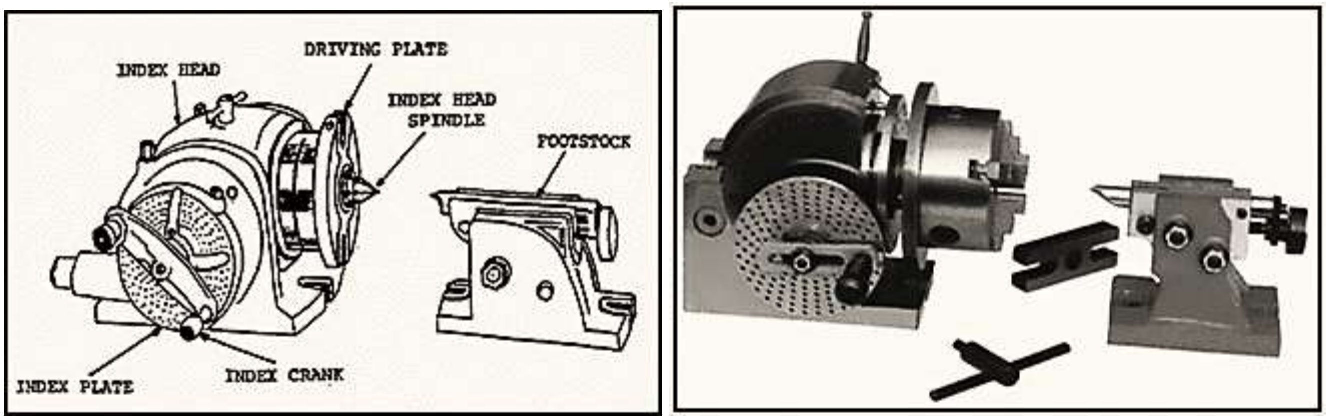
CNC Milling Machine Parts

Milling Machine Accessories:

Collets: Serve to step up or increase the taper sizes so that small-shank tools can be fitted into large spindle recesses. They are similar to drilling machine sockets and sleeves except that their tapers are not alike. Spring collets are used to hold and drive straight-shanked tools, similar to lathe collets. Collets are available in several fractional sizes.

Spindle Adapters: Used to adapt arbors and milling cutters to the standard tapers used for milling machine spindles. With the proper spindle adapters, any tapered or straight shank cutter or arbor can be fitted to any milling machine, if the sizes and tapers are standard.

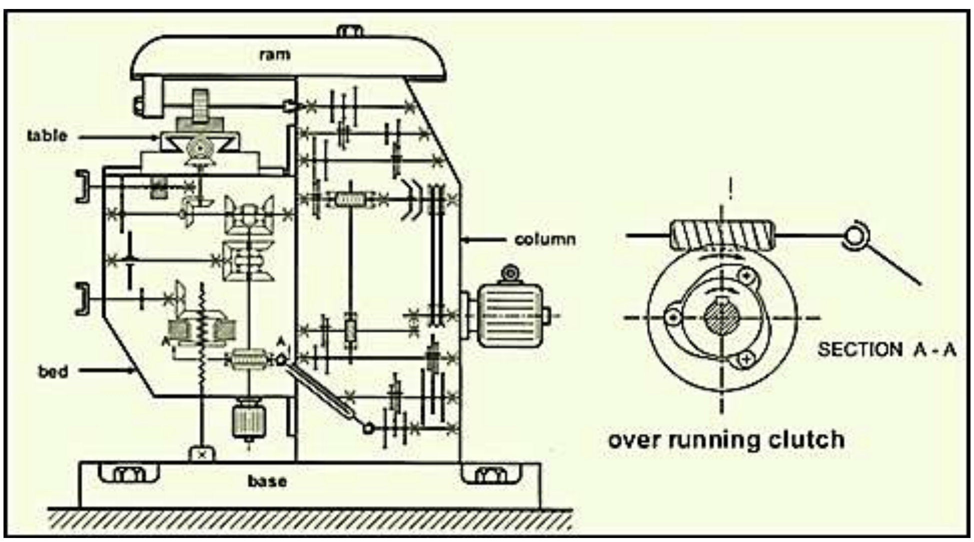
Indexing Fixture: Is an indispensable accessory for the milling machine. Basically, it is a device for mounting workpieces and rotating them a specified amount around the work piece's axis, as from one tooth space to another on a gear or cutter. It consists of an index head, also called a dividing head, and a footstock, similar to the tailstock of a lathe. The index head and the footstock are attached to the worktable of the milling machine by T-slot bolts.



Indexing Fixture

Kinematic System:

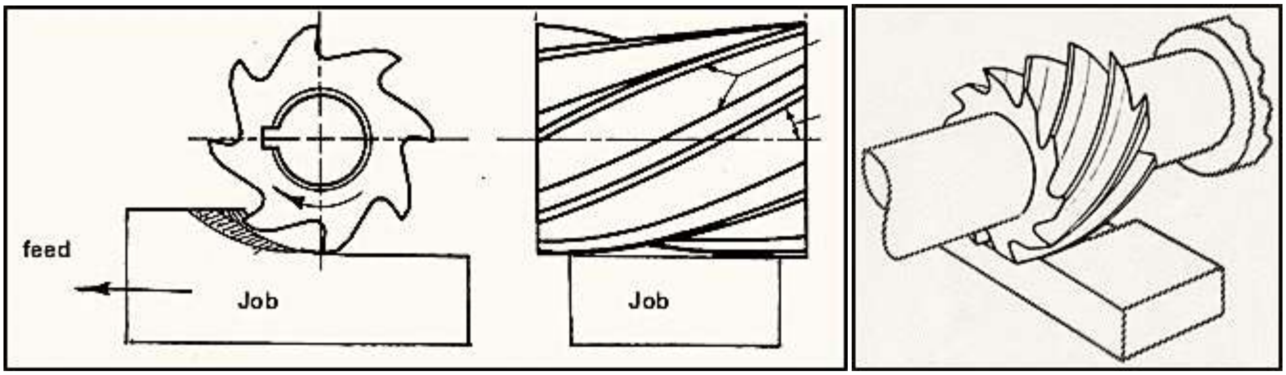
The kinematic system comprising of a number of kinematic chains of several mechanisms enables transmission of motions (and power) from the motor to the cutting tool for its rotation at varying speeds and to the worktable for its slow feed motions along **X, Y and Z** directions. In some milling machines the vertical feed is given to the milling (cutter) head. The more versatile milling machines rotates the worktable and tilting the vertical milling spindle about **X** and / or **Y**-axes, as shown below.



Milling Cutter Types:

Milling machines are mostly general purpose and have wide range of applications requiring various types and size of milling cutters. Intermittent cutting nature and usually complex geometry necessitate making the **milling cutters mostly by HSS,** which is unique for high tensile and transverse rupture strength, fracture toughness and formability almost in all respects, i.e., forging, rolling, powdering, welding, heat treatment, machining (in annealed condition) and grinding. Tougher grade **cemented or carbides** are also used without or with coating, where feasible, for high productivity and product quality.

Slab or Plain Milling Cutters: Are hollow straight cylinders of 40 (1 ½") to 90 mm (3 1/2") outer diameter, having **4 to 16** straight or helical equispaced **flutes or cutting edges,** used in horizontal arbors to machine flat surface, commonly in high-speed carbon steel (HSS), as shown below.

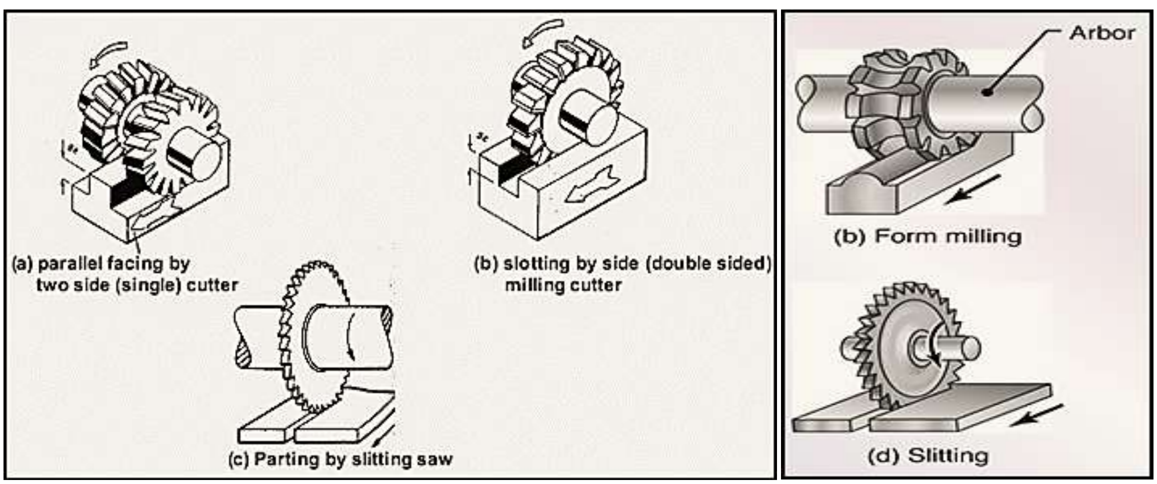


Machining a flat surface by slab milling.

Side Milling Cutters: Each tooth has a peripheral cutting edge and another cutting edge on one face in case of single side cutter and two more cutting edges on both the faces leading to double sided cutter. One sided cutters are used to produce one flat surface or steps comprising two flat surfaces at right angle as shown below. Both sided cutters are used for making rectangular slots bounded by three flat surfaces.

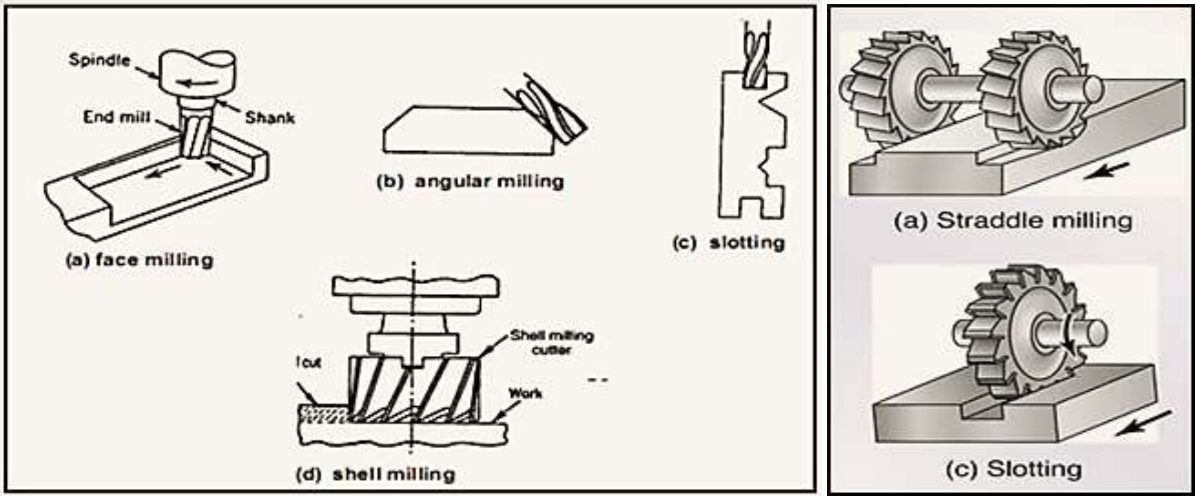
Slot Milling Cutters: Is done by another similar cutter having only one straight peripheral cutting on each tooth. These cutters may be made from a single piece of HSS and its teeth may be of carbide blades brazed on the periphery or clamped type uncoated or coated carbide inserts for high production machining.

Slitting Saw: These milling cutters are very similar to the slotting cutters having only one peripheral cutting edge on each tooth. However, the slitting saws are larger in diameter and thin; possess large number of cutting teeth of small size; are used only for slitting or parting.



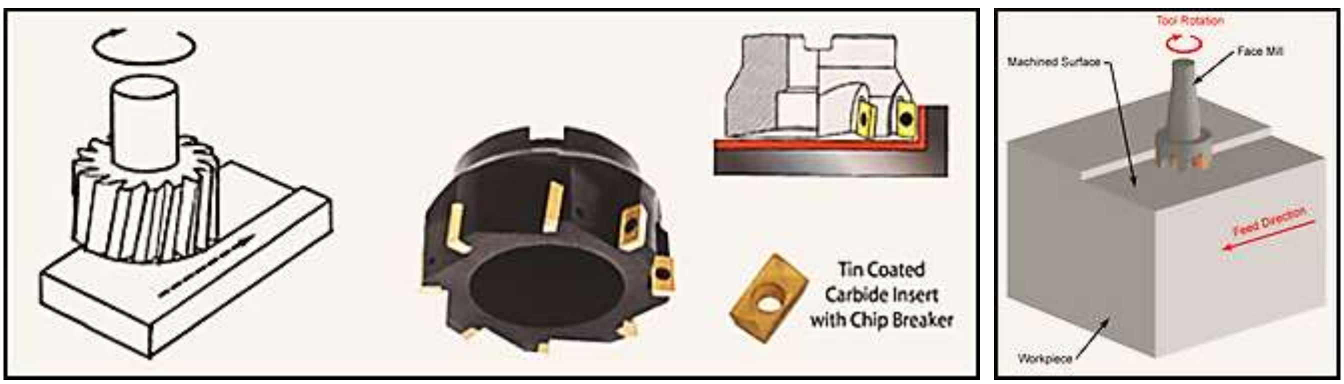
Side milling cutters and slitting saw

End Milling Cutters: The shape and the common applications of end milling cutters (sharpened type) are shown below. Milling cutters are mostly are made of HSS metal tools; 4 to 12 straight or helical teeth on the periphery and face. Diameter ranges from about **1 mm to 40 mm**; very versatile and widely used in vertical spindle type milling machines; end-milling cutters requiring larger diameter are made as a separate cutter body, which is fitted in the spindle through a taper shank arbor as shown in the same figure.



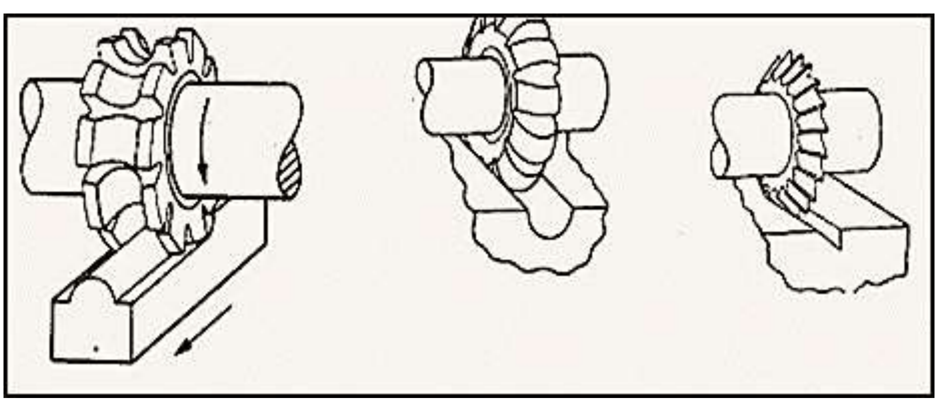
End milling cutters and shell milling

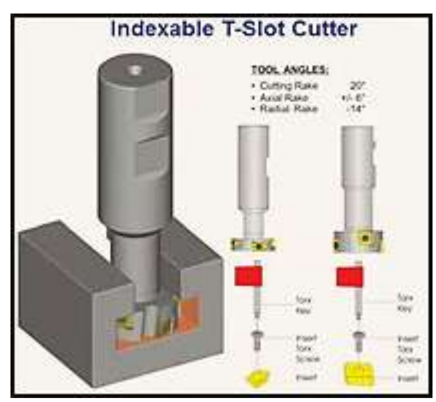
Face Milling Cutters: The shape, geometry and typical use of face milling cutters are usually large in diameter (80 to 800 mm) and heavy as shown below, used only for machining flat surfaces in different orientations. Mounted directly in the vertical and / or horizontal spindles; coated or uncoated carbide inserts are clamped at the outer edge of the carbon steel body as shown; generally used for high production machining of large jobs.



Face milling cutters

Tool Form Cutters: Are used for making grooves or slots as indicated below. Form cutters may be also end mill type like T-slot cutter Are also used widely for cutting slots or flutes; form milling types of different cross section, e.g. the flutes of twist drills (see below), milling cutters, reamers, hobs, taps, short thread milling cutters, etc.


Tool form cutters

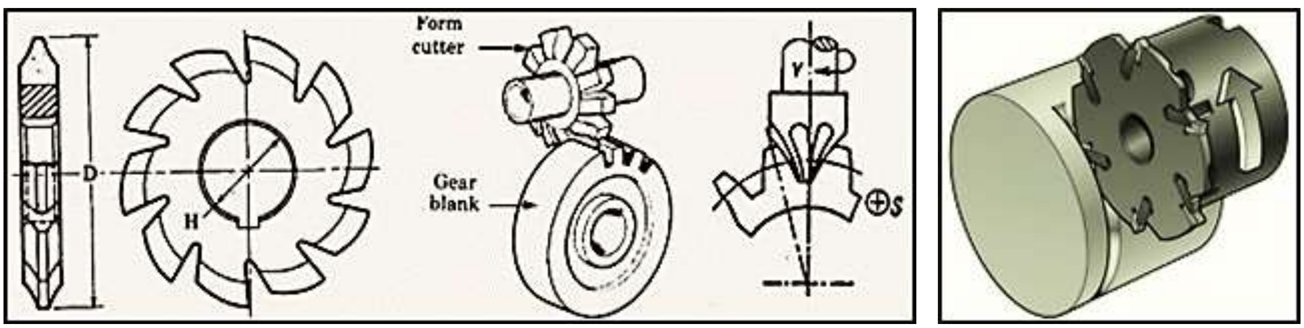


T-slot cutter

Spline Shaft Cutters: These disc type HSS form relieved cutters are used for cutting the slots of external spline shafts having 4 to 8 straight axial teeth. Flute is a space between the cutting teeth providing chip space and regrinding. Sometimes are referred to as "teeth" or "gullet". The number on an end mill determines the feed rate.

Spline shaft cutter and form milling cutter with drill flutes

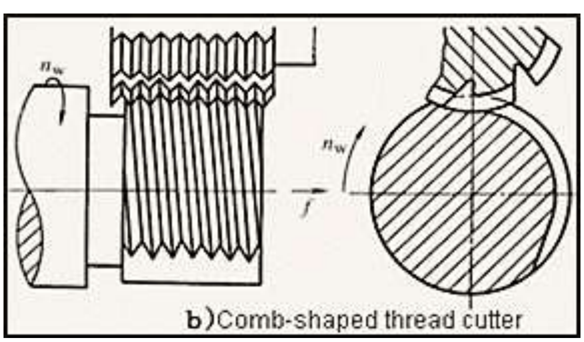
Gear Teeth Milling Cutters: Are made of HSS and available mostly in disc form like slot milling cutters and also in the form of end mill for producing teeth of large module gears. Such form relieved cutters can be used for producing teeth of straight and helical toothed external spur gears and worm wheels as well as straight toothed bevel gears.



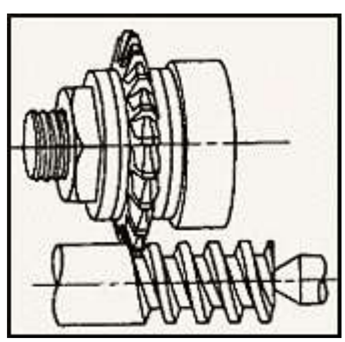
Gear milling cutters and their use

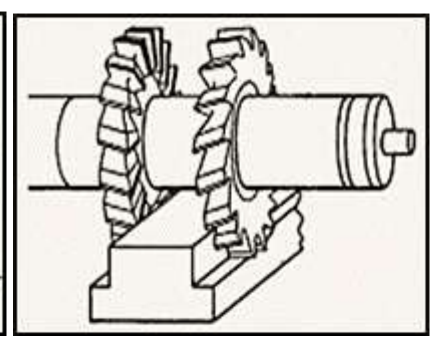
Thread Milling Cutters: Solid grooves with equi-spaced cutters, in HSS or carbide, are used in automatic single purpose milling machines for cutting the threads in large lot production of screws, bolts etc. The tool as shown below cuts both internal and external threads.

Straddle Milling Cutters: For faster and accurate machining two parallel vertical surfaces at a definite distance, two separate side-milling cutters are mounted at appropriate distance on the horizontal milling arbor as shown below.



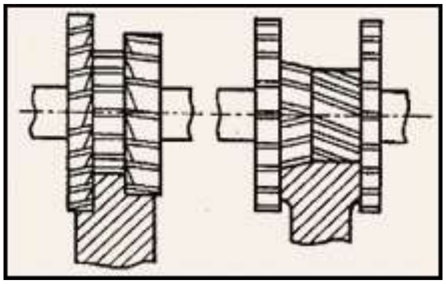
Short thread milling


Long thread milling

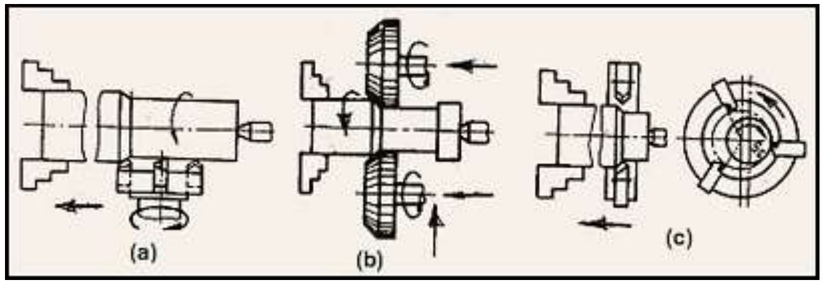

Straddle milling cutter

Gang Milling Cutters: For quick production comprising a number of parallel flat or curved surfaces a proper combination of several cutters are mounted tightly on the same horizontal milling arbor as indicated below.

Rotary Milling Cutters: For large heavy and odd shaped jobs, its speed (rpm) is essentially kept low. For enhancing productivity and better cutting fluid action rotary tools like milling cutters are used as shown below.

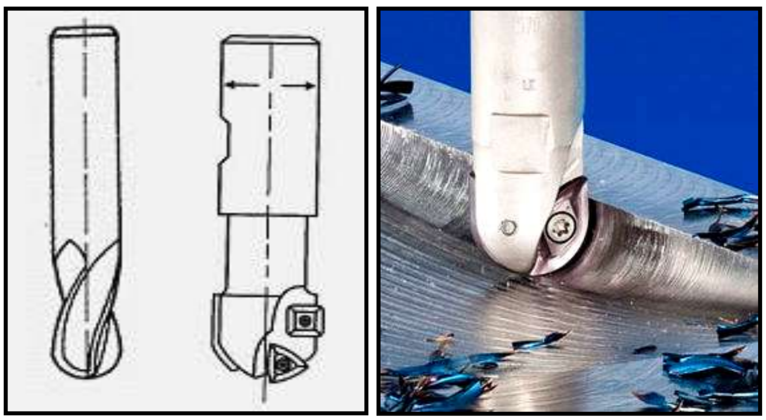


Gang milling cutters



Rotary milling cutters

Ball-Nose End Milling Cutters: Small HSS end mill is often used in CNC milling machines for machining free form 3D or 2-D contoured surfaces. Ball nose end milling cutters may be made of HSS, solid carbide or steel body with coated or uncoated carbide inserts clamped at its end as can be seen in the figure, commonly employed for many machining works like cam milling, keyway cutting, hob cutting, and so on.



Ball nose end cutters

Milling Calculations:

Mill Cutter Speed: Uses the same formula as drilling, usually measured in terms of the rate at which the outside or periphery of the tool moves in relation to the work. The term for this velocity is also "**surface feet per minute**", abbreviated as **sfm**. As referred above, every tool manufacturer has a recommended table of **sfm values** for their tools. The peripheral and rotational velocities of the tool are related as shown in the following equation:

$$1)\quad V = \pi * d * N$$

V = Recommended peripheral velocity for the tool being used
d = Diameter of the mill cutter
N = Rotational velocity of the tool.

Peripheral Rotation: Is shown in the following equation:

$$2)\quad N\,[rpm] = \frac{12\,[in/ft] * V\,[sfm]}{(\pi * d\,[in/rev])} = \frac{Cs * 4}{d} =$$

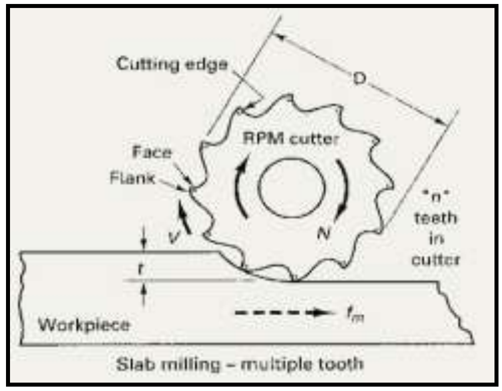
Where:

Cs = Cutting speed (ft/min);
4 = Constant in calculation for RPM, using velocity in ft/min (except metric);
d = Diameter of the mill cutter (in inches).

Obs.: The peripheral velocity is also expressed in units of **feet/min** (**or m/min**) and the tool diameter is typically measured in units of inches (or units of mm), the spindle or tool velocity, ***N*** may be found in the following manner:

Example: A mill facing cutter **2 1/2-inch** (2.500-inch) is necessary to face a common aluminum block using 200 **ft/min**. The formula would be as follows:

$$N = \frac{200 \times 4}{2.500} = \frac{800}{2.500} = 320 \text{ RPM}$$



In **metric system** (**m/min)** to find the RPM:

$$\textbf{3)}\quad N = \frac{Cs\ (m/min) \times 320}{D\ (mm)}$$

Where:

Cs = Recommended cutting speed (in meters per minute or m/min)
320 = A constant for all metric RPM calculations (using m/min)
D = Diameter of the mill cutter in millimeters (or mm).

Example: A mill face cutter **50.0 mm** is necessary to drill a medium-carbon steel, with a recommended cutting speed of **20 meters per minute**. The formula would be set up as follows:

$$\textbf{4)}\quad N = \frac{Cs\ (m/min) \times 320}{D\ (mm)}$$

$$N = \frac{20\ (m/min) \times 320}{50\ (mm)} = 128 \text{ RPM}$$

To calculate the cutting speed or feedrate, once the rpm has been computed, use the following formula:

$$\textbf{5)}\quad Cs = N * f_t * n$$

Where:

Cs = Cutting speed or feedrate [in/min];
N = spindle speed [rpm];
ft = feed per tooth [in/tooth];
n = number of teeth considered per revolution of the cutter;

Example: Calculate the speed and feed for a **1.0" diameter**, **4 teeth** mill cutter **flute-type HSS**, using a manual milling machine to face an aluminum block. First, see the recommended peripheral velocity on table above, keeping the values - ***Cs,V*** = ~ 250 ft/min.

N [rpm] = 12 * V / (π * D)
N = 12 in/ft * 250 ft/min / (π * 1.0)
N **= 950 rpm**

Calculate the **cutting speed or feedrate**, considering an appropriate initial **feed per tooth** (chip load) as:

ft = ~ 0.008 in./tooth (see table above):

Cs[in/min] = N [rpm] x ft [in/tooth] x n
Cs = 950 rpm * 0.008 in/tooth * 4 teeth/rev

***Cs* = 30 in/min**

Obs.: When applying oil manually, scale the feedback to 60%, so:

Cs** = 30 x 0.6 =* ***18 in/min (final answer).

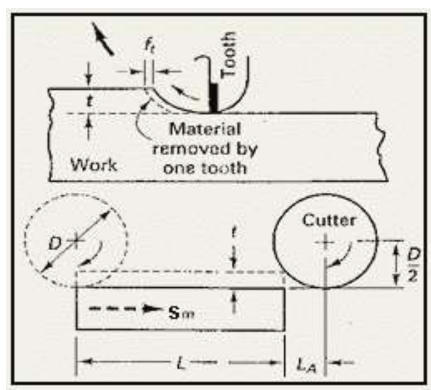
Machining time in milling operations:

There are different types of milling operations done by different types of milling cutters:

- Plain milling by slab milling cutter mounted on arbor;
- End milling by solid but small end mill cutters being mounted in the spindle through collet;
- Face milling by large face milling cutters being directly fitted in the spindle.

The machining time or time of cutting, **Tc** for a plain milling flat surfaces can be determined as:

a) **Tc = Lc / Sm**



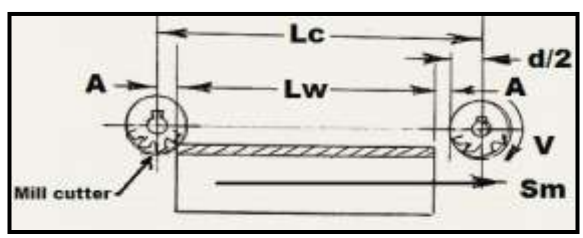
Where:

Lc = Total length of piece [in; mm];
Sm = Mill table feed [in/min; mm/min];

Consider the length **Lc** as:

b) **Lc = Lw + 2A + d/2**

Where:



Lw = Length of work piece (see sketch below) [in, mm];
A = Cutter approach (0.1875 in to 0.375 in; 5 to 10 mm,);
D = Cutter diameter (in, mm);

Consider the table feed **Sm** as:

c) **Sm = ft * n * N**

Where:

ft = feed per tooth [ft/tooth, mm/tooth);
n = number of teeth;
N = Rotational speed, rpm.

Since milling is an intermittent cutting process, *V*, Cs, should be taken lower (20 ~ 40%) of the recommended for continuous machining like turning. Also *ft* should be taken reasonably low (within 0.1 to 0.5 mm) (0.004 to 0.02 in) depending upon the tooth–size, work material and surface finish desired.

Formulas for Speed, Feed and Power Calculation:

To find	Having	Formula
Peripheral cutting speed - V	Cutter diameter , d Rotational speed, N	*V,Cs = π.d.N*
Rotational speed - N	Cutter diameter, d Peripheral cutting speed, V	*N = (Cs * 4) /d (in)* *N = Cs * 320/d (mm)* *N = (12 * V) /π * d (in)* *N = 1000 * V/π * d (mm)*
Cutting speed - Cs	Rotational speed, N Cutter diameter , d	*Cs = N * d/4(in)* *Cs = N * d/320 (mm)*
Feed per tooth - ft	Machine feed rate, Fr Rotational speed, N Number of teeth, n	*ft = Fr/N * n*
Machine feed rate - Fr	Feed per tooth, ft Rotational speed, N Number of teeth, n	*Fr = N * ft * n*
Feed per revolution - f	Machine feed rate, Fr Rotational speed, N	*f = Fr/N*
Cutting power input - P	Width of cut, W Depth of cut, h Machine feed rate, F Workpiece material power constant, C	*P = (W *h *F)/C*

Work Material	C (Constant)
Magnesium	4.0
Aluminum	4.0
Copper	2.0
Brass	2.5
Bronze	2.0
Cast Iron --	
Nodular	1.2
Malleable	1.0
Chilled	0.6
Carbon Steel --	
Up to 150 Brinell	0.7
300 Brinell	0.6
400 Brinell	0.5
500 Brinell	0.4
Stainless Steel -	
Ferritic	0.8
Austenitic	0.8
Martensitic	0.6
Others	0.6

Note: The formulae below are also used:

SFM = 0.262 x d x N - (where, SFM - Surface Feet per Minute = Cs)
RPM = (3.82 x SFM) / d – (where, RPM = N) - **3.82** is a constant derived from 12/pi - feet to diameter in inches.
IPR = IPM / RPM or CHIP LOAD x F – (where, IPR – Inches per Revolution);
IPM = RPM. x IPR – (where, IPM – Inches per Minute)
CHIP LOAD = IPM / (RPM x F) or IPR / F – (where, CHIP LOAD - material removal rate; F – Feed rate)

Example: Determine the machining time, *T* using a plain milling with a rectangular surface of length 100 mm and width 50 mm by a helical fluted plain HSS milling cutter of diameter 60 mm and 6 teeth. Assume *A = 5 mm*, *V = 40 m/min* and *ft = 0.1 mm/tooth*.

Tc = Lc / Sm
Lc = Lw + 2A + d/2 = 100 + (2*5) + 30 = **140 mm**
N = 1000 * V /π * d = 1000 * 40 / π * 60 = 212 rpm – use **200 rpm**
Sm = ft * n * N = 0.1 * 6 * 200 = **120 mm/min**
Tc = Lc / Sm = 140 / 120 = **1.17 min.**

Planing Machines & Planers:

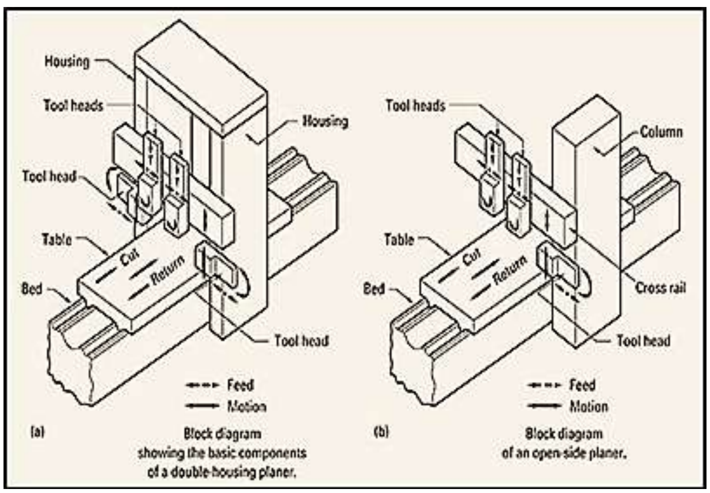
Planing Machines: Planing machines are also used for producing flat surfaces in different planes. James Coulter and Herbert Harpin of West Yorkshire achieved the application for a patent. In 1872, they state, “our invention relates to improvements effected by us in the ordinary planing machines for iron in adapting it for cutting stone”. In other words, they took the established process of planing iron and modified it to work successfully on stones. However, the major differences between planing machines from shaping machines are:

- ✓ Though in principle both shaping and planing machines produce flat surface in the same way by the combined actions of the Generatrix and Directrix but in planing machine, instead of the tool, the workpiece reciprocates giving the fast cutting motion and instead of the job, the tool(s) is given the slow feed motion(s).

- ✓ Compared to shaping machines, planing machines are much larger and more rugged and generally used for large jobs with longer stroke length and heavy cuts. In planing machine, the workpiece is mounted on the reciprocating table and the tool is mounted on the horizontal rail which, again, can move vertically up and down along the vertical rails.

- ✓ Planing machines are more productive (than shaping machines) for longer and faster stroke, heavy cuts (high feed and depth of cut) possible and simultaneous use of a number of tools. As in shaping machines, in planing machines:

- The length and position of stroke can be adjusted
- Only single point tools are used
- The quick return persists
- Form tools are often used for machining grooves of curved section
- Both shaping and planing machines can also produce large curved surfaces by using suitable attachments.



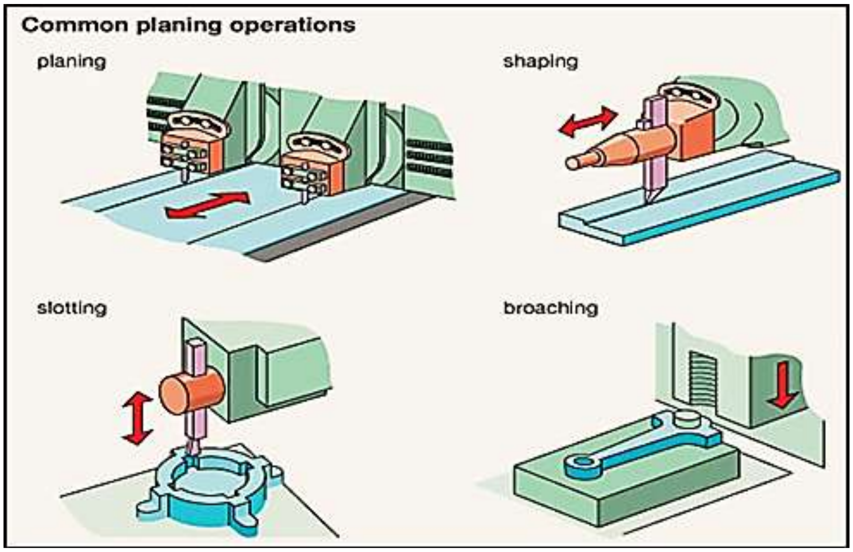
The simple kinematic system of the planing machine enables transmission and transformation of rotation of the main motor into reciprocating motion of the large work table and the slow transverse feed motions (horizontal and vertical) of the tools. The reciprocation of the table, which imparts cutting to the job, is attained by rack-pinion mechanism.

The rack is fitted with the table at its bottom surface and the pinion is fitted on the output shaft of the speed gear box which not only enables change in the number of stroke per minute but also quick return of the table. The blocks holding the cutting tools are moved horizontally along the rail by screw-nut system and the rail is again moved up and down by another screw nut pair as indicated above.

The basic principles of machining by relative tool-work motions are quite similar in shaping machine and planing machine. The fast straight path cutting motion is provided by reciprocation of the tool or job and the slow, intermittent transverse feed motions are imparted to the job or tool. In respect of machining applications also these two machine tools are very close.

All the operations done in shaping machine can be done in planing machine. But large size and stroke length and higher rigidity enable the planing machines do more heavy duty work on large jobs and their long surfaces. Simultaneous use of number of tools further enhances the production capacity of planing machines.

Planing metal with a machine tool is similar to planing wood with a hand plane. The planer produces a flat surface by its cutting action. A single-edged tool is held in a toolhead on a rigid cross member called a rail. The workpiece is mounted on a table that is supported by tracks in a heavy bed, as shown below:



Planers: A planer is a type of metalworking machine tool that is analogous to a shaper, but larger, and with the entire workpiece moving beneath the cutter, instead of the cutter moving above a stationary workpiece. The work table is moved back and forth on the bed beneath the cutting head either by mechanical means, such as a rack and pinion gear, or by a hydraulic cylinder. There are two types of planers for metal machining: double-housing and open-side.

- Planing is used for large workpieces too big for shapers;
- Planing machines have largely been replaced by planing mills;
- In planing, large workpieces and their support tables are slowly moved against the tool head.

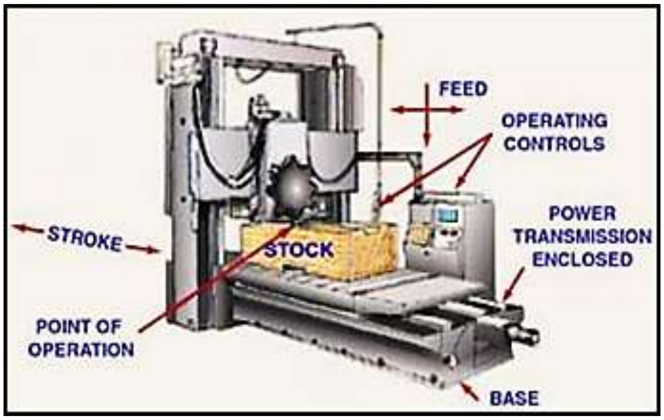
Planer Tools: As a planer tool has to take up heavy cut and coarse feed during a long cutting stroke, the tools are made heavier and larger in cross section. Metal planers can vary in size from a table size of 30" × 72" to 20' × 62', and in weight from around 20,000 lbs to over 1,000,000 lbs. A planer tool may also be classified as right hand or left hand and roughing or finishing. Shaper and planer are both reciprocating machine tools and both of them are primarily intended to produce flat surface, but they differ very much in construction, operation and use.

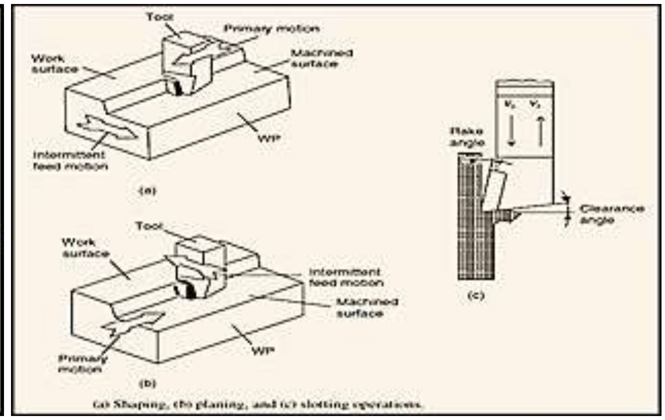
Cutting Calculations:

- **Feed:** Distance of the tool head at each cutting stroke, expressed in inches (mm) per double stroke.
- **Cutting Depth:** Thickness of metal removed in one cut, and measured by the perpendicular distance between the machined and no machined surface expressed in inches (mm).
- **Machining Time:** The machining time required for every complete cut for a complete operation.
- **Cutting Speed:** Rate when metal is removed during the forward cutting stroke. Using this principle, we define that the velocity of return stroke is more than the cutting stroke:

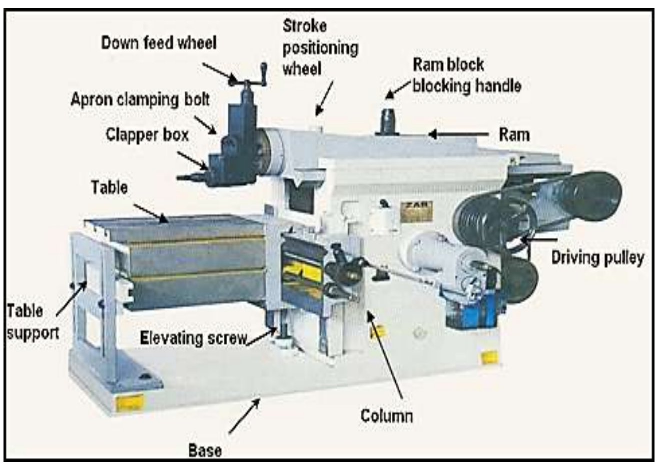
$$Cs = \frac{n \times L \times (1 + Q)}{1000} \text{ (m/min)} \text{ – or – } \frac{n \times L \times (1 + Q)}{12} \text{ (ft/min)}$$ – (planing, shaping and slotting calculations).

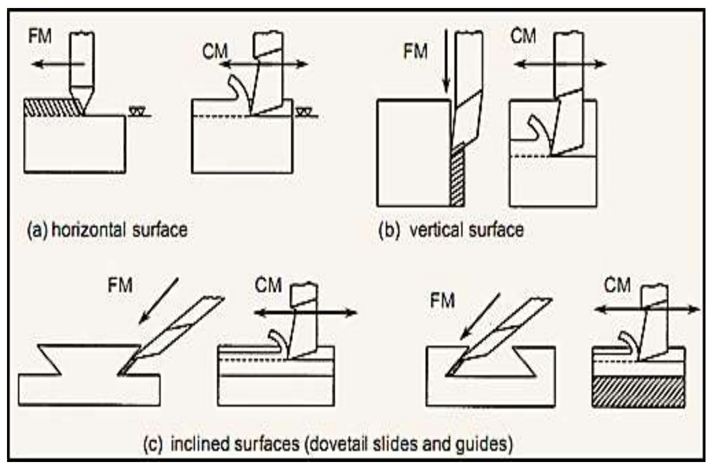
Cs = Feed;
n = Number of strokes per minutes;
L = Stroke length, (in) (mm);
Q = Ratio of cutting speed motion to return (usually less than 1.0).





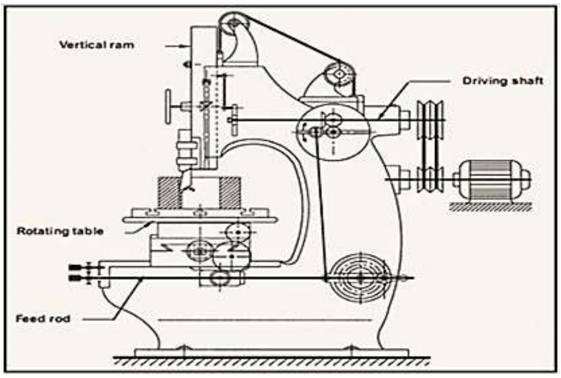
Shaping Machines: As referred before, **shaper** and **planer** are both reciprocating machine tools to produce flat surfaces, but they differ very much in construction, operation and use. The cutting motion provided by the linear forward motion of the reciprocating tool and the intermittent feed motion provided by the slow transverse motion of the job along with the bed, result in producing a flat surface by gradual removal of excess material layer by layer in the form of chips. Straight grooves of various curved sections are also made in shaping machines by using specific form tools. The single point straight or form tool is clamped in the vertical slide which is mounted at the front face of the reciprocating ram whereas the workpiece is directly or indirectly through a vice is mounted on the bed.

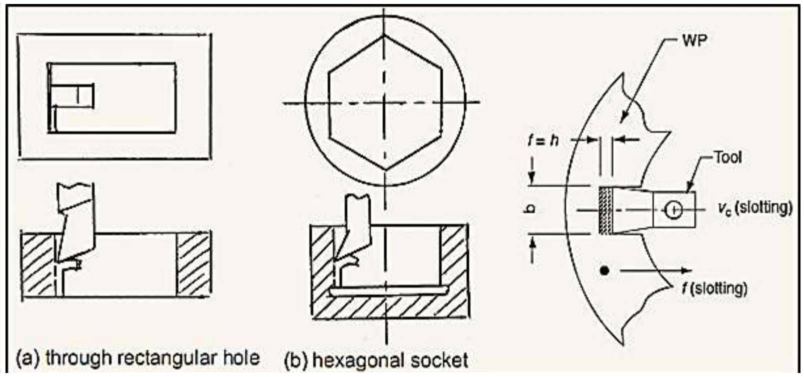




Slotting Machines: Slotting machines **are technically vertical shapers.** The vertical slide holding the cutting tool is reciprocated by a crank and connecting rod mechanism. The workpiece to be machined is mounted directly or in a vise on the worktable. The intermittent rotation of the feed rod is transmitted from the lead-screws for the two linear feeds and to the worm wheel for rotating the worktable. Slotting machines can also perform machining operations of keyways, cutting of internal and external teeth on large gears. The stroke length of the ram varies from 300 to 1800 mm, using a machining process at ±15° from vertical. The usual applications of slotting machines are:

- Internal flat surfaces and curved surface of circular section;
- Enlargement and / or finishing non-circular holes bounded by a number of flat surfaces;
- Blind geometrical holes like hexagonal socket;
- Internal grooves and slots of rectangular and curved sections;
- Internal keyways and splines, straight tooth of internal spur gears;
- Internal oil grooves, etc., which are not possible in shaping machines.





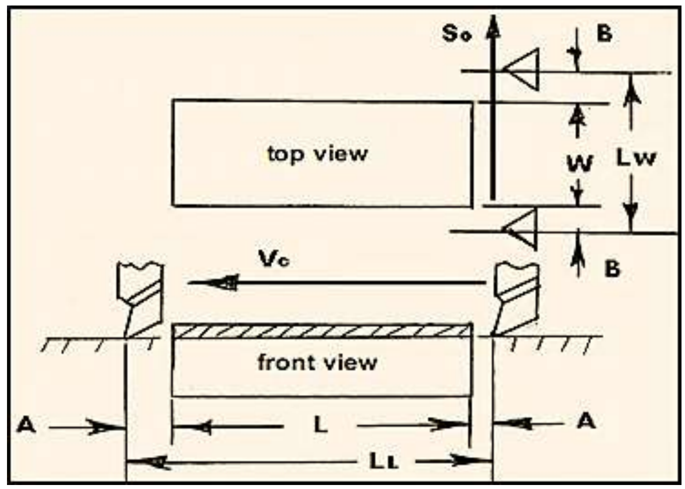
Planing, Shaping and Slotting Calculations: Planing machine calculations are also almost determined in the same as in plain milling machines. The only difference is that in planing machine, the cutting strokes and feed travels are imparted to the workpiece and the tool respectively, just opposite to that of shaping machine. Though both shaping and planing are reciprocating type, planing machine may allow a higher Vc.

Cutting velocity has to be determined by:

$$V_C = \frac{N_s}{1000}\left[L_L(1+Q)\right] \text{ m/min}$$

Therefore:

$$N_s = (1000V_C)/\left[L_L(1+Q)\right]$$



Where,

Vc = Cutting velocity, m/min;
Ns = Number of strokes per min;
L = Length of the workpiece;
LL = Length of the stroke, mm = L + 2A (see sketch);
W = Width of workpiece;
Lw = Total width of workpiece = W + 2B (see sketch);
A, B = Approach and over run;
Q = Time of return stroke divided by time of cutting stroke.

Example: Determine the number of strokes of a planing machine for a workpiece with the following data:

L = 100 mm, A = 5,
W = 60, B = 2
Q = Time of return stroke divided by time of cutting stroke = 2/3
Vc = 40 m/min
Feed = 0.2 mm/stroke

$$N_s = (1000V_C)/\left[L_L(1+Q)\right]$$

Ns = (1000x40)/[(100+5+5)(1+2/3)] = **218 strokes/min**.

Using the formulae above, the **total machining time**, **Tc**, can be determined form the expression:

$$T_C = \frac{L_w}{N_s s_0} \text{ min}$$

Tc = Total machining time, minutes;
Lw = Total width of workpiece = W + 2B
Ns = Number of strokes/minute
Feed = mm/stroke.

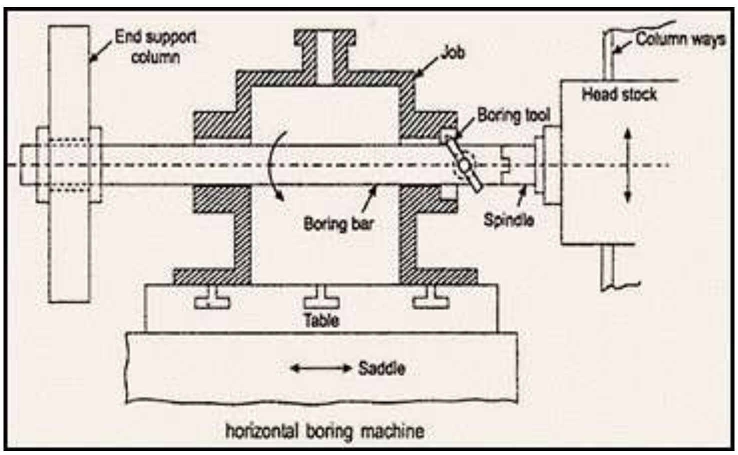
Example: Determine the total machining time for same workpiece indicated on example above, using Ns = 200:

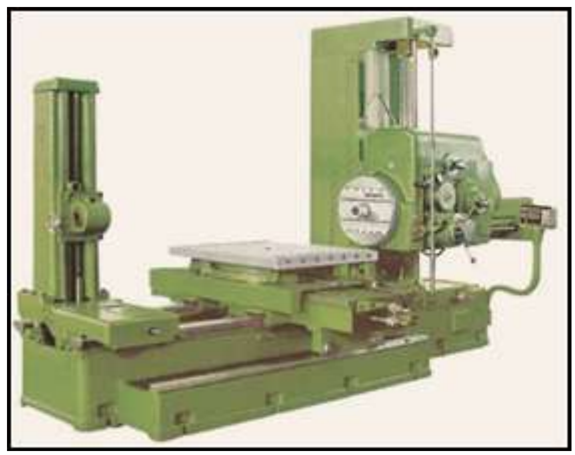
Tc = Total machining time, minutes;
Lw = 60 + 2 x 2
Ns = Number of strokes/minute = 200
Feed = 0.2 mm/stroke.

Tc = $\frac{(60 + 2 + 2)}{(0.2 \times 200)}$ = **1.6 minutes**

Boring Machines:

Boring is the process of **enlarging a hole** that has already been drilled (or cast), by means of a single-point cutting tool (or a boring head containing several tools), as in boring a gun barrel or an engine cylinder. Boring is also used to achieve greater accuracy of the diameter of a hole, cutting a tapered hole, or can be viewed as the internal-diameter counterpart to turning, which cuts external diameters.





Boring brings the hole to the proper size and finish, and achieves three things:

- A drill or reamer can only be used if the desired size is "standard" or if special tools are ground; the boring tool can work to any diameter and it will give the required finish by adjusting speed, feed and nose radius; precision holes can be bored using micro adjustable boring bars.

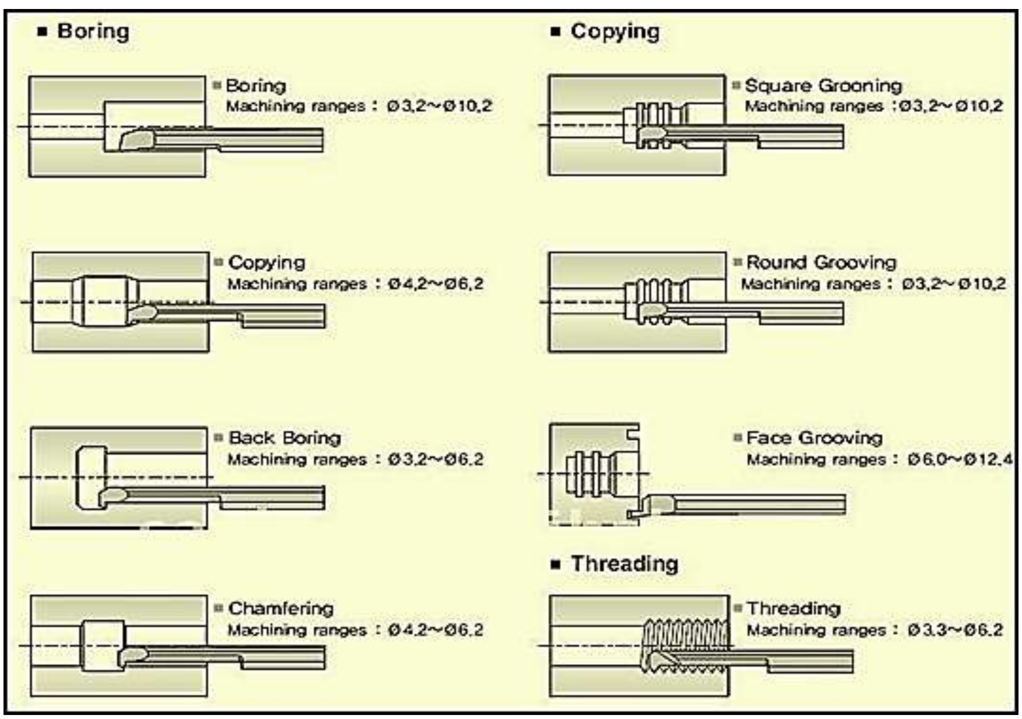
Types of Boring Machines: There are three **main types - table, planer and floor**. The table type is the most common and more versatile, known as universal type. Boring machines, like most other machine tools, are also classified as **horizontal or vertical types**. Boring operations can be also performed on other than boring machines, such as lathes, milling machines and machining centers.

Floor-Type Horizontal Boring Machine (HBM): The floor type HBM is used for especially tall or long workpieces and to handle medium to very large-sized parts. Typically, the tool head and part move in three linear axes. The **Z-axis is the main axis** that drives the workpiece towards the work spindle, followed by a **cross traversing X-axis** and a **vertically traversing Y-axis**. If a rotary table is incorporated, the work spindle is usually referred to as the **C-axis** and its centerline is the **B-axis**. The table is separate from the boring machine, fastened to the floor or bolted to the runway. The entire column and column base move left and right (the X-axis) along special ways on the runway.

Vertical Boring Machined (VBM): A vertical boring machine is like a **lathe** with the headstock resting on the floor for machining over **24 inches in diameter**. These machines can make only round cuts plus facing and contouring cuts. Today's VBMs are often **listed as turning, facing and boring machines**.

Jig Borers: The workpiece rotates around a vertical axis, while boring bar/head moves linearly, (as a vertical lathe). Charles De Vlieg developed a highly precise model, which he called a JIGMIL. The accuracy of this machine convinced the USAF to accept John Parson's idea for CNC machine tools.

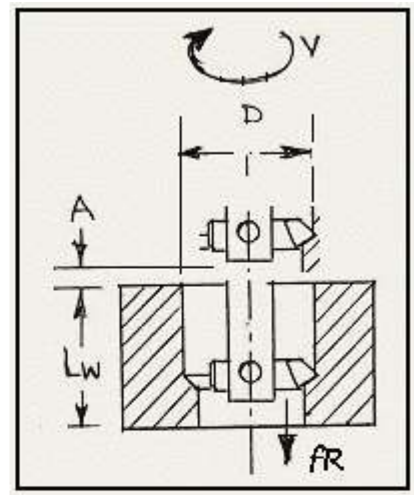
Cutting Tools and Workpieces: Cutting tools are also common single point, HSS or carbides. A tapered hole can also be made by swiveling the head. Workpieces are commonly **3 ft to 13 ft** (1 to 4 meters) in diameter, but can be as large as **20 m** (66 ft). Power requirements can be as much as 200 horsepower (150 kW). Tools are made for the high-tech, and small component materials to various applications (fittings, valves, medical parts, automotive, semiconductor and machinery parts) and other machining types (Turning, boring, grooving, threading etc.). **Lineboring:** Implies the conventional machines. **Backboring:** Is the process of reaching through an existing hole and then boring on the "back" side of the workpiece (relative to the machine headstock).



Boring Calculations:

In boring, facing, cutting and cut off, the cutting speed for a given RPM decreases as the tool progresses toward the center of the piece being cut. The required **N** is calculated, as descript before, using the outside (largest) diameter of the part for a given **V**. See manufacturing tables according to material classes and used tools (HSS or Carbide tools).

A.) Machine Speed:



$$N = \frac{k * V}{\pi * D_1}$$

V = **Surface feet per minute**, D1 in inches: **k = 12;**
V = **Millimeters per minute**, D1 in mm: **k = 320;**
N = Tool rotation (rpm);
$\mathbf{D_1}$ = Finished (larger) diameter (see sketch).

B.) Cutting Time:

$$Ct = \frac{(Lw + A)}{f_R * N} \text{ - or - } Ct = \frac{\pi * D_1 * (L w + A)}{12 * V * f_R}$$

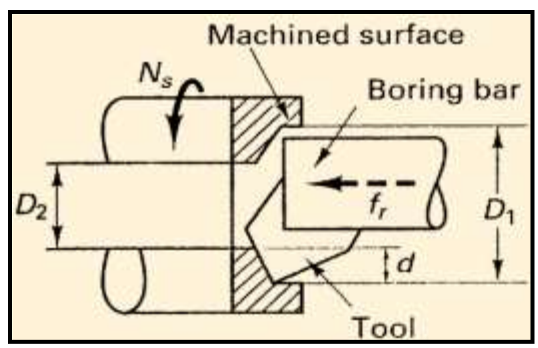
Where:

Lw = Length of hole (in) or (mm) – [* attention to conversions feet, inches, meters or millimeters].
A = Approach and over run = $D_1/2$;
fr = Feedrate (in/min) or (mm/min);
$\mathbf{f_R}$ = Feed (in/rev) (mm/rev).

C.) Material Removal Rate:

$$MRR = \frac{\text{Volume Removed}}{\text{Cutting Time}} (ft^3/min)\ (mm^3/min) = \frac{\pi * L * D^2 * fR * N}{4 * L} = \frac{0.7854\ (D^2 * fR * N)}{4}$$

Where:



$\mathbf{D_1}$ = Finished diameter (in) (mm);
$\mathbf{D_2}$ = Initial (smaller) diameter;
$\mathbf{f_R}$ = Feedrate (ft/rev) (in/rev) (mm/rev);
t = Width of the cutting tool (in) (mm);
N = Tool rotation (rpm)
L = $\frac{D_1}{2}$ for solids - $\frac{D_2 - D_1}{2}$ for tubular workpieces

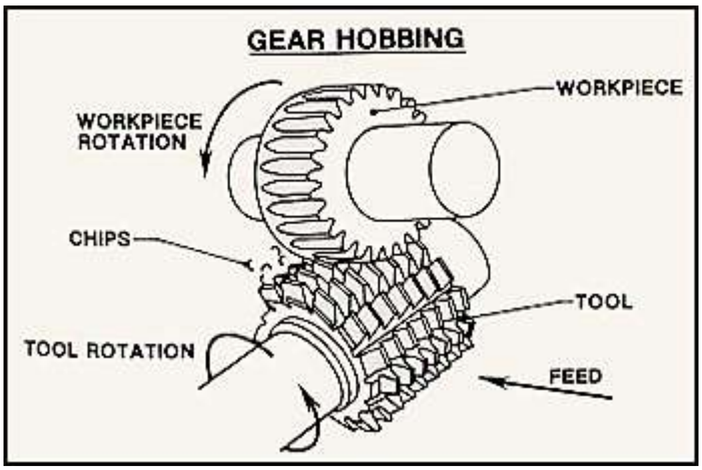
Cutting speeds in m/min and ft/min for HSS tools

MATERIAL	TURNING AND BORING				THREADING	
	ROUGH CUT		FINISH CUT			
	m/min	ft./min	m/min	ft./min	m/min	ft./min
MACHINE STEEL	27	90	30	100	11	35
TOOL STEEL	21	70	27	90	9	30
CAST IRON	18	60	24	80	8	25
BRONZE	27	90	30	100	8	25
ALUMINUM	61	200	93	300	18	60

Hobbing & Gear Shaping Machines:

Hobbing is a machining process for **making gears, splines and sprockets** on a hobbing machine, made with a special type of milling machine. The **teeth or splines** are progressively cut into the workpiece by a series of cuts made by a cutting tool called a **hob**. Each gear hobbing machine typically consists of a chuck and tailstock, to hold the workpiece or a spindle, a spindle on which the hob is mounted, and a drive motor. Modern hobbing machines are fully automated machines that come in many sizes, because they need to be able to produce anything from tiny instrument gears up to 10 ft (3.0 m) diameter marine gears.

The first hobbing machine was patented in Chemnitz in the year 1900. Over 100 years of hobbing machine know how have in fact withstood the stormy times of German history, enabling MAG today to bring innovative and reliable hobbing machine technology. Satisfied customers all over the world prefer the MAG Modul Gear Technology, which is always aware of their commitment and responsibility to those that trusts in high quality. The company in Chemnitz became part of the MAG Group at the end of 2010. The Chemnitz operations can rightfully be termed as the cradle of gear hobbing, with the gear hobbing process invented there in 1897.



Common Gear Generation Types: The gear quality is performed, through **three** manufacturing stages, i.e. **the rough cutting, the heat treatment and the finishing process**. One of the most adopted methods in gear finishing is the **gear skiving or hard hobbing**. As every cutting process based on the rolling principle, **gear skiving** is an exceptional multi-parametric which can be fully optimized. This research illustrates an involved algorithm that simulates rigorously the skiving process and yields data, such as the dimensions of the non-deformed chips and consequently the cutting force components.

1. Face Gear: The face gear is a circular disc with a ring of teeth cut in its side face; hence, the name face gear. Tooth elements are tapered towards its center. The mate is an ordinary spur gear. It offers no advantages over the standard bevel gear, except that it can be fabricated on an ordinary shaper gear-generating machine.

2. Double Enveloping Worm Gear: This worm set uses a special worm shape in that it partially envelops the worm gear as viewed in the direction of the worm gear axis. Its big advantage over the standard worm is much higher load capacity. However, the worm gear is very complicated to design and produce, and sources for manufacture are few.

3. Hypoid Gear: Is a bevel gear, which was originated as a special development for the automobile industry. This permitted the drive to the rear axle to be nonintersecting, and thus allowed the auto body to be lowered. However, it is complicated to design and is the most difficult to produce on a bevel gear generator.

Details of Involute Gearing: Pressure Angle: The pressure angle is defined as the angle between the line-of-action (common tangent to the base circles and a perpendicular to the line-of-centers. From the geometry of these figures, it is obvious that the pressure angle varies (slightly) as the center distance of a gear pair is altered. The base circle is related to the pressure angle and pitch diameter (DP) by the equation:

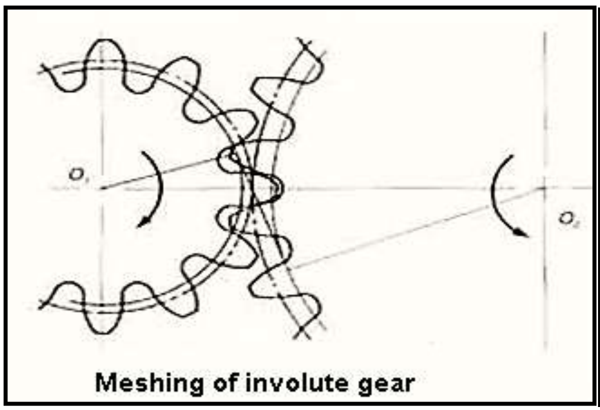
DP = d cos x α – Where, d and α are standard values, or alternately.

DP = d' cos x α' – Where, d' and α' are the exact operating values.

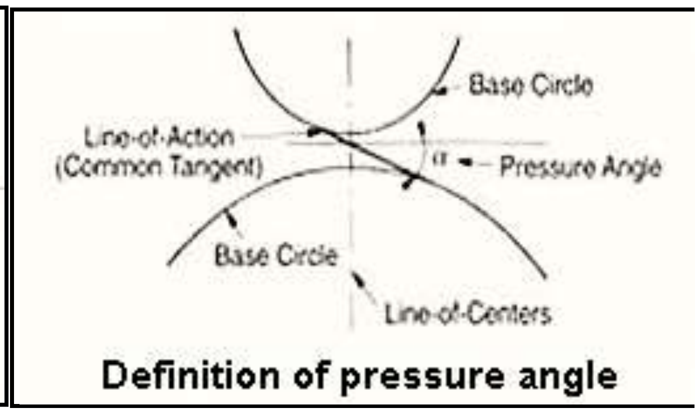
A pair of gears can only mesh correctly if the **pitches** and the pressure angles are the same. Pitch comparison can be **module m**, **circular p**, **base P_b.**

$$P_b = \pi\, m \cos \alpha$$

Proper Meshing and Contact Ratio: The contact point of the two involutes, as figure below shows. The common tangent of the two base circles as rotation occurs. The common tangent is called **line-of-contact**. The basic formula shows that the larger the **pressure angles** the **smaller** the base circle. Then, for standard gears, **14.5º** pressure angle gears have base circles much nearer to the roots of teeth than **20º** gears. **Pressure angles**: when are different, the base pitches cannot be identical. The length of the line-of-action is shown below.

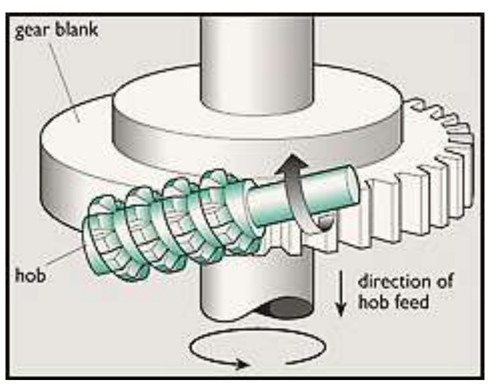


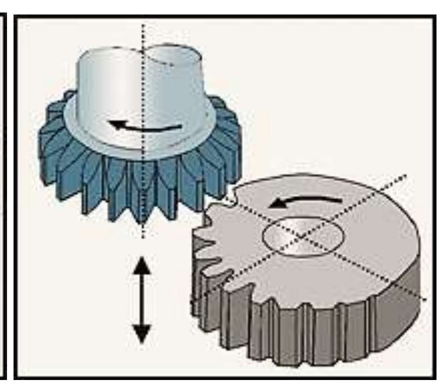
Meshing of involute gear



Definition of pressure angle

Gear Shaping Machines: A gear shaper is a machine tool for **cutting the teeth** of **internal** or **external** gears. The name shaper relates to the fact that the cutter engages the part on the forward stroke and pulls away from the part on the return stroke, just like the clapper box on a planer shaper. The cutting tool is also gear shaped having the same pitch as the gear to be cut. However, number of cutting teeth must be less than that of the gear to be cut for internal gears. For external gears the number of teeth on the cutter is limited only by the size of the shaping machine. For larger gears, the blank is sometimes gashed to the rough shape to make shaping easier.





Broaching Machines:

Broaching is a machining process that uses a toothed tool, called a broach, to remove material. There are two main types of broaching: linear and rotary. In linear broaching, which is the more common process, the broach is run linearly against a surface of the workpiece to perform the cut. Linear broaches are used in a broaching machine, which is also sometimes shortened to broach. Broaching is used when precision machining is required, especially for odd shapes. Commonly machined surfaces include circular and non-circular holes, splines, keyways, and flat surfaces. Typical workpieces include small to medium sized castings, forgings, screw machine parts, and stampings.

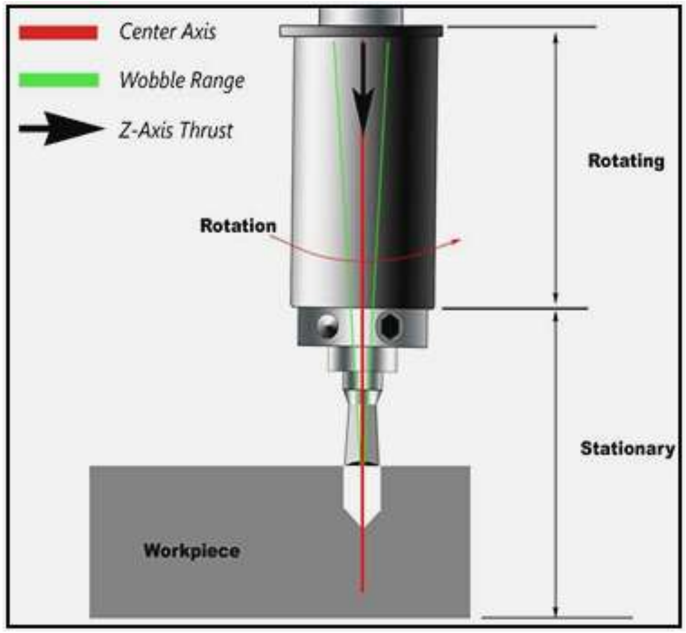
Broaches are shaped similar to a saw, except the teeth height increases over the length of the tool. The profile of the machined surface is always the inverse of the profile of the broach. Hence design, construction and operation of broaching machines, requiring only one such linear motion, are very simple. Only alignments, rigidity and reduction of friction and wear of slides and guides are to be additionally considered for higher productivity, accuracy and surface finish. Most of the broaching machines have hydraulic drive for the cutting motion. Electro-mechanical drives are also used preferably for high speed of work but light cuts.

Broaching machines are generally specified by:

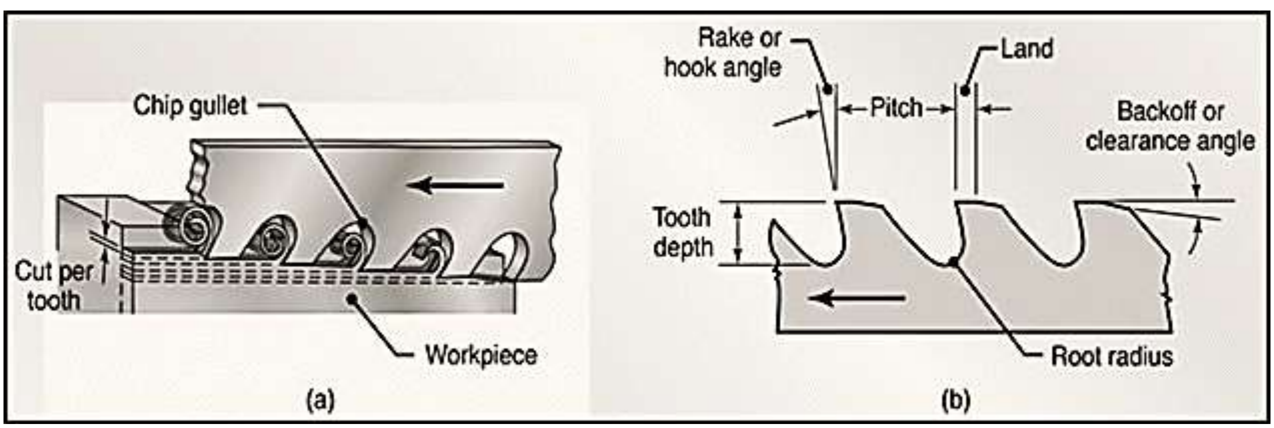
- Type: horizontal, vertical etc.
- Maximum stroke length
- Maximum working force (pull or push)
- Maximum cutting velocity possible
- Power
- Foot print

Horizontal Broaching Machines: Are used for internal broaching and external broaching work, and are the most versatile in application and performance, most widely employed for various types of production, as typically shown below. The horizontal broaching machines are usually hydraulically driven and occupy large floor space.

Vertical Broaching Machines: Vertical broaching machines, as shown below, occupies less floor space, are more rigid as the ram is supported by base, mostly used for external or surface broaching though internal broaching is also possible and occasionally done. Broaching machines commonly have capacity of **3 ton, 6 ton and 10 ton**. These machines are very durable and cost effective, available for workpiece dimensions of **600 mm, 1000 mm, 1200 mm** and can be customized as per specification given by the user.

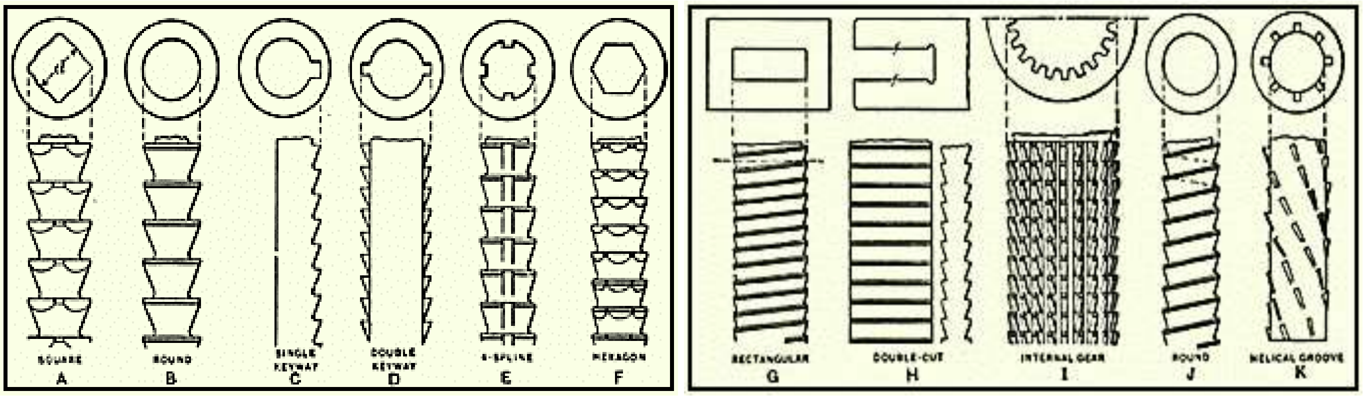


Broaching Principles: Broaching is also a machining process for removal of a layer of material of desired width and depth usually in one stroke by a slender rod or bar type cutter having a series of cutting edges with gradually increased protrusion, as indicated below. The rise per tooth (RPT), also known as the step or feed per tooth, determines the amount of material removed and the size of the chip. The broach can be moved relative to the workpiece or vice-versa. Because all of the features are built into the broach no complex motion or skilled labor is required to use it. A broach is effectively a collection of single-point cutting tools arrayed in sequence, cutting one after the other; its cut is analogous to multiple passes of a shaper.

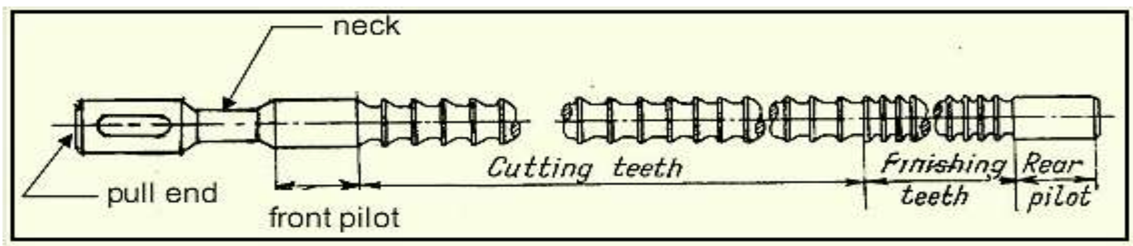


Broaching speeds vary from **20 to 120 surface feet per minute (SFPM)**. This results in a complete cycle time of **5 to 30 seconds**. The only limitations on broaching are that there are no obstructions over the length of the surface to be machined, the geometry to be cut does not have curves in multiple planes, and that the workpiece is strong enough to withstand the forces involved.

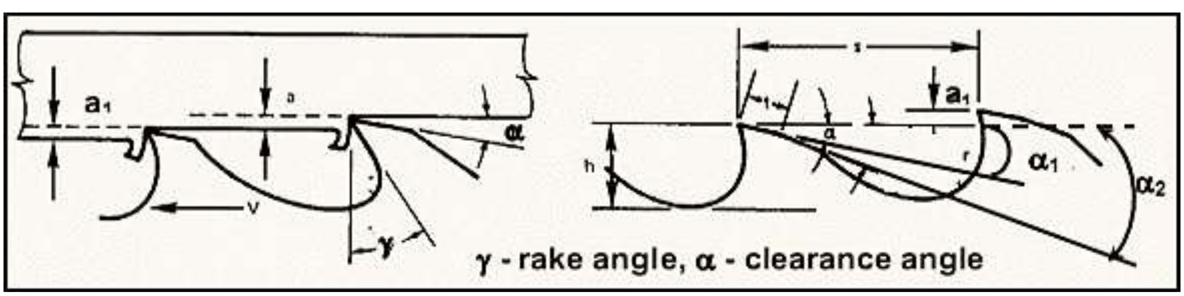
Specifically for internal broaching a hole must first exist in the workpiece so the broach can enter. Also, there are limits on the size of internal cuts. Common internal holes can range from **0.125 to 6 in** (3.2 to 150 mm) in diameter but it is possible to achieve a range of **0.05 to 13 in** (1.3 to 330 mm). Surface broaches' range is usually **0.075 to 10 in** (1.9 to 250 mm), although the feasible range is **0.02 to 20 in** (0.51 to 510 mm). Tolerances are usually **±0.002 in** (±0.05 mm), but in precise applications a tolerance of **±0.0005 in** (±0.01 mm) can be held.



Material of Broaches: For ease of manufacture and re-sharpening the complex shape and cutting edges, broaches are mostly made of **HSS** (high-speed steel). To enhance cutting speed, productivity and product quality, now-a-days cemented carbide segments (assembled) or replaceable inserts are also used specially for stronger and harder work materials like cast irons and steels. **TiN-coated** carbides provide much longer tool life in broaching. Since broaching speed (velocity) is usually quite low, ceramic tools are not used.



Geometry of Broaching Teeth: The cutting teeth of HSS broaches are provided with positive radial or orthogonal rake (**5° to 15°)** and sufficient primary and secondary clearance angles **(2° to 5° and 5° to 20°** respectively) as indicated below. Small in-built chip breakers are alternately provided on the roughing teeth of the broach as can be seen above to break up the wide curling chips and thus preventing them from clogging the chip spaces and increasing forces and tool wear. Materials that are more ductile need wider and frequent chip breakers.



Broaching Operations: The broaching operation needs only one motion which is the cutting motion, imparted to the tool. The magnitude of cutting velocity, V is decided based on the tool-work materials and the capability of the broaching machine. In broaching metals and alloys, **HSS broaches** are used at cutting velocity of **10 to 20 m/min** and **carbide broaches** at **20 to 40 m/min**. The value of tooth rise varies within **0.05 mm to 0.2 mm** for **roughing** and **0.01 to 0.04 mm** for **finishing**. Some cutting fluids are preferably used mainly for lubrication and cooling at the chip – tool interfaces.

Grinding Machines:

A grinding machine, often shortened to grinder, is a machine tool used for grinding, which is a type of machining using an abrasive wheel as the cutting tool. Each grain of abrasive on the wheel's surface cuts a small chip from the workpiece via shear deformation. Grinding is used to finish workpieces that must show high surface quality (e.g., low surface roughness) and high accuracy of shape and dimension. As the accuracy in dimensions in grinding is on the order of **0.000025 mm**, in most applications it tends to be a finishing operation and removes comparatively little metal, about **0.25 to 0.50 mm depth**. However, there are some roughing applications in which grinding removes high volumes of metal quite rapidly. Thus, grinding is a diverse field.

Grinding Characteristics:

Belt Grinder: For all kind of applications like finishing, deburring, and stock removal.

Bench Grinder: Has two wheels of different grain sizes for roughing and finishing operations and is secured to a workbench or floor stand. Its uses include shaping tool bits or various tools that need to be made or repaired.

Cylindrical Grinder: Use centers and the centerless types. A cylindrical grinder may have multiple grinding wheels. It is used to make precision rods, tubes, bearing races, bushings, and many other parts.

Surface Grinder: Has a head, which is lowered, and the workpiece is moved back and forth past the grinding wheel on a table that has a permanent magnet for use with magnetic stock. Surface grinders can be manually operated or have CNC controls. This type includes the wash grinder.

Cutter Grinder: These usually can perform the minor function of cutting metals, or other specialist tool room grinding operations and includes the D-bit grinder.

Jig Grinder: Has a variety of uses for finishing jigs, dies, and fixtures. The primary function is grinding the realm of holes and pins, but can also be used for surface grinding, and finish the work started on a mill.

Gear Grinder: Is usually employed as the **final machining** process when manufacturing a high-precision gear. The primary function of these machines is to remove the remaining few **thousandths of an inch** of material left by other manufacturing methods (such as gashing or hobbing).

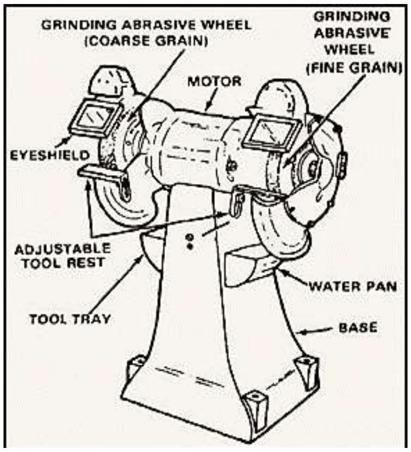
Grinding Machine Types:

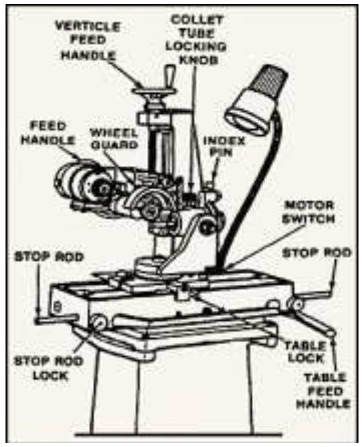
Floor Mounted Grinding Machines: The typical floor-mounted grinding machine is secured to the floor by bolts. The floor mounted grinding machine below shows two 12 in. diameter x 2 in. wide grinding abrasive wheels. The two wheel arrangements have a **coarse grain wheel for roughing purposes** on one end and a **fine grain wheel for finishing purposes** on the other end; this saves the time that would be otherwise consumed in changing wheels.

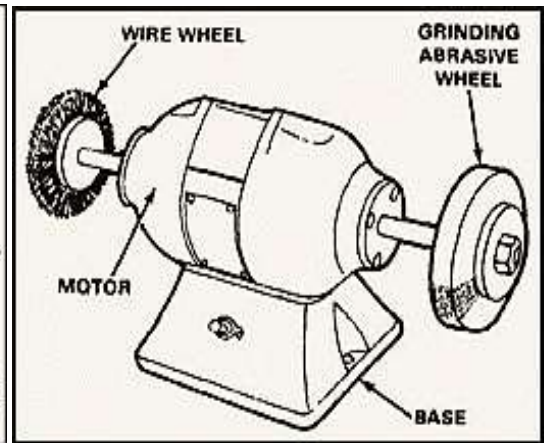
Bench Grinding Machines: Are also mounted for convenience of operation, each tool provided with an adjustable table tool rest and an eye shield for protection, with **one coarse grinding wheel** and **one fine grinding wheel**. The motor may be equipped with a thermal over-load switch to stop the motor if excessive wheel pressure is applied, preventing burning out. The motor revolve at **3.450 RPM** maximum with maximum cutting speeds, for a 7 in. grinding wheels of about **6,300** surface feet per minute (SFPM).

Bench-Type Cutter Grinders: The bench-type cutter grinder, as shown below, was designed primarily to grind end mills. It can also **grind a large variety of small wood and steel cutters** as well as **slitting saw** cutters up to 12 inches in diameter using a saw grinding attachment. Capacity of the typical bench-type cutter grinder is as follows:

Bench Grinding and Buffing Machines: More suitable for general grinding, **not recommended** for tool grinding, since it contains no tool rests, eye shields, or wheel guards. This machine has a 4 in. diameter wire wheel on one end, **used for cleaning**, and an abrasive wheel on the other end used for general grinding. The **1/4-HP** electric motor revolves at a maximum of **3,450 RPM**.



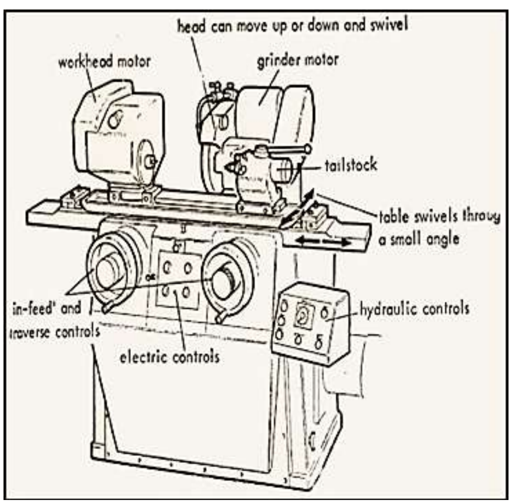


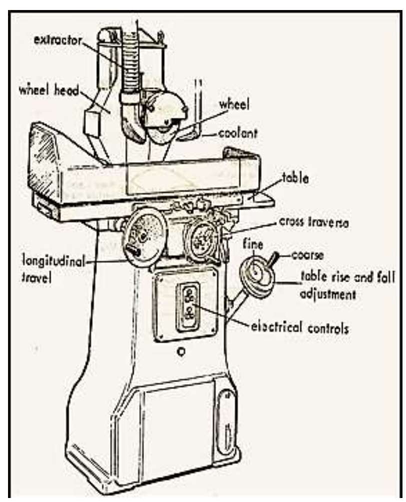


Floor Mounted Grinding Machines Bench-Type Cutter Grinders Bench Grinding and Buffing Machines

Universal Grinding Machines: Include a wheel head, which incorporate the spindle and drive motor; a sliding table that moves the wheel head to and from the workpiece; a headstock, which locates, holds, and drives the workpiece; and a tailstock, which holds the other end of the work. **Universal grinding** is divided into three general operations: **plain cylindrical, conical grinding (taper grinding), and internal grinding**.

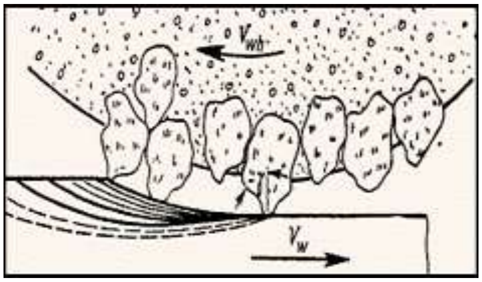
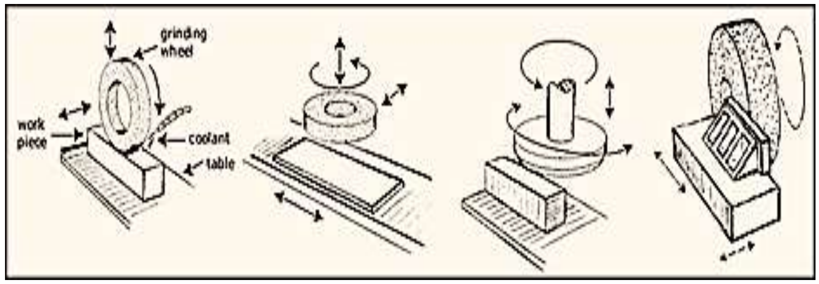
Reciprocating Surface Grinding Machines: The work-pieces are fastened to the table and moved beneath the grinding abrasive wheel by hand or power feed. This type of grinding machine has an internal pump and a piping network for recirculation of a coolant to the workpiece and wheel. The grinding abrasive wheel, mounted to a horizontal spindle is straight and cuts in a circumferential way surface only.





Universal Grinding Machines Reciprocating Surface Grinding Machine

Grinding Operations: Is a material cutting process which engages an abrasive tool whose cutting elements are grains of abrasive material known as grit. These grits are characterized by sharp cutting points, high hot hardness, chemical stability, and wear resistance. A suitable bonding material holds the grits together.

Calculating Wheel Size and Speeds: Grinding wheel speeds commonly used in precision grinding vary from **5,500 to 9,500 surface feet per minute** (sfpm), or ft/min.. To find the wheel speed in **sfpm**, multiply the spindle speed (rpm) by the wheel circumference (inches) and divide the product by **12**.

$$sfpm = \frac{\pi \times D \times rpm}{12}$$

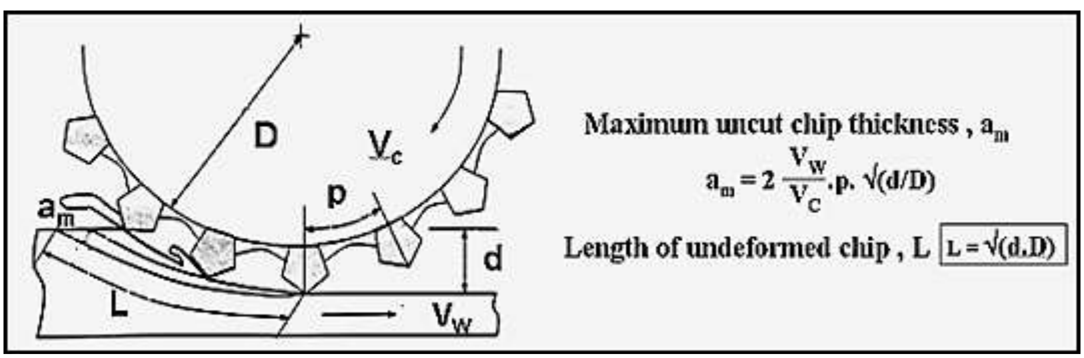
Sfpm = Cutting speed of wheel (ft/min);
Rpm = Revolutions per minute of wheel;
D = Wheel diameter (in inches).

Work for Cylindrical Grinding: In universal cylindrical grinding, it is difficult to recommend any work speeds since these are dependent upon whether the material is rigid enough to hold its shape, whether the diameter of the workpiece is large or small, and so forth. Listed below are areas to consider when performing cylindrical grinding:

- The typical work or surface speeds are: steel shafts, **50 to 55 m/min**; hard steel rolls, **80 to 85 m/min**; chilled iron rolls, **80 to 200 m/min**; cast iron pistons, **150 to 400 m/min**; crankshaft bearings, **45 to 50 m/min**; and crankshaft pins, **35 to 40 m/min**;

- Higher work speeds increase the cutting action of the wheel and may indicate that a harder wheel and a smaller depth of cut be used to reduce wheel wear.

Feeds: The feed of the grinding wheel is the distance the wheel moves laterally across the workpiece for each revolution of the piece in cylindrical grinding or in each pass of the piece in surface grinding.

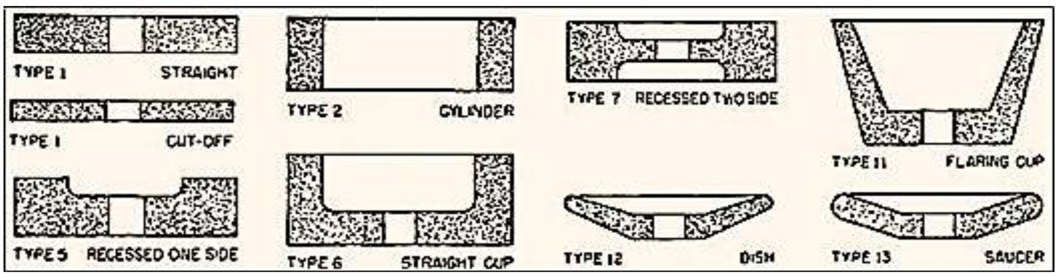


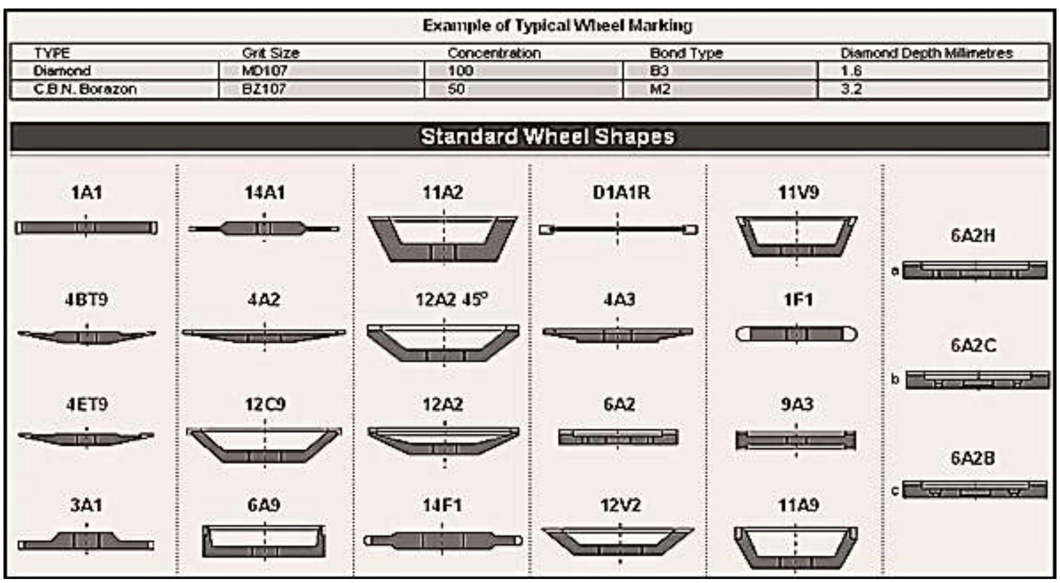
Depth of Cut: The depth of cut depends on such factors as the material from which the work is made, heat treatment, wheel and work speed, and condition of the machine. Methods for determining depth of cuts are also recommended by manufacturers´ tables. In roughing, the depth of cut also depends on the hardness of the material. In cylindrical grinding, in addition to these factors, the cut depends on the diameter of the work. Generally, a cut of **0.001 to 0.003 inch in depth** is used, depending on the size and condition of the grinding machine.

Coolants: Most grinding machines are equipped with coolant systems. The coolant is directed over the point of contact between the grinding wheel and the work. This prevents distortion of the workpiece due to uneven temperatures caused by the cutting action. Clear water may be used as a coolant. An inexpensive coolant, consists of a **solution of approximately 1/4 pound of sodium carbonate (salt soda) dissolved in 1 gallon of water**, or by dissolving **soluble cutting oil in water**. For grinding aluminum and its alloys, a clear water coolant will produce good results.

Specification of Grinding Wheels: Grinding wheel consists of hard abrasive grains called **grits**, which perform the cutting or material removal. The type of the abrasive material used commonly identifies a grinding wheel. The conventional wheels include aluminum oxide, **silicon carbide** wheels while diamond, and **cBN** (cubic boron nitride) wheels fall in the category of super abrasive wheel. A grinding wheel requires two types of specification:

Geometrical Specification: Is the type of grinding machine and the grinding operation to be performed in the workpiece and includes the wheel diameter, width and depth of rim and the bore diameter. The wheel diameter, for example can be as high as **400 mm (16 inches)** for high efficiency grinding or as small as less than 1mm (0.04 inch) for internal grinding. Standard wheel configurations for conventional grinding wheels are shown below.





Types of Abrasives:

Aluminum Oxide: Pure Al_2O grit is advantageous in fine tool grinding operation, and heat sensitive operations on ferrous materials. Regular or brown aluminum oxide (doped with TiO_3), possesses lower hardness and higher toughness than the white Al_2O_3 for heavy duty grinding. Al_2O_3 alloyed with chromium oxide (<3%) is pink in color. The types are: **monocrystalline,** Al_2O_3 grits are efficient in medium pressure heat sensitive operation on ferrous materials; **microcrystalline**, Sintered Al_2O_3 grit is practically suitable for stock removal grinding. **microcrystalline Al_2O_3 alloyed with zirconia**, for high material removal grinding, not recommended for precision grinding.

Silicon Carbide: Is harder than alumina but less tough and inferior to Al_2O_3 because of its chemical reactivity with iron and steel. Black carbide containing at least 95% SiC is less hard but tougher than green SiC and is efficient for grinding soft nonferrous materials. Green silicon carbide contains at least 97% SiC; is harder than the black variety, and used for grinding cemented carbide.

Diamond Grit: Is best suited for grinding cemented carbides, glass, sapphire, stone, granite, marble, concrete, oxide, non-oxide ceramic, fiber reinforced plastics, ferrite and graphite. The most important are: **natural diamond grits,** very sharp cutting edge and free cutting action and is exclusively used in metallic, electroplated and brazed bond; **monocrystalline diamond grits**, designed for particularly demanding application; **polycrystalline diamond grits**, most suitable for grinding of cemented carbide with low pressure. These grits are used in resin bond.

Cubic Boron Nitride (cBN): Is the second hardest material, **and a better choice** for efficient grinding of HSS, alloy steels, HSTR alloys, because of its chemical stability. The cBN grits are available as mono crystalline type with medium strength and blocky mono-crystals with much higher strength.

Bonds:

Vitrified Bond: Suitable for high stock removal even at dry condition, and be safely used in wet grinding. It cannot be used where mechanical impact or thermal variations may occur.

Resin Bond: Used for heavy duty grinding because of their ability to withstand shock load and vibration absorbing. It finds better use with diamond and cBN in grinding cemented carbide and steel respectively.

Fiberglass Reinforced Resin Bond: Used as cut off wheels, but requires added strength under high-speed operation.

Shellac Bond: This bond was used for flexible cut off wheels, limited to grinding wheels in fine finish of rolls.

Oxychloride Bond: Less common type, may be used in disc grinding operation, under dry conditions.

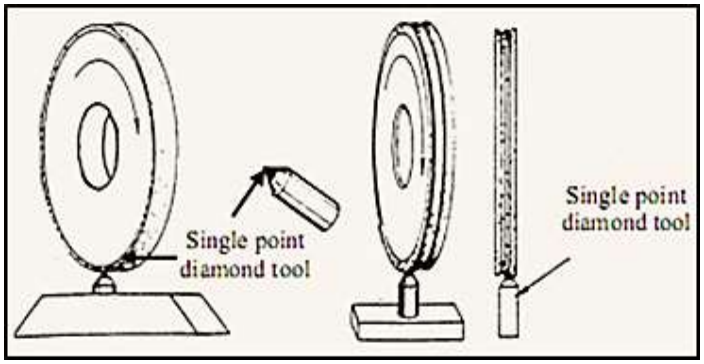
Rubber Bond: Rubber bond was once popular for finish grinding on bearings and cutting tools.

Metal Bond: Extremely high toughness of metal bonded wheels makes these very effective in those applications where form accuracy as well as large stock removal is desired.

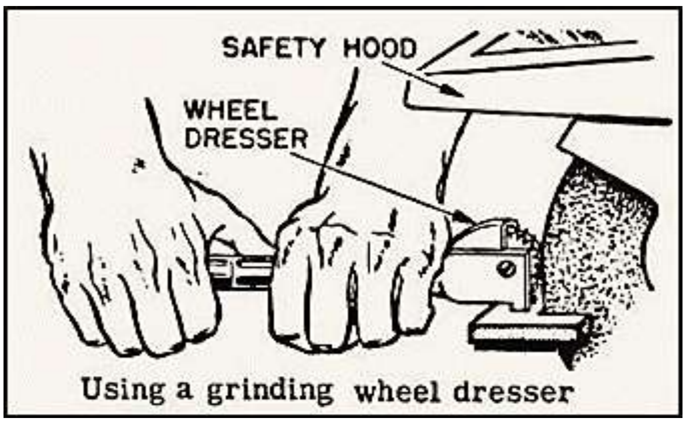
Electroplated Bond: This bond allows large (30-40%) crystal exposure above, without need of any truing or dressing, specially used for making small diameter wheels, form and thin super abrasive wheels.

Brazed Bond: This is relatively a recent development, allows crystal exposure as high 60-80%. This bond is particularly suitable for very high material removal either with diamond or cBN wheels. The bond strength is much greater than provided by the electroplated bond, and is expected to replace the electroplated bond in many applications.

Truing Grinding Wheels: Is the act of **regenerating the required geometry of grinding wheels**, when the geometry is a special form or a flat profile. **The truing operation** produces the macro-geometry, also required on new conventional wheels to ensure concentricity with a specific mounting system. Truing are made by setting a high quality single point crystal into a usually cylindrical shank of a specific diameter and length by brazing or casting around the diamond, as shown on the sketch below. In the photograph below, see the **1/4 caret single point diamond** used for truing a grinding wheel.



Dressing Grinding Wheels: Is the conditioning of the wheel surface, which ensures that grit cutting edges are exposed from the bond and thus able to penetrate into the workpiece material. The dressing operation make the abrasive grains sharp and free cutting proper to remove any residue left by material being ground. Dressing therefore produces a micro-geometry. The structure of the micro geometry of grinding wheel, determines its cutting ability with a wheel of given composition. Dressing can substantially influence the condition of the grinding tool.



Grinding Wheels Marking System:

Conventional grinding wheels: The standard marking system for conventional abrasive wheel can be as follows:

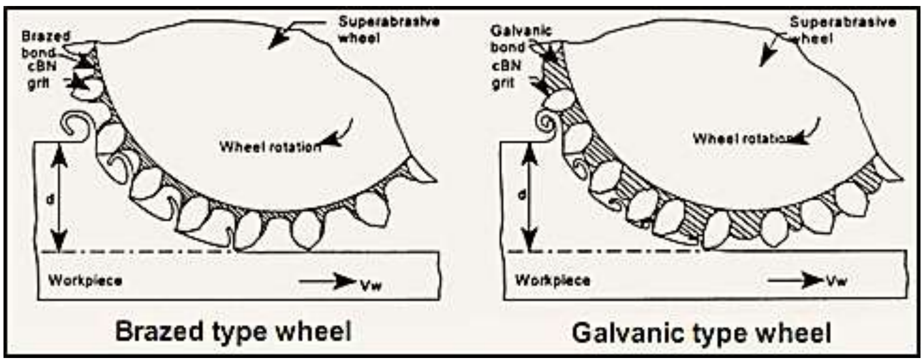
Example: 51 A 60 K 5 V 05:

- The number '51' is manufacturer's identification number indicating exact kind of abrasive used;
- The letter 'A' denotes that the type of abrasive is aluminum oxide. In case of silicon carbide the letter 'C' is used;
- The number '60' specifies the average grit size in inch mesh. For a very large size grit this number may be as small as 6. For a very fine grit the designated number may be as high as 600;
- The letter 'K' denotes the hardness of the wheel, which means the amount of force required to pull out a single bonded abrasive grit by bond fracture. The letter symbol can range between 'A' and 'Z', 'A' denoting the softest grade and 'Z' denoting the hardest one;
- The number '5' denotes the structure or porosity of the wheel. This number can assume any value, between 1 to 20, '1' indicating high porosity and '20' indicating low porosity;
- The letter code 'V' means that the bond material used is vitrified. The codes for other bond materials used in conventional abrasive wheels are B (resinoid), BF (resinoid reinforced), E(shellac), O (oxychloride), R(rubber), RF (rubber reinforced), S(silicate);
- The number '05' is a wheel manufacturer's identifier;

Super abrasive grinding wheels: Somewhat different, as indicated below:

Example: **R D 120 N 100 M 4**:

- The letter 'R' is manufacture's code indicating the exact type of super abrasive used.
- The letter 'D' denotes that the type of abrasive is diamond. In case of cBN the letter 'B' is used.
- The number '120' specifies the average grain size in inch mesh. However, a two number designation (e.g. 120/140) is utilized for controlling the size of super abrasive grit. The two number designation of grit size along with corresponding designation in micron is given in table 28.1.
- Like conventional abrasive wheel, the letter 'N' denotes the hardness of the wheel. However, resin and metal bonded wheels are produced with almost no porosity and effective grade of the wheel is obtained by modifying the bond formulation.
- The number '100' is known as concentration number indicating the amount of abrasive contained in the wheel. The number '100' corresponds to an abrasive content of 4.4 carats/cm3. For diamond grit, '100' concentration is 25% by volume. For cBN the corresponding volumetric concentration is 24%.
- The letter 'M' denotes that the type of bond is metallic. The other types of bonds used in super abrasive wheels are resin, vitrified or metal bond, which make a composite structure with the grit material. However, another type of super abrasive wheel with both diamond and cBN is also manufactured where a single layer of super abrasive grits are bonded on a metal perform by a galvanic metal layer or a brazed metal layer.



Marking Standards: Every grinding wheel is marked with a small tag, as shown below. For example, using a wheel marked as **A36-L5-V23**. The **"A"** refers to the abrasive, which is aluminum oxide. The **"36"** represents the grain size. The **"L"** shows the grade or degree of hardness, which is medium. The **"5"** refers to the structure of the wheel and the **"V"** refers to the bond type.