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 & Fax: 703-988-0088

www.PDHonline.org
www.PDHcenter.com

An Approved Continuing Education Provider
## Introduction

### Lathes
- Metalworking lathes
- Classification of lathes
- Turning operations
- Semiautomatic and automatic lathes
- Accessories
- Live centers and dead centers
- Rests and micrometer supports
- Lathe cutting tools
- Lathe calculations
- Graduate micrometer and measurements
- Tools and inserts
- Common holders with inserts
- Goose-neck holders with inserts

### Drilling Machines
- Classification of drilling machines
- Application of drilling machines
- Types of drills
- Drill sizes and geometry
- Drill point angles
- Drill holding & clamping of workpieces
- Drill bits geometry and nomenclature
- Types of drills with inserts
- Drill materials
- Types of drill bits
- Holding of drill bits
- Devices for holding workpieces
- Drilling calculations

### Milling Machines
- Milling operations
- Classification of milling machines
- Machining centers
- Milling machine parts
- Accessories
- Kinematic system
- Milling cutters description
- Types of milling cutters
- Milling calculations

### Planing Machines & Planers
- Planers
- Planing machine and planer tools
- Cutting calculations

## Shaping Machines

### Slotting Machines
- Planing, shaping and slotting calculations

### Boring Machines
- Types of boring machines
- Boring types
- Boring calculations

### Hobbing & Gear Shaping Machines
- Common gear generation types
- Details of involute gearing
- Proper meshing and contact ratio
- Gear Shaping Machines

### Broaching Machines
- Horizontal broaching machines
- Vertical broaching machines
- Broaching principles
- Broaching configuration
- Materials of broaches
- Geometry of broaching teeth
- Broaching operations

### Grinding Machines
- Grinding characteristics
- Grinding machine types
- Grinding operations
- Grinding applications
- Calculating wheel sizes and speeds
- Coolants
- Specification of grinding wheels
- Types of abrasives
- Bonds
- Truing grinding wheels
- Dressing grinding wheels
- Grinding wheels marking system
- Marking standards

## References & Links
CONVENTIONAL MACHINING TECHNOLOGY – FUNDAMENTALS

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 which 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
handwheels 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 onto or off of 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: centre 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: centre 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
Chuck 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;
- swivelong 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 centre 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.
(a) 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 centre 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:

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

(b) Automatic:

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

a) Semiautomatic 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 centre 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 presetting 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.

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.

- Chucking type;
- relatively large size;
- requiring limited number of machining operations;
- lesser floor space occupied;
- easy loading and unloading of blanks and finished jobs;
- relieving the spindles of bending loads due to job – weight;
- 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 processing type:** 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 processing type:** 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 work stations 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 type:** 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.

b) **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](image)

**Swiss type automatic lathe:** The characteristics and applications of these single spindle automatic lathes are:

- **Multispindle automatic lathes:** For further increase in 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

- **Lot or mass production:** Thin slender rod or tubular jobs, like components of small clocks and wrist watches, 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 lathe:** 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 lathe:** 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 preferably and 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 cam shaft;
- Relatively smaller size and power but have higher spindle speeds.
Tool layout: A typical tool layout for a particular job being machined in a single spindle automatic lathe is schematically shown below.

![Tool layout for a typical job in a single automatic lathe.](image)

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 draw-bar, or by a collet closing cap on the spindle.

![Travelling or fixed Steady Rest](image)

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 centre. Because the centre 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 clear 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.

![Correctly Drilled Hole vs. Incorrectly Drilled Hole](image)

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 dovetail 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.

Live centers and dead centers:

- Mounting the work between centers using a "dog: The centers are referred to as 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 can be used to hold and automatically center round or hexagonal workpieces. 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.

![3-Jaw chuck and 4-Jaw chuck](image)

✓ **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 handwheel or hand lever to move the drawbar. The spring machine collet and collet attachment together form the collet chuck, as shown below.

![Collet with three slits](image)

✓ **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 workpieces 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.

![Faceplate](image)

**Rests and micrometer supports:**

Workpieces often need extra support, especially long, thin workpieces 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 workpieces 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.** The 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.

- **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.

✓ **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.

✓ **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, as described below, 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 cutting 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 cutting.
- The heel is the portion of the tool bit base immediately below and supporting the face.

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:
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: Besides the common shaped tool bits, 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 left-hand turning tool bits and 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:

- the insert's basic shape
- its relief or clearance angle
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:

- the 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 chipbreaker 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 chipbreaker design.

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** ($\lambda$) is a measure of at what angle the insert is mounted in the toolholder. The **rake angle** ($\gamma$) 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.

**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** ($V_c$), (rotational or peripheral speed) in **ft/min** or **m/min**.
The cutting feed \((f_n)\) in \(\text{ft/rev}\) or \(\text{mm/rev}\) 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 \(\text{ft}\) or \(\text{mm}\) 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 \((\kappa_r)\) 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.

Lathe calculations:

Before any operation can be done, a thorough knowledge of the variable factors of \textit{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 \textit{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 \textit{feed rate} \((f_R)\) is expressed in \(\text{ft/rev}\) and the \textit{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) = \[
\frac{\text{Cutting Speed, } C_s \text{ (ft/min)} \times 12}{\pi \times D}
\]

2) Cutting Speed, V, Vc, Cs, SFM, (ft/min) = \[
\frac{\pi \times D \times N \text{ (Spindle speed, RPM)}}{12}
\]

This can be closely approximated as:

3) Spindle speed, spindle rotation, N (RPM) = \[
\frac{C_s \times 4}{D} \text{ - (or - RPM = } \frac{C_s \times 320}{D})
\]

Where:

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) = \[
\frac{200 \times 4}{1/2''} = 1600 \text{ RPM}
\]

If no cutting speed tables are available, remember that, generally hard materials require a slower cutting speed than soft or ductile materials. 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. (Always 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) = \[
\frac{\text{Cutting speed, } C_s \text{ (in/min) \times Spindle rotation, N (RPM)}}
\]

5) Cutting speed Cs, Vc, (in/min) = Spindle speed, N (RPM) \times Cutting feed, \( f_R \) (ft/rev) (in/rev);

6) Cutting speed, Cs, Vc (ft/min) = \pi \times D \times N /12 \text{ [Where: } \pi (3.1416); D \text{ (spindle diameter, inches); N (RPM)}\].
Note: Feed rate \( f_R \) 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 \( V_c, C_s \) 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, \( C_t \) (min) = \( \frac{\text{Cut length, } L, \ L_w \text{ (in)}}{\text{N (RPM)} \times \text{Feed } f_R \text{ (in/rev)}} \\) Cutting speed, \( C_s \) (in/min)

Or:

\[
C_t = \frac{(L + A)}{N \times f_R} \quad \text{or} \quad C_t = \frac{Lc \times np \text{ (see sketch)}}{N \times f_R}
\]

Where:

- \( C_t \) = is cutting time per “pass” [min];
- \( L_w \) = length cut [in] [mm] – \( L, \ L_c = L_w + 2A \) [in] [mm]
- \( A \) = is “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, usually measured in thousands 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 (thousandths of an in.) - \( D_c = 5 \times \text{feed rate (in/rev)} \) or \( D_c \) (in) = \( \frac{(D_1 - D_2)}{2} \)

Material Removal Rate (MRR):

9) \( \text{MRR} = \frac{\text{Volume Removed}}{\text{Cutting time}} \) \( \text{in}^3/\text{min} \) (mm\(^3\)/min) = \( \frac{\pi \times L \times (D_1^2 - D_2^2)}{4 \times L \times N \times f_R} \)

Cutting speeds \( V_c, C_s \) references for turning and threading with HSS tool bits:

<table>
<thead>
<tr>
<th>MATERIAL</th>
<th>STRAIGHT TURNING SPEED</th>
<th>THREADED TURNING SPEED</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>FEET PER MINUTE</td>
<td>METERS PER MINUTE</td>
</tr>
<tr>
<td>LOW-CARBON STEEL</td>
<td>80-100</td>
<td>24.4-30.5</td>
</tr>
<tr>
<td>MEDIUM-CARBON STEEL</td>
<td>60-80</td>
<td>18.3-24.4</td>
</tr>
<tr>
<td>HIGH-CARBON STEEL</td>
<td>35-40</td>
<td>10.7-12.2</td>
</tr>
<tr>
<td>STAINLESS STEEL</td>
<td>40-50</td>
<td>12.2-15.2</td>
</tr>
<tr>
<td>ALUMINUM AND ITS ALLOYS</td>
<td>200-300</td>
<td>61.0-91.4</td>
</tr>
<tr>
<td>ORDINARY BRASS AND BRONZE</td>
<td>100-200</td>
<td>30.5-61.0</td>
</tr>
<tr>
<td>HIGH-TENSIILE BRONZE</td>
<td>40-60</td>
<td>12.2-18.3</td>
</tr>
<tr>
<td>CAST IRON</td>
<td>50-80</td>
<td>15.2-24.4</td>
</tr>
<tr>
<td>COPPER</td>
<td>80-80</td>
<td>18.3-24.4</td>
</tr>
</tbody>
</table>

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):

<table>
<thead>
<tr>
<th>Material</th>
<th>Rough Cuts</th>
<th>Finish Cuts</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>in/rev</td>
<td>mm/rev</td>
</tr>
<tr>
<td>Carbon steel</td>
<td>0.010–0.020</td>
<td>0.25–0.5</td>
</tr>
<tr>
<td>Tool steel</td>
<td>0.015–0.025</td>
<td>0.4–0.65</td>
</tr>
<tr>
<td>Cast iron</td>
<td>0.015–0.025</td>
<td>0.4–0.65</td>
</tr>
<tr>
<td>Bronze</td>
<td>0.015–0.030</td>
<td>0.4–0.75</td>
</tr>
<tr>
<td>Aluminum</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Material</th>
<th>Rough Cuts</th>
<th>Finish Cuts</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>ft/min</td>
<td>m/min</td>
</tr>
<tr>
<td>Carbon steel</td>
<td>90</td>
<td>27</td>
</tr>
<tr>
<td>Tool steel</td>
<td>70</td>
<td>21</td>
</tr>
<tr>
<td>Cast iron</td>
<td>60</td>
<td>18</td>
</tr>
<tr>
<td>Bronze</td>
<td>27</td>
<td>100</td>
</tr>
<tr>
<td>Aluminum</td>
<td>200</td>
<td>61</td>
</tr>
</tbody>
</table>

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

<table>
<thead>
<tr>
<th>Material</th>
<th>Condition</th>
<th>HSS</th>
<th>Carbide</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td>12mm</td>
<td>25mm</td>
</tr>
<tr>
<td>WROUGHT MARTENSITIC Free machining grades</td>
<td>160</td>
<td>33</td>
<td>0.08</td>
</tr>
<tr>
<td>Lower C/lower Cr grades</td>
<td>175</td>
<td>26</td>
<td>0.08</td>
</tr>
<tr>
<td>Lower C/lower Cr grades</td>
<td>300</td>
<td>14</td>
<td>0.05</td>
</tr>
<tr>
<td>Higher C/higher Cr grades</td>
<td>280</td>
<td>14</td>
<td>0.08</td>
</tr>
<tr>
<td>Higher C/higher Cr grades</td>
<td>320</td>
<td>12</td>
<td>0.04</td>
</tr>
<tr>
<td>WROUGHT FERRITIC Free machining grades 12 - 17% Cr grades</td>
<td>160</td>
<td>33</td>
<td>0.08</td>
</tr>
<tr>
<td>WROUGHT AUSTENITIC Free machining grades Other grades (304, 316, 321 etc)</td>
<td>160</td>
<td>21</td>
<td>0.08</td>
</tr>
<tr>
<td>WROUGHT DUPLEX</td>
<td>230</td>
<td>17</td>
<td>0.08</td>
</tr>
<tr>
<td>CAST PLAIN Cr</td>
<td>210</td>
<td>15</td>
<td>0.04</td>
</tr>
<tr>
<td>CAST AUSTENITIC</td>
<td>170</td>
<td>11</td>
<td>0.04</td>
</tr>
</tbody>
</table>

Table 1. Recommended values for precision boring/turning:

<table>
<thead>
<tr>
<th>Workpiece material</th>
<th>Hardness [Bhn]</th>
<th>Cutting speed - High-speed steel [Vc [ft/min], Vc [m/min]]</th>
<th>Cutting speed - Carbide uncoated [Vc [ft/min], Vc [m/min]]</th>
<th>Feed rate per revolution [f [inch], f [mm]]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>MIN</td>
<td>MAX</td>
<td>MIN</td>
<td>MAX</td>
</tr>
<tr>
<td>Cast iron</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Steel - plain carbon</td>
<td>85</td>
<td>1500</td>
<td>40</td>
<td>500</td>
</tr>
<tr>
<td>Steel - alloys</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Steel - tool</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Steel - stainless</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Aluminum alloys</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Copper alloys</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Nickel alloys</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Titanium</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
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**

\[
\text{RPM} = \frac{\text{CS} \times 4}{D} = \frac{90 \times 4}{2} = 180 - (Cs = 90) \text{table}
\]

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

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

**Finishing cut**

\[
\text{RPM} = \frac{100 \times 4}{1.850} = 216.\text{rpm} - (Cs = 100) \text{table}
\]

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

\[
\text{Average finishing cut time} = \frac{16}{0.003 \times 216} = 24.7 \text{ min}
\]

**Cutting Speeds:**

- Rate at which point on work circumference travels past cutting tool, 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.
  - **Too high:** cutting-tool breaks down rapidly.
  - **Too low:** time lost, low production rates.

**Spindle rotation in metric values:**

10) \(N = \frac{\text{Cs} \times 320}{D}\) = Where, \(D\) (mm) and Cs (see recommended tables) = Cutting speed (m/min).

**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).
Turning calculations sketch:

On machines where the workpiece revolves, the cutting tool should be set in for only half the amount of the diameter to be removed:

**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**.
Micrometer 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**.

Micrometer 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 of cut 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.
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 spotfacing. 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 sensitive drilling machines: These small capacity (= 0.5 kW) upright (vertical) single spindle drilling machines are mounted (bolted) 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 table top 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 work table movement are also done manually. These low cost drilling machines have tall tubular columns and are generally used for small jobs and light drilling.

Goose-neck holders with inserts:

Solution 1: The workpiece burr does not contact the guide bush and no breakage will be caused.

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

Fig.1, Fig. 2 and Fig.3
Box column drilling machines: These box shaped column type drilling machines (Fig.3) are much more strong, rigid and powerful than the pillar drills. In column drills the feed gear box 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 work table 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](image)

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.
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.

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 base, for quick release keyhole mounting, has a location spigot that enables the drill to be removed and replaced. The spigot sleeve also allows for optional drill bushes to be fitted to centralize the drill when starting on radial or uneven faces, as shown in Fig. 10.

**Micro (or mini) 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**: Unlike the grouted stationary drilling machines, 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.](image)

**Pneumatic drilling machines**: 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](image)
**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.

**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.

General purpose drills may be classified as;

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

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

• **According to number of flutes:**
  Δ Two fluted – most common;
  Δ Single flute – e.g., gun drill (robust);
  Δ Three or four flutes – called slot drill;
• According to helix angle of the flutes:
  Δ Usual – 20° to 35° – most common;
  Δ Large helix : 45° to 60° suitable for deep holes and softer work materials;
  Δ Small helix : for harder / stronger materials;
  Δ Zero helix : spade drills for high production drilling micro-drilling and hard work materials.

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

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

• According to specific applications:
  Δ Centre drills - for small axial hole with 60° taper end to accommodate lathe centre for support;
  Δ Step drill and subland 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.

- **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**: This method was 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**: Trepanning 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**: 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 small chips are easily removed by the flutes of the drill. 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 workpiece's 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 (or of a boring head containing several such tools), 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 90° to 140° for aluminum alloys.
- Use 70° to 80° for magnesium alloys.
- Use 118° to 135° for high strength steels.
Effect of changing the drill point angle:

- The rake angle is influenced by the drill point angle.

Lip relief (clearance) angles:

- Lip relief angles are normally 10° to 15°;
- Lip relief for high strength or tougher steels is between 7° to 12° (Backing for cutting edge);
- Lip relief for softer or free machining materials may be between 12° to 18°;

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.

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, described in the drill bit sizes article. A comprehensive drill bit and tap size chart lists metric and imperial sized drill bits alongside the required screw tap sizes.

The term drill may refer to either a drilling machine or a drill bit for use in a drilling machine. 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, 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).

Most drill bits for commercial purposes use have straight shanks. For heavy duty drilling in industry, bits with tapered shanks are sometimes used.

- 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, pressured-oil gun drill bits, etc.), 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

<table>
<thead>
<tr>
<th>Workpiece material</th>
<th>Point angle</th>
<th>Helix angle</th>
<th>Lip relief angle</th>
</tr>
</thead>
<tbody>
<tr>
<td>Aluminum</td>
<td>90 to 135°</td>
<td>32 to 48°</td>
<td>12 to 26°</td>
</tr>
<tr>
<td>Brass</td>
<td>90 to 118°</td>
<td>0 to 20°</td>
<td>12 to 26°</td>
</tr>
<tr>
<td>Cast iron</td>
<td>90 to 118°</td>
<td>24 to 32°</td>
<td>7 to 20°</td>
</tr>
<tr>
<td>Mild steel</td>
<td>118 to 135°</td>
<td>24 to 32°</td>
<td>7 to 24°</td>
</tr>
<tr>
<td>Stainless steel</td>
<td>118 to 135°</td>
<td>24 to 32°</td>
<td>7 to 24°</td>
</tr>
<tr>
<td>Plastics</td>
<td>60 to 90°</td>
<td>0 to 20°</td>
<td>12 to 26°</td>
</tr>
</tbody>
</table>

### Types of drills with inserts:

![Diagram of drill types](image)

### Drill materials:

Many different materials are used for or on drill bits, depending on the required application. 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. They are used only for drilling wood; even working with hardwoods rather than softwoods can noticeably shorten their lifespan.
- Bits made from **high carbon steel** 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.

**Other materials:**

- **Tungsten carbide** and other carbides 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. However, it is becoming common in job shops to use solid carbide bits. In very small sizes it is difficult to fit carbide tips; 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 later 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.
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.

Drill bits with no point angle 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 drill bit will suffer premature wear. Long series drill bits are unusually 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.

**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:** (also called a step drill bit, as shown below) is a roughly conical bit with a stair-step profile. The larger-size bits have blunt tips and are used for hole enlarging. Unibits are commonly used on sheet metal and in general construction. 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, particle board, drywall, acrylic, and laminate. They can be used on very thin sheet metal, but metals tend to cause premature bit wear and dulling, but are ideal for use in electrical work where thin steel, aluminum or plastic boxes and chassis are encountered. The short length of the unibit and ability to vary the diameter of the finished hole is an advantage in chassis or front panel work. The finished hole can often be made quite smooth and burr-free, especially in plastic.

**Left-hand bits:** 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.

With the increased use of the more versatile CNC machines, their use is less common than when specialized machines were required for machining tasks. Screw extractors are essentially left-hand bits of specialized shape, used to remove common right-hand screws whose heads are broken or too damaged to allow a screwdriver tip to engage, making use of a screwdriver impossible.

**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 centre 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 also effective in soft plastic.

In metal, the lip and spur drill bit is confined to drilling only the thinnest and softest sheet metals in a drill press. This means these bits tend to bind in metal; given a workpiece of sufficient thinness, they have a tendency to punch through and leave the bit's cross-sectional geometry behind. 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 Jenning 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.
**Gimlet bits:** The gimlet bit is a very old design. The bit is the same style as that used in the gimlet, a self-contained tool 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.

**Hinge sinker bits:** The hinge sinker bit is an example of a custom drill bit design for a specific application. Many European kitchen cabinets are made from particle board or medium-density fiberboard (MDF) with a laminated plastic veneer. Those 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.

**Adjustable wood bits:** An adjustable wood bit, also known as an 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 on the bit, 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.
Masonry drill bits: The masonry bit shown here is a variation of the twist drill bit. The tool is relatively soft steel, and is 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: 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: this operation is always done, if necessary, after drilling or boring holes for accuracy and good surface finishing. Different types of reamers of standard sizes are available for different applications. The figure below shows 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. By rotating the outer sleeve, the jaws can be opened and closed. 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: The chuck is fitted with a Morse-taper shank which fits into a corresponding Morse taper in the spindle. The size of Morse taper is identified from smallest to largest by the numbers 1, 2, 3, 4, 5, and 6. 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.

Morse standard taper shanks

<table>
<thead>
<tr>
<th>No. of Taper</th>
<th>Taper per Foot</th>
<th>Taper per Inch</th>
<th>Small End of Plug D</th>
<th>Diameter End of Socket A</th>
<th>Shank Length B</th>
<th>Shank Depth S</th>
<th>Shank Depth of Hole H</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0.62460</td>
<td>0.05205</td>
<td>0.252</td>
<td>0.3561</td>
<td>2 11/32</td>
<td>2 3/8</td>
<td>2 3/8</td>
</tr>
<tr>
<td>1</td>
<td>0.59858</td>
<td>0.04988</td>
<td>0.369</td>
<td>0.475</td>
<td>2 3/16</td>
<td>2 7/16</td>
<td>2 3/4</td>
</tr>
<tr>
<td>2</td>
<td>0.59941</td>
<td>0.04995</td>
<td>0.572</td>
<td>0.700</td>
<td>3 3/8</td>
<td>2 7/16</td>
<td>2 3/4</td>
</tr>
<tr>
<td>3</td>
<td>0.60235</td>
<td>0.05019</td>
<td>0.778</td>
<td>0.938</td>
<td>3 3/8</td>
<td>3 1/8</td>
<td>3 1/2</td>
</tr>
<tr>
<td>4</td>
<td>0.62326</td>
<td>0.05193</td>
<td>1.020</td>
<td>1.231</td>
<td>4 1/4</td>
<td>4 1/4</td>
<td>4 1/4</td>
</tr>
</tbody>
</table>

At the end of the taper shank, two flats are machined, leaving a portion known as the tang. This tang fits in a slot on the inside of the spindle and its main purpose is for the removal of the shank. Drills are also available with Morse-taper shanks which fit directly into the spindle without the need for a chuck.

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

Hardened and Zinc Plated - Fully drop forged. Used for Morse-taper shanks from MT sleeves, collets or spindles.

<table>
<thead>
<tr>
<th>Size</th>
<th>H</th>
<th>Length, L</th>
<th>MT Sockets $</th>
<th>Sleeves N°</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.80</td>
<td>4.6</td>
<td>1</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>0.80</td>
<td>5.6</td>
<td>2</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>1.0</td>
<td>6.2</td>
<td>3</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>1.25</td>
<td>7.0</td>
<td>4 - 5 - 6</td>
<td></td>
</tr>
<tr>
<td></td>
<td>Set of 4</td>
<td>1 - 6</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
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.

![Vise and Drill Press](image1.png)

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.

![Clamp Sets](image2.png)

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

![V-Blocks](image3.png)

**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.

![Angle Plates and Parallels](image4.png)
Drilling calculations:

The speed of a drill 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 \text{sfm}. Every tool manufacturer has a recommended table of \text{sfm values} for their tools.

The peripheral and rotational velocities of the tool are related as shown in the following equation:

\begin{align*}
1) \quad V, V_c &= \pi * D * N
\end{align*}

Where:
\begin{align*}
V, V_c &= \text{recommended peripheral velocity for the tool being used (ft/min), (m/min)}; \\
D &= \text{diameter of the drill bit (ft) (m)}; \\
N &= \text{rotational velocity of the tool}.
\end{align*}

The peripheral velocity is commonly expressed in units of \text{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 \text{ft/min units}), \(N\) is found in the following manner:

\begin{align*}
2) \quad N \text{ [rpm]} &= \frac{12 \text{ [in/ft]} * V_c \text{ [ft/min]}}{\pi * D \text{ [in]}} = \frac{V_c * 4}{D}
\end{align*}

Where:
\begin{align*}
V, V_c &= \text{Cutting speed (ft/min)}; \\
4 &= \text{Constant for calculation for RPM, (12 / \pi) using velocity in ft/min (except metric)}; \\
D &= \text{Diameter of the drill bit (in inches)}.
\end{align*}

\textbf{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 \text{ ft/min}. The formula would be set up as follows:

\begin{align*}
N &= \frac{200 \times 4}{0.500} = \frac{800}{0.500} = 1600 \text{ RPM}
\end{align*}

The metrical peripheral velocity (\(V\) in \text{m/min}), \(N\) is found in the following manner:

\begin{align*}
3) \quad N &= \frac{V_c \text{ (m/min)} \times 320}{D \text{ (mm)}}
\end{align*}

Where:
\begin{align*}
V, V_c &= \text{Recommended peripheral speed (in m/min)}; \\
320 &= \text{A constant for all metric RPM (1000 / \pi) calculations (when} V_c \text{ is in m/min)}; \\
D &= \text{Diameter of the twist drill in millimeters (or mm)}.
\end{align*}

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

\begin{align*}
4) \quad N &= \frac{V_c \text{ (m/min)} \times 320}{D \text{ (mm)}}
\end{align*}

\begin{align*}
N &= \frac{21.4 \text{ (m/min)} \times 320}{15 \text{ (mm)}} = 456.533 \text{ RPM or 457 RPM}
\end{align*}

\textbf{Note:} The factors 4 and 320 used in formulae are the approximated results (from 12 dividing \(\pi\) and 1000 dividing \(\pi\)), 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:

\[
5) \quad Cs = N \times fR
\]

Where:

- \( Cs \) = 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):**

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

**Recommended average feed rates for 2 Flute HSS Drills:**

<table>
<thead>
<tr>
<th>Drill Diameter [in.]</th>
<th>Feedrate, ( fR ) [in/rev]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Under 1/8&quot;</td>
<td>up to 0.002</td>
</tr>
<tr>
<td>1/8&quot; to 1/4&quot;</td>
<td>0.002 to 0.004</td>
</tr>
<tr>
<td>5/16&quot; to 1/2&quot;</td>
<td>0.005 to 0.008</td>
</tr>
<tr>
<td>5/8&quot; to 3/4&quot;</td>
<td>0.009 to 0.011</td>
</tr>
<tr>
<td>1&quot; and over</td>
<td>0.012 to 0.020</td>
</tr>
</tbody>
</table>

**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.

\[
N \ [rpm] = 12 \times \frac{V}{(\pi \times D)} = 12 \times \frac{100 \text{ ft/min}}{(\pi \times 0.25)} = 1528 \text{ rpm (use 1500 rpm)} - 50\% \text{ less = 750 rpm}
\]

\[
N \ [rpm] = 4 \times \frac{V}{D} = 4 \times \frac{100 \text{ ft/min}}{0.25} = 1600 \text{ rpm} - 50\% \text{ less = 800 rpm}
\]

Always choose the best velocity option according to working material. From the table above, lookup the minimum recommended feed per revolution for a 1/4" HSS drill bit:

\( fR = \sim 0.002 \text{ in/rev} \)

\[
Cs \ [\text{in/min}] = N \ [\text{rpm}] \times f \ [\text{in/rev}]
\]

\[
Cs \ [\text{in/min}] = 1500 \text{ rev/min} \times 0.002 \text{ in/rev} = 3.0 \text{ in/min} - 50\% \text{ less = 1.5 in/min}
\]
6) Cutting Time (min):

\[
Ct = \frac{(L + A)}{N \times f_R} \quad \text{or} \quad Ct = \frac{\pi \times D \times (L_w + 2A_1)}{12 \times V \times f_R} \quad \text{or} \quad Ct = \frac{\pi \times D \times (L_w + 2A_1)}{1000 \times V \times f_R}
\]

Where:

- \(A\) (Allowance) = \(\sim D/2\);
- \(A_1\) = approach and overrun (0.062 ~ 0.18 in) - (~ 2 to 5 mm)
- \(f_R\) = feed per revolution [in/rev] or [mm/rev] – see recommended tables
- \(V, V_c\) = peripheral velocity = \(\pi \times D \times N\) [ft/min] – [m/min]
- \(L, L_w\) = Length of hole (in) or (mm) – [*Attention to conversions feet, inches, meters or millimeters].

7) Material Removal Rate (MRR):

\[
MRR = \frac{\text{Volume Removed}}{\text{Cutting Time}} (\text{ft}^3/\text{min}) (\text{mm}^3/\text{min}) = \frac{\pi \times L \times D^2 \times f_R \times N}{4 \times L^4} = \frac{\pi \times D^2 \times f_R \times N}{4}
\]

**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:**

<table>
<thead>
<tr>
<th>Workpiece material</th>
<th>Hardness [Bhn]</th>
<th>Cutting material</th>
<th>Cutting speed (V_c) [ft/min]</th>
<th>Cutting speed (V_c) [m/min]</th>
<th>Feed rate per revolution (f) [inch]</th>
<th>Feed rate per revolution (f) [mm]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cast irons</td>
<td>190...320</td>
<td>High speed steel</td>
<td>MIN: 33 MAX: 295 MIN: 10 MAX: 90</td>
<td>MIN: 0.002 MAX: 0.008 MIN: 0.050 MAX: 0.200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Steel - plain carbon</td>
<td>85...200</td>
<td>High speed steel</td>
<td>MIN: 49 MAX: 148 MIN: 15 MAX: 45</td>
<td>MIN: 0.002 MAX: 0.008 MIN: 0.050 MAX: 0.200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Steel - alloys</td>
<td>35...50Rc</td>
<td>High speed steel</td>
<td>MIN: 18 MAX: 65 MIN: 5 MAX: 20</td>
<td>MIN: 0.002 MAX: 0.008 MIN: 0.050 MAX: 0.200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Steel - tool</td>
<td>50...58Rc</td>
<td>High speed steel</td>
<td>MIN: 18 MAX: 65 MIN: 5 MAX: 20</td>
<td>MIN: 0.002 MAX: 0.008 MIN: 0.050 MAX: 0.200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Steel - stainless</td>
<td>150...450</td>
<td>High speed steel</td>
<td>MIN: 16 MAX: 33 MIN: 5 MAX: 10</td>
<td>MIN: 0.002 MAX: 0.008 MIN: 0.050 MAX: 0.200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Aluminum alloys</td>
<td>30...150</td>
<td>High speed steel</td>
<td>MIN: 18 MAX: 377 MIN: 5 MAX: 115</td>
<td>MIN: 0.002 MAX: 0.008 MIN: 0.050 MAX: 0.200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Copper alloys</td>
<td>80...100Rb</td>
<td>High speed steel</td>
<td>MIN: 68 MAX: 230 MIN: 20 MAX: 70</td>
<td>MIN: 0.002 MAX: 0.008 MIN: 0.050 MAX: 0.200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Nickel alloys</td>
<td>80...380</td>
<td>High speed steel</td>
<td>MIN: 33 MAX: 65 MIN: 10 MAX: 20</td>
<td>MIN: 0.002 MAX: 0.008 MIN: 0.050 MAX: 0.200</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Titanium</td>
<td>250...375</td>
<td>High speed steel</td>
<td>MIN: 18 MAX: 49 MIN: 5 MAX: 15</td>
<td>MIN: 0.002 MAX: 0.008 MIN: 0.050 MAX: 0.200</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
### Recommended values for gun drilling – carbide tools:

<table>
<thead>
<tr>
<th>Workpiece material</th>
<th>Cutting speed [Vc (feet/min) / Vc (m/min)]</th>
<th>Gun Drill Diameters [inch]</th>
<th>Feed - f [inch/revolution]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>[Bhn]</td>
<td>MIN</td>
<td>MAX</td>
</tr>
<tr>
<td>Cast Iron soft</td>
<td>120-220</td>
<td>220</td>
<td>350</td>
</tr>
<tr>
<td>Cast Iron hard</td>
<td>220-320</td>
<td>150</td>
<td>200</td>
</tr>
<tr>
<td>Ductile Iron</td>
<td>140-260</td>
<td>220</td>
<td>300</td>
</tr>
<tr>
<td>Malleable Iron</td>
<td>110-240</td>
<td>250</td>
<td>350</td>
</tr>
<tr>
<td>Steel - soft</td>
<td>65-220</td>
<td>425</td>
<td>675</td>
</tr>
<tr>
<td>Steel - Medium</td>
<td>200-325</td>
<td>225</td>
<td>450</td>
</tr>
<tr>
<td>Steel - Hard</td>
<td>325-450</td>
<td>150</td>
<td>200</td>
</tr>
<tr>
<td>Stainless Steel-Soft</td>
<td>135-275</td>
<td>250</td>
<td>300</td>
</tr>
<tr>
<td>Stainless Steel-Hard</td>
<td>275-425</td>
<td>150</td>
<td>225</td>
</tr>
<tr>
<td>Aluminum alloys, except Die casting</td>
<td></td>
<td>650</td>
<td>198</td>
</tr>
<tr>
<td>Aluminum Die casting</td>
<td></td>
<td>650</td>
<td>198</td>
</tr>
<tr>
<td>Magnesium</td>
<td></td>
<td>650</td>
<td>198</td>
</tr>
<tr>
<td>Brass and Bronze</td>
<td>850</td>
<td>950</td>
<td>150</td>
</tr>
<tr>
<td>Copper</td>
<td>350</td>
<td>450</td>
<td>107</td>
</tr>
</tbody>
</table>

### Common RPM and Feed Rate

<table>
<thead>
<tr>
<th>Dia.</th>
<th>1/16&quot;</th>
<th>1/8&quot;</th>
<th>3/16&quot;</th>
<th>1/4&quot;</th>
<th>5/16&quot;</th>
<th>3/8&quot;</th>
<th>1/2&quot;</th>
<th>5/8&quot;</th>
<th>3/4&quot;</th>
<th>1&quot;</th>
</tr>
</thead>
<tbody>
<tr>
<td>RPM</td>
<td>6000</td>
<td>3000</td>
<td>2000</td>
<td>1450</td>
<td>1150</td>
<td>975</td>
<td>725</td>
<td>600</td>
<td>500</td>
<td>375</td>
</tr>
<tr>
<td>HP</td>
<td>0.01</td>
<td>0.04</td>
<td>0.11</td>
<td>0.18</td>
<td>0.28</td>
<td>0.38</td>
<td>0.57</td>
<td>0.80</td>
<td>1.10</td>
<td>1.75</td>
</tr>
<tr>
<td>Feed</td>
<td>0.001</td>
<td>0.002</td>
<td>0.004</td>
<td>0.005</td>
<td>0.006</td>
<td>0.007</td>
<td>0.008</td>
<td>0.009</td>
<td>0.010</td>
<td>0.012</td>
</tr>
</tbody>
</table>

### 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  | 270  | 360 |
| 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. With various attachments, milling machines can be used for boring, slotting, facing, drilling, keyways, racks, gears, tapping, reaming and cutting workpieces.

Milling Operation

The basic function of milling machines, including the operations shown in the pictures above and below, is to produce flat surfaces in any orientation, as well as, surfaces of revolution, helical surfaces and contoured surfaces of various configurations. Such functions are accomplished by slowly feeding the workpiece into the equispaced multiedge circular cutting tool rotating at moderately high speed as indicated below. 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.
Classification of milling machines:

(a) According to nature of purposes of use: General purpose – commonly 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.

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 work table 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.

Ram-type: Characterized by 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 work-table 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 arbour. The work-table 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 arbour knee type machines** but possess one additional swiveling motion of the worktable.

**Universal swivel head type:** Is a versatile milling machine and not only possess both horizontal milling arbour 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, due to the fact that the cutter and the surface being cut are in plain view. The table may or may not have swivelng features.

Note: According to mechanization / automation and production rate milling machines are mostly general purpose and used for pieces or small lot production. But 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.

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 master piece.
Computer Numerical Controlled (CNC): Flexible automation by developing and using CNC has made a great breakthrough 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.
Machining centers:

Are programmed machines commanded by a computer numerical control (CNC), in which computers play an integral part of the control. 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.

The programs produce a computer file that 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".

In other 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. In either case, the series of steps needed to produce any part is highly automated and produces a part that closely matches the original CAD design.
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. The feed change gearing is enclosed within the knee. It is supported and can be adjusted by the elevating screw. The knee is fastened to the column by dovetail ways. 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 only) is attached to the saddle and can be swiveled approximately 45° in either direction.

(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 table is the rectangular casting located on top of the saddle. It contains several T-slots for fastening the work or workholding devices. The table can be moved by hand or by power. To move the table by hand, engage and turn the longitudinal hand crank. To move it by power, engage the longitudinal directional feed control lever. The longitudinal directional control lever can be positioned to the left, to the right, or in the center. Place the end of the directional feed control lever to the left to feed the table to the left. Place it to the right to feed the table to the right. Place it in the center position to disengage the power feed, or to feed the table by hand.

(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 is the horizontal beam to which the arbor support is fastened. 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: The arbor support is a casting containing a bearing which aligns the outer end of the arbor with the spindle. This 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.

(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
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. Spring collets are 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.

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 work-table 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 work table and tilting the vertical milling spindle about X and/or Y axes, as shown below.
Milling cutters description:

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.

Milling cutters are classified as:

(a) Profile sharpened cutters – where the geometry of the machined surfaces is not related with the tool shape:

- slab or plain milling cutter;
- straight or helical fluted;
- side milling cutters – single side or both sided type;
- slotting cutter;
- slitting or parting tools;
- end milling cutters – with straight or taper shank face milling cutters.

(b) Form relieved cutters – where the job profile becomes the replica of the tool-form:

- form cutters;
- gear (teeth) milling cutters;
- spline shaft cutters;
- tool form cutters;
- T-slot cutters;
- thread milling cutter.

Types of Milling Cutters:

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.

Side and slot 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.
**Slotting 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 or parting tools:** 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.

![Diagram of slotting milling cutters and slitting saw](image)

**End milling cutters or end mills:** The shape and the common applications of end milling cutters (profile sharpened type) are shown below. The common features and characteristics of such 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 arbour as shown in the same figure.

![Diagram of end milling cutters and shell milling](image)
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.

Tool form cutters: are generally used for making grooves or slots of various profiles 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.

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.
**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](image)

**Thread milling cutters**: Such shank type 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. Both internal and external threads are cut by the tool as shown below.

**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 arbour as shown below.

![Short thread milling Long thread milling Straddle milling cutter](image)

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

**Turning by rotary milling cutters**: During turning like operations in 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 Turning by rotary milling cutters](image)
Ball-nose end milling cutters (or ball like hemispherical end): 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](image)

**Milling Calculations:**

The speed of a mill cutter 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:

\[ 1) \quad V = \pi \times d \times N \]

Where:

- \( V \) = recommended peripheral velocity for the tool being used
- \( d \) = diameter of the mill cutter
- \( N \) = rotational velocity of the tool.

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:

\[ 2) \quad N [\text{rpm}] = \frac{12 \ [\text{in/ft}] \times V [\text{sfm}]}{\frac{\pi \times d [\text{in/rev}]}{d}} = C_s \times 4 = \]

Where:

- \( C_s \) = cutting speed (ft/min);
- \( 4 \) = a constant in calculation for RPM, using velocity in ft/min (except metric);
- \( d \) = diameter of the mill cutter (in inches).

**Example:** A mill face 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)** of measurement, a slight different formula is used to find the RPM:

3) \[ N = \frac{C_S \text{ (m/min)}}{D \text{ (mm)}} \times 320 \]

Where:

- \( C_S \) = 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:

4) \[ N = \frac{C_S \text{ (m/min)}}{D \text{ (mm)}} \times 320 \]

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

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

5) \[ C_S = N \times f_t \times n \]

Where:

- \( C_S \) = Cutting speed or feedrate [in/min];
- \( N \) = spindle speed [rpm];
- \( f_t \) = feed per tooth [in/tooth];
- \( n \) = number of teeth considered per revolution of the cutter;

<table>
<thead>
<tr>
<th>Recommended feeds per tooth (ft) for HSS Milling Cutters:</th>
<th>Recommended Peripheral Velocity (V) for HSS Milling Cutters:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Material</td>
<td>Feed, ( f_t ) [in/tooth]</td>
</tr>
<tr>
<td>Aluminum</td>
<td>0.004 to 0.012</td>
</tr>
<tr>
<td>Magnesium</td>
<td>0.004 to 0.012</td>
</tr>
<tr>
<td>Brass (soft)</td>
<td>0.004 to 0.012</td>
</tr>
<tr>
<td>Brass (hard)</td>
<td>0.002 to 0.010</td>
</tr>
<tr>
<td>Bronze (soft)</td>
<td>0.004 to 0.012</td>
</tr>
<tr>
<td>Bronze (hard)</td>
<td>0.002 to 0.010</td>
</tr>
<tr>
<td>Copper</td>
<td>0.004 to 0.012</td>
</tr>
<tr>
<td><strong>Steel</strong></td>
<td></td>
</tr>
<tr>
<td>100 Bhn</td>
<td>0.004 to 0.006</td>
</tr>
<tr>
<td>200 Bhn</td>
<td>0.004 to 0.005</td>
</tr>
<tr>
<td>300 Bhn</td>
<td>0.003 to 0.004</td>
</tr>
<tr>
<td>400 Bhn</td>
<td>0.002 to 0.003</td>
</tr>
<tr>
<td>500 Bhn</td>
<td>0.001 to 0.002</td>
</tr>
<tr>
<td><strong>Stainless</strong></td>
<td></td>
</tr>
<tr>
<td>Stainless (hard)</td>
<td>0.002 to 0.004</td>
</tr>
<tr>
<td>Titanium (soft)</td>
<td>0.002 to 0.005</td>
</tr>
<tr>
<td>Titanium (hard)</td>
<td>0.001 to 0.003</td>
</tr>
</tbody>
</table>
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:

\[ Cs, V = \sim 250 \text{ ft/min} \]

Next, calculate the spindle speed:

\[ N [\text{rpm}] = \frac{12 * V}{(\pi * D)} \]
\[ N = \frac{12 \text{ in/ft} * 250 \text{ ft/min}}{(\pi * 1.0)} \]
\[ N = 950 \text{ rpm} \]

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

\[ ft = \sim 0.008 \text{ in/tooth} \text{ (see table above):} \]

\[ Cs[\text{in/min}] = N [\text{rpm}] x ft [\text{in/tooth}] x n \]
\[ Cs = 950 \text{ rpm} * 0.008 \text{ in/tooth} * 4 \text{ teeth/rev} \]
\[ Cs = 30 \text{ in/min} \]

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

\[ Cs = 30 \times 0.6 = 18 \text{ in/min (final ans).} \]

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 arbour;
- 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, \( T_c \) for a plain milling flat surfaces can be determined as:

a) \[ T_c = \frac{L_c}{S_m} \]

Where:

\[ L_c = \text{Total length of piece [in; mm];} \]
\[ S_m = \text{Mill table feed [in/min; mm/min];} \]

Consider the length \( L_c \) as:

b) \[ L_c = L_w + 2A + \frac{d}{2} \]

Where:

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

Consider the table feed \( S_m \) as:

c) \[ S_m = ft * n * N \]
Where:

$ft = \text{feed per tooth (ft/tooth, mm/tooth)}$;

$n = \text{number of teeth}$;

$N = \text{Rotational speed, rpm}$.

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:**

<table>
<thead>
<tr>
<th>To find</th>
<th>Having</th>
<th>Formula</th>
<th>Work Material</th>
<th>C (Constant)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Peripheral cutting speed - $V$</td>
<td>Cutter diameter, d</td>
<td>$V, Cs = \pi.d.N$</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Rotational speed, N</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Rotational speed - $N$</td>
<td>Cutter diameter, d</td>
<td>$N = (Cs * 4) / d \ (\text{in})$</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Peripheral cutting speed, V</td>
<td>$N = Cs * 320/d \ (\text{mm})$</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>$N = (12 * V) / \pi * d \ (\text{in})$</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td>$N = 1000 * V / \pi * d \ (\text{mm})$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cutting speed - $Cs$</td>
<td>Rotational speed, N</td>
<td>$Cs = N * d/4\ (\text{in})$</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Cutter diameter, d</td>
<td>$Cs = N * d/320 \ (\text{mm})$</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Feed per tooth - $ft$</td>
<td>Machine feed rate, F</td>
<td>$ft = Fr/N * n$</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Rotational speed, N</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Number of teeth, n</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Machine feed rate - $Fr$</td>
<td>Feed per tooth, ft</td>
<td>$Fr = N * ft * n$</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Rotational speed, N</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Number of teeth, n</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Feed per revolution - $f$</td>
<td>Machine feed rate, F</td>
<td>$f = Fr/N$</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Rotational speed, N</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cutting power input - $P$</td>
<td>Width of cut, W</td>
<td>$P = (W * h * F)/C$</td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Depth of cut, h</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Machine feed rate, F</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td>Workpiece material power constant, C</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Note:** The formulae below are also used:

$\text{SFM} = 0.262 \times d \times N$ - (where, SFM - Surface Feet per Minute = Cs)

$\text{RPM} = (3.82 \times \text{SFM}) / d$ – (where, RPM = N) - 3.82 is a constant derived from 12/pi - feet to diameter in inches.

$\text{IPR} = \text{IPM} / \text{RPM}$ or $\text{CHIP LOAD} \times F$ – (where, IPR – Inches per Revolution);

$\text{IPM} = \text{RPM} \times \text{IPR}$ – (where, IPM – Inches per Minute)

$\text{CHIP LOAD} = \text{IPM} / (\text{RPM} \times F)$ or $\text{IPR} / F$ – (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 $A = 5 \ \text{mm}$, $V = 40 \ \text{m/min}$ and $ft = 0.1 \ \text{mm/tooth}$.

$$Tc = \frac{Lc}{Sm}$$

$Lc = Lw + 2A + d/2 = 100 + (2*5) + 30 = 140 \ \text{mm}$

$N = 1000 \times V / \pi \times d = 1000 \times 40 / \pi \times 60 = 212 \ \text{rpm}$ – use $200 \ \text{rpm}$

$Sm = ft \times n \times N = 0.1 \times 6 \times 200 = 120 \ \text{mm/min}$

$Tc = Lc / Sm = 140 / 120 = 1.17 \ \text{min}$.
<table>
<thead>
<tr>
<th>Material</th>
<th>Hardness, Brinell</th>
<th>End Mills</th>
<th>Depth of Cut 0.250&quot;</th>
<th>Face Mills and Shell End Mills</th>
<th>Plain or Slab Mills</th>
<th>Side Mills</th>
<th>Form Relieved Cutters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Plain Carbon Steels, AISI 1010 to 1030</td>
<td>150 to 200</td>
<td>3/8</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .012</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>AISI B1111, B1112, B1113</td>
<td>140 to 180</td>
<td>3/4</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .005</td>
<td></td>
</tr>
<tr>
<td>Plain Carbon Steels, AISI 1040 to 1095</td>
<td>120 to 180</td>
<td>1 - up</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>All Alloy Steels Having 3% Carbon Content or Less</td>
<td>180 to 220</td>
<td>3/8</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>All Alloy Steels Having More Than 3% Carbon Content</td>
<td>180 to 220</td>
<td>1 - up</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>Tool Steel</td>
<td>200 to 250</td>
<td>3/4</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>Cast Iron</td>
<td>150 to 180</td>
<td>3/4</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>Zinc Alloys</td>
<td>100 to 150</td>
<td>3/4</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>Cast Aluminum Alloy–as Cast</td>
<td>80 to 100</td>
<td>3/4</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>Cast Aluminum Alloy–Hardened</td>
<td>80 to 100</td>
<td>3/4</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>Wrought Aluminum Alloy–Cold Drawn</td>
<td>80 to 100</td>
<td>3/4</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>Wrought Aluminum Alloy–Hardened</td>
<td>80 to 100</td>
<td>3/4</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>Magnesium Alloys</td>
<td>80 to 100</td>
<td>3/4</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>Ferritic Stainless Steel</td>
<td>80 to 100</td>
<td>3/4</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>Austenitic Stainless Steel</td>
<td>80 to 100</td>
<td>3/4</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>Martensitic Stainless Steel</td>
<td>80 to 100</td>
<td>3/4</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
<tr>
<td>Plastics</td>
<td>80 to 100</td>
<td>3/4</td>
<td>.001 .002 .004 .006</td>
<td>.001 .002 .004 .006</td>
<td>.012 .010</td>
<td>.008 .008 .008 .004</td>
<td></td>
</tr>
</tbody>
</table>
Planing Machines & Planers:

Planing machines are also basically used for producing flat surfaces in different planes. 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 also;

- 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 motion 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:

The application for a patent was achieved by James Coulter and Herbert Harpin of West Yorkshire. 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.

**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.

- 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.

There are two types of planers for metal machining: double-housing and open-side. The double-housing variety has vertical supports on both sides of its long bed; the open-side variety has a vertical support on only one side.

**Planing machine and planer tools:**
The cutting tools used on planers are all single point cutting tools. They are in general similar in shapes and tool angles to those used on a lathe and shaping machine. 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.

Planer tools may be solid, forged type or bit type. 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 tool bits are made of high speed side, allowing the workpiece to extend beyond the bed. Steel, stellite or cemented carbide and they may be brazed, welded or clamped on a mild steel shank. Cemented carbide tipped tools is used for production work.

The typical tools used in a planer are:

- Right hand round nosed roughing tool for cast iron;
- Right hand round nosed roughing tool for steel;
- Square nosed side facing roughing tool for cast iron;
- Goose-neck finishing tool for cast iron and steel;
- Left hand dovetail end cutting roughing tool for cast iron.

**Cutting calculations:**

- **Feed:** is the distance of the tool head traveling at each cutting stroke, expressed in inches (mm) per double stroke.
- **Cutting Depth:** is the 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:** cutting speed, feed, length of cutting stroke, changing tools, number of double strokes per minute, and the machining time required for every complete cut are known; cutting time for a complete operation may be calculated.
- **Cutting Speed:** is the 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:

\[
Cs = \frac{n.L.(1 + Q)}{1000} \text{ (m/min) } \text{ or } \frac{n.L.(1 + Q)}{12} \text{ (ft/min)}
\]

Where:
- \(n\) = 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:

Just like a planer or planing machine, the main functions of shaping machines are to produce flat surfaces in different planes. As referred before, shaper and planer are both reciprocating machine tools intended 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.

The vertical in feed is given either by descending the tool holder or raising the bed or both. 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.
It is important to be aware that shaping machines are neither productive nor versatile. However, its limited applications include:

Machining flat surfaces in different planes as shown below;
Making features like slots, steps etc. which are also bounded by flat surfaces;
Forming grooves bounded by short width curved surfaces by using single point but form tools;
Cutting external keyway and splines, smooth slitting or parting, cutting teeth of rack for repair, etc.;

However, due to very low productivity, less versatility and poor process capability, shaping machines are not employed for lot and even batch production. Such low cost primitive machine tools may be reasonably used only for little or few machining work on one or few pieces required for repair and maintenance work in small machine shops.

**Slotting Machines:**

Slotting machines are technically vertical shapers, but essential distinction can be made by the true vertical shaper, whose slide can be moved from the vertical, as the slotting tool is fixed on a vertical plane. 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 work table. A longitudinal and cross feeds and a rotary feed motion is also provided in the work table.
The intermittent rotation of the feed rod is derived from the driving shaft with the help of a four bar linkage as shown in the kinematic diagram, shown above. 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 work table.

The working speed, number of strokes per minute, may be changed, if necessary, by changing the belt-pulley ratio or using an additional “speed gear box”. The feed values are changed mainly by changing the amount of angular rotation of the feed rod per stroke of the tool, by adjusting the amount of angle of oscillation of the paul, simply by rotating the tapered paul by 180° as done in shaping machines.

Slotting machines are very similar to shaping machines in respect of machining principle, tool-work motions and general applications, however, characterized by:

- Vertical tool reciprocation with down stroke acting;
- Longer stroke length;
- Less strong and rigid;
- An additional rotary feed motion of the work table;
- Used mostly for machining internal surfaces.

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 and possible machining 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 from:

\[ \frac{V_c}{1000} = \frac{N_s}{L} (1 + Q) \]  m/min

Therefore:

\[ N_s = \left( \frac{1000V_c}{L} \right) \left( 1 + \frac{Q}{L} \right) \]

Where,

\( V_c \) = cutting velocity, m/min;
\( N_s \) = number of strokes per min;
\( L \) = length of the workpiece;
\( L_L \) = length of the stroke, mm = \( L + 2A \) (see sketch);
\( W \) = width of workpiece;
\( L_w \) = 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:

\( L = 100 \text{ mm}, A = 5, \)
\( W = 60, B = 2 \)
\( Q = \text{time of return stroke divided by time of cutting stroke} = 2/3 \)
\( V_c = 40 \text{ m/min} \)
\( S_o = \text{feed of the job} = 0.2 \text{ mm/stroke} \)

\[ N_s = \left( \frac{1000V_c}{L} \right) \left( 1 + \frac{Q}{L} \right) \]
\[ N_s = (1000 \times 40) / [(100 + 5 + 5)(1 + 2/3)] = 218 \text{ strokes/min.} \]

Using the formulae above, the \textbf{total machining time, } T_c, \text{ can be determined form the expression:}
Where:

\[ T_c = \frac{L_w}{N_s S_0} \text{ min} \]

\[ \begin{align*}
T_c &= \text{Total machining time, minutes; } \\
L_w &= \text{total width of workpiece} = W + 2B \\
N_s &= \text{Number of strokes/minute} \\
S_0 &= \text{feed of the job} = \text{mm/stroke.}
\end{align*} \]

Example:

Determine the total machining time for same workpiece indicated on example above, using \( N_s = 200 \):

\[ \begin{align*}
T_c &= \text{Total machining time, minutes; } \\
L_w &= 60 + 2 \times 2 \\
N_s &= \text{Number of strokes/minute} = 200 \\
S_0 &= \text{feed of the job} = 0.2 \text{ mm/stroke.}
\end{align*} \]

\[ T_c = (60 + 2 + 2) = 1.6 \text{ minutes} \]

\[ \frac{(0.2 \times 200)}{ } \]

Boring Machines:

Boring is commonly known by 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.

The original hole is made with a drill, or it may be a cored hole in a casting. Boring achieves three things:

- Boring brings the hole to the proper size and finish. 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 also be 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. These parts are usually somewhat rectangular in shape, though they may be asymmetrical or irregular. The available cutting tools only limit the size of cut, the rigidity of the spindle, and the available horsepower.

A horizontal boring machine has its work spindle parallel to the ground and work table. Typically there are 3 linear axes in which the tool head and part move. 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 centre line 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. The runway must be carefully aligned and leveled when it is first installed, and then checked at intervals as the machine is used.

**Vertical Boring Machined (VBM):** A vertical boring machine is like a lathe with the headstock resting on the floor. This machine is needed because even the largest engine lathes cannot handle work much over **24 inches in diameter**. Today's VBMs are often listed as turning, facing and boring machines. These machines can make only round cuts plus facing and contouring cuts.

**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.

**Boring types:**

**Lineboring:** (or line-boring) implies the conventional machines. **Backboring** (or back-boring) is the process of reaching through an existing hole and then boring on the "back" side of the workpiece (relative to the machine headstock).

**Universal Boring and Facing Heads:** A universal boring and facing head will carry out the same boring operations as a standard, manually adjusted. The shapes vary with the different metals that are being worked, and also with the class of work performed. The boring process can also be executed on various other machine tools:

- Universal machines, such as lathes (turning centers) or milling machines (machining centers);
- In horizontal boring mills, the workpiece sits on a table while the boring bar rotates around a horizontal axis, as a specialized horizontal milling machine.

**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).
Micro carbide tools for small bores: These 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.).

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 described 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:

\[ N = \frac{k \cdot V}{\pi \cdot D_1} \]

\( V = \text{surface feet per minute} \), \( D_1 \) in inches: \( k = 12 \);
\( V = \text{millimeters per minute} \), \( D_1 \) in mm: \( k = 320 \);
N = tool rotation (rpm);
D₁ = finished (larger) diameter (see sketch).

B.) Cutting Time:

\[ Ct = \frac{(L_w + A)}{f_r \cdot N} \text{ or } Ct = \frac{\pi \cdot D_1 \cdot (L_w + A)}{12 \cdot V \cdot f_r} \]

Where:

\[ L_w = \text{Length of hole (in) or (mm)} - [* \text{attention to conversions feet, inches, meters or millimeters}] \]

\[ A = \text{Approach and over run} = D_1/2; \]

\[ f_r = \text{feedrate (in/min) or (mm/min)}; \]

\[ f_r = \text{feed (in/rev) (mm/rev)}. \]

C.) Material Removal Rate:

\[ MRR = \frac{\text{Volume Removed}}{\text{Cutting Time}} \left( \frac{ft^3/min}{mm^3/min} \right) = \frac{\pi \cdot L \cdot D_2^2 \cdot f_r \cdot N}{4 \cdot L} = 0.7854 \left( \frac{D_2^2 \cdot f_r \cdot N}{4} \right) \]

Where:

\[ D_1 = \text{finished diameter (in) (mm)}; \]

\[ D_2 = \text{initial (smaller) diameter}; \]

\[ f_r = \text{feedrate (ft/rev) (in/rev) (mm/rev)}; \]

\[ t = \text{width of the cutting tool (in) (mm)}; \]

\[ N = \text{tool rotation (rpm)} \]

\[ L = \frac{D_1}{2} \text{ for solids} - \frac{D_2 - D_1}{2} \text{ for tubular workpieces} \]

Note: Excessive heat at the tool point will result in premature failure. Therefore, 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 tool diameters.

Hobbing & Gear Shaping Machines:

Gear hobbing, as any cutting process based on the rolling principle, is a complicated gear fabrication method. Although a variety of simulating methods has been proposed, each of them somehow reduces the actual three-dimensional (3D) process to planar models, primarily for simplification reasons. The paper describes an effective and
factual simulation of gear hobbing, based on virtual kinematics of solid models representing the cutting tool and the work gear.

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. Compared to other gear forming processes it is relatively inexpensive but still quite accurate, thus it is used for a broad range of parts and quantities. It is the most widely used gear cutting process for creating spur and helical gears and more gears are cut by hobbing than any other process since it is relatively quick and inexpensive.

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. Top-speed direct drives, high-precision hobbing heads, integrated solutions and reliable automation are considered as standards.

Moreover, the latest-generation gear cutting tools made from the most “avant-garde” materials. 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.

Modern hobbing machines, also known as hobbers, 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. 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.

Most hobs are single-thread hobs, but double or triple-thread hobs increase production rates. The down-side is that they are not as accurate as single-thread hobs. Depending on type of gear teeth to be cut, there are custom made hobs and general purpose hobs. Custom made hobs are different from other hobs as they are suited to make gears with modified tooth profile. The tooth profile is modified to add strength and reduce size and noise of gears.

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.
Common gear generation types:

1. **Face gear**: This is a gear that is limited to 900 intersecting axes. 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**: This is a deviation from a bevel gear that 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. It looks very much like the spiral bevel gear. 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:

\[
dp = d \cos x \alpha \quad \text{Where: } d \text{ and } \alpha \text{ are the standard values, or alternately.}
\]

\[
dp = d' \cos x \alpha' \quad \text{Where: } d' \text{ and } \alpha' \text{ are the exact operating values.}
\]

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. It is for this reason that 14.5º gears encounter greater undercutting problems than 20º gears.

**Proper meshing and contact ratio:**

**Standard gears meshing:** The contact point of the two involutes, as figure below shows, slides along. The common tangent of the two base circles as rotation occurs. The common tangent is called the line-of-contact, or line-of-action.

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 \(P_b\). The pressure angles must be identical becomes obvious from the following equation for base pitch:

\[
P_b = \pi m \cos \alpha
\]

**Pressure angles:** when are different, the base pitches cannot be identical. The length of the line-of-action is shown below.

**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, the 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.

The principal motions involved in rotary gear shaper cutting are of the following:

- **Cutting Motion**: Is the downward linear motion of the cutter spindle together with the cutter.
- **Return Stroke**: Is the upward linear travel of the spindle and cutter to its starting position.
- **Indexing Motion**: Slow speed continuous rotation of the cutter spindle and work spindle to provide circular feed.
- **Speed**: The two speeds are regulated considering for each rotation of the cutter. The gear revolves through \( n/N \) revolutions, where "n" is the number of teeth of the cutter, and "N" is the number of teeth to be cut on the workpiece.
- **Completion of Cutting Operation**: The indexing and reciprocating motions continue until the required numbers of teeth to the required depth are cut all along the periphery of the gear blank.

**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.

In rotary broaching, the broach is rotated and pressed into the workpiece to cut an axis symmetric shape. A rotary broach is used in a lathe or screw machine. In both processes, the cut is performed in one pass of the broach, which makes it very efficient.

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. Even though broaches can be expensive, broaching is usually favored over other processes when used for high-quantity production runs.

Broaches are shaped similar to a saw, except the teeth height increases over the length of the tool. Moreover, the broach contains three distinct sections: one for roughing, another for semi-finishing, and the final one for finishing. Broaching is an unusual machining process because it has the feed built into 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 con-
sidered 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

There are different types of broaching machines which are broadly classified by:

- According to nature of work:
  - internal broaching
  - external (surface) broaching

- According to configuration:
  - horizontal
  - vertical

- According to number of slides or stations:
  - single station type
  - multiple station type
  - indexing type

**Horizontal broaching machines:**

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. Whereas, broaching enables remove the whole material in one stroke only by the gradually rising teeth of the cutter called broach. The amount of tooth rise between the successive teeth of the broach is equivalent to the inch feed given in shaping.

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**. Most of the time is consumed by the return stroke, broach handling, and work-piece loading and unloading. 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.

![Diagram of broaching tools](image)

Machining by broaching is preferably used for making straight through **holes of various forms and sizes of section**, **internal and external** through straight or helical slots or grooves, external surfaces of different shapes, teeth of external and internal splines and small spur gears etc. Above and below (schematically) is **shown how a through hole** is enlarged and finished by broaching.

![Diagram of broaching process](image)

Surface finishes are usually between **16 and 63 microinches (μin)**, but can range from **8 to 125 μin**. There may be minimal burrs on the exit side of the cut. The best broaching works on softer materials, such as brass, bronze, copper alloys, aluminum, graphite, hard rubbers, wood, composites, and plastic. However, it still has a good machinability rating on mild steels and free machining steels.

When broaching, the machinability rating is closely related to the hardness of the material. For broach steels the ideal hardness range is between **16 and 24 Rockwell C (HRC)**; a hardness greater than **HRC 35** will dull the broach quickly. Broaching is more difficult on harder materials, stainless steel and titanium alloys are possible.
Broaching configuration:

Both pull and push type broaches are made in the form of slender rods or bars of varying section having along its length one or more rows of cutting teeth with increasing height (and width occasionally). Push type broaches are subjected to compressive load and hence are made shorter in length to avoid buckling. The general configuration of pull type broaches, which are widely used for enlarging and finishing preformed holes, is schematically shown below.

Material of broaches:

Being a cutting tool, broaches are also made of materials having the usual cutting tool properties, i.e., high strength, hardness, toughness and good heat and wear resistance. 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. More ductile materials 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.
Broaching is getting more and more widely used, wherever feasible, for high productivity as well as product quality. Various types of broaches have been developed and are used for wide range of applications. Broaches can be broadly classified in several aspects, such as:

- Internal or external broaching
- Pull type or push type
- Ordinary cut or progressive type
- Solid, Sectional or modular type
- Profile sharpened or form relieved type

The wide range of internal broaching tools and their applications include:

- through holes of different form and dimensions
- non-circular holes and internal slots
- internal keyway and splines
- teeth of straight and helical fluted internal spur gears

The external surface broaching competes with milling, shaping and planing and, wherever feasible. External broaching tools may be both pull and push type. Major applications of external broaching are:

- un-obstructed outside surfacing; flat, peripheral and contour surfaces
- grooves, slots, keyways etc. on through outer surfaces of objects
- external splines of different forms
- teeth of external spur gears or gear sectors

![Broaching Types Diagram]

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: Belt grinding is a versatile process suitable for all kind of applications like finishing, deburring, and stock removal.
**Bench grinder:** Usually have 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. The workpiece is rotated and fed past the wheel(s) to form a cylinder. 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.

**Tool and cutter grinder:** These usually can perform the minor function of the drill bit grinder, 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:

- Grinding wheel travel: 7 1/2 in. vertical;
- Grinding wheel travel: 5 1/2 in. horizontal;
- Table travel: 6 in.;
- Slitting saws with attachment: 12 in. diameter;
- Distance between centers: 14 inches;
- Swing on centers (diameter): 4 1/2 in. diameter;
- Swing in work head (diameter): 4 1/2 in. diameter.

**Bench Grinding and Buffing Machines:** More suitable for general grinding, cleaning, and buffing, 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. One of the two wheels can be removed and another one is mounted for buffing and polishing purposes. The 1/4-HP electric motor revolves at a maximum of 3,450 RPM.

**Universal Grinding Machines:** The basic components of a universal grinder 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:** These are horizontal-type surface grinding machines. The work-pieces are fastened to the table and moved beneath the grinding abrasive wheel by hand or power feed. A magnetic chuck may also be used for fastening the workpiece to the table. This type of grinding machine has an internal pump and a piping network for automatic application and 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. The grinding wheel speed is adjustable.

Universal Grinding Machines

Reciprocating Surface Grinding Machine

Types of flat surface grinding machine configurations are shown below:

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. The grits are held together by a suitable bonding material to give shape of an abrasive tool.
Grinding applications:

- Dimensional accuracy
- Good surface finish
- Good form and location accuracy
- Applicable to both hardened and unhardened material
- Surface finishing
- Slitting and parting
- Descaling, deburring
- Stock removal (abrasive milling)
- Finishing of flat as well as cylindrical surface
- Grinding of tools and cutters and re-sharpening of the same.

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. The wheel speed can change by changing the spindle speed or by using a larger or smaller wheel. To also find the wheel speed in sfpm, multiply the spindle speed (rpm) by the wheel circumference (inches) and divide the product by 12.

\[
sfpm = \frac{\pi \times D \times \text{rpm}}{12}
\]

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

The maximum speed listed on a grinding wheel is not necessarily the speed at which it will cut best. Always see the manufacturers’ tables about the maximum speed based on the strength of the wheel. The recommended speed provides a margin of safety and the wheel usually cut better at a lower speed. Take a trial cut. If the wheel acts too soft, increase the speed. If it acts too hard, decrease the speed.

Work or surface speed 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. The following methods are recommended for determine feeds:
For roughing, the table should traverse about three quarter the wheel width per revolution or pass of the workpiece. For an average finish, the wheel should traverse one-third to one-half the width of the wheel per revolution or pass of the workpiece. In surface grinding with wheels less than 1 inch in width, the table traverse speed should be reduced about one-half.

The volume of chip produced by individual grit depends upon the maximum grit depth of cut, wheel workpiece contact length and grit width of cut. The figure below shows a grinding wheel with a single layer configuration having the tips of all the grits in the same level, engaged in a grinding mode.

**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.

For finishing, the depth of cut is always slight, generally from **0.0005 inch** to as little as **0.00005 inch**. An indication of the depth of cut is given by the volume of sparks thrown off. Also, an uneven amount of sparks indicates that the workpiece or wheel is not concentric.

**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.

In addition, coolant keeps the chips washed away from the grinding wheel and point of contact, thus permitting free cutting. Clear water may be used as a coolant, but various compounds containing alkali are usually added to improve its lubricating quality and prevent rusting of the machine and workpiece.

An inexpensive coolant often used for all metals, except aluminum, consists of a **solution of approximately 1/4 pound of sodium carbonate (salt soda) dissolved in 1 gallon of water**. Another good coolant is made by dissolving **soluble cutting oil in water**. For grinding aluminum and its alloys, a clear water coolant will produce fairly good results.

**Specification of grinding wheels:**

Grinding wheel consists of hard abrasive grains called grits, which perform the cutting or material removal. A grinding wheel is commonly identified by the type of the abrasive material used. The conventional wheels include alumi-
num oxide and 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:

(a) Geometrical specification;
(b) Compositional 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 and super-abrasive grinding wheels are shown below.

**Compositional specification**: Means compositional specification specified encompassing the following parameters:

1) Type of grit material;
2) Grit size;
3) Bond strength of the wheel, commonly known as wheel hardness;
4) Structure of the wheel denoting the porosity i.e. the amount of inter grit spacing;
5) Type of bond material;
6) The wheel manufacturer may add their own identification code prefixing or suffixing (or both) the standard code.
Types of abrasives:

**Aluminum oxide**: May have variation in properties arising out of differences in chemical composition and structure associated with the manufacturing process. Pure Al₂O₃ grit with defect structure like voids leads to unusually sharp free cutting action with low strength and is advantageous in fine tool grinding operation, and heat sensitive operations on hard, ferrous materials.

Regular or brown aluminum oxide (doped with TiO₂), possesses lower hardness and higher toughness than the white Al₂O₃ recommended for heavy duty grinding and semi finishing. Al₂O₃ alloyed with chromium oxide (<3%) is pink in colour.

- **Monocrystalline**: Al₂O₃ grits make a balance between hardness and toughness and are efficient in medium pressure heat sensitive operation on ferrous materials.

- **Microcrystalline**: Sintered Al₂O₃ grit is the latest development particularly known for its toughness and self-sharpening characteristics practically suitable for stock removal grinding.

- **Microcrystalline Al₂O₃ alloyed with zirconia**: makes extremely tough grit mostly suitably for high pressure, high material removal grinding on ferrous material and are not recommended for precision grinding.

**Silicon carbide**: Is harder than alumina but less tough and inferior to Al₂O₃ 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.

- **Natural diamond grits**: Are characterized by its random shape, very sharp cutting edge and free cutting action and is exclusively used in metallic, electroplated and brazed bond.

- **Monocrystalline diamond grits**: Are known for their strength and designed for particularly demanding application. These are also used in metallic, galvanic and brazed bond.

- **Polycrystalline diamond grits**: Are more friable than monocrystalline one and found to be most suitable for grinding of cemented carbide with low pressure. These grits are used in resin bond.

**Cubic boron nitride (cBN)**: Beyond diamond, it 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 monocrystalline type with medium strength and blocky mono-crystals with much higher strength, with following characteristics:

- Medium strength crystals are more friable and used in resin bond for those applications where grinding force is not so high.
- High strength crystals are used with vitrified, electroplated or brazed bond where large grinding force is expected.

**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, but recommended for very high speed grinding because of possible breakage of the bond under centrifugal force.
**Resin bond:** Conventional abrasive, 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. Resin bond is not recommended with alkaline grinding fluid due a possible chemical attack leading to weakening.

**Fiberglass reinforced resin bond:** Commonly used with cut off wheels which requires added strength under high speed operation.

**Shellac bond:** This bond was used for flexible cut off wheels. At present its use is limited to grinding wheels engaged in fine finish of rolls.

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

**Rubber bond:** Its principal use is in thin wheels for wet cut-off operation. Rubber bond was once popular for finish grinding on bearings and cutting tools.

**Metal bond:** Is extensively used with super abrasive wheels. 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. This bond is specially used for making small diameter wheels, form and thin super abrasive wheels. Presently it is the only bond for making wheels for abrasive milling and ultra-high speed grinding.

**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. In practice the effective macro-geometry of a grinding wheel is of vital importance and accuracy of the finished workpiece, is directly related to effective wheel geometry.

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. Also, in the photograph below you can 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 prop-
er 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.
- The number ‘100’ 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 by the manufacturer with a stencil or a small tag. The manufacturers have worked out a standard system of markings, as shown below. For example, use 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.
# Grinding Wheel Selection and Application

<table>
<thead>
<tr>
<th>SUITABLE FOR</th>
<th>WHEEL MATERIAL</th>
<th>GRAIN</th>
<th>GRADE</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>External Cylindrical Grinding</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Good all-around wheels; best adapted to soft steel</td>
<td>Aluminox</td>
<td>2946</td>
<td>L</td>
</tr>
<tr>
<td>Hardened steel</td>
<td>Alumund</td>
<td>3836</td>
<td>L</td>
</tr>
<tr>
<td>Soft steel of small diam.</td>
<td>Aloxite</td>
<td>401</td>
<td>N</td>
</tr>
<tr>
<td>Reamers, drills and general tool work</td>
<td>Aluminox or Alumund</td>
<td>46</td>
<td>K</td>
</tr>
<tr>
<td>Hard steel, dry grinding</td>
<td>Aluminox or Alumund</td>
<td>80</td>
<td>M</td>
</tr>
<tr>
<td>Cast iron and bronze</td>
<td>Crystolon</td>
<td>100</td>
<td>I</td>
</tr>
<tr>
<td></td>
<td>Crystolon</td>
<td>45</td>
<td>L</td>
</tr>
<tr>
<td><strong>Facing Shoulders</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Ordinary work</td>
<td>Aluminox or Alumund</td>
<td>60</td>
<td>H or I</td>
</tr>
<tr>
<td>Fine finish</td>
<td>Aluminox or Alumund</td>
<td>80</td>
<td>H</td>
</tr>
<tr>
<td><strong>Surface Grinding</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Hardened steel</td>
<td>Alumund or Aluminox</td>
<td>46</td>
<td>H</td>
</tr>
<tr>
<td>Hardened, high-speed steel or very thin pieces of hardened carbon steel</td>
<td>Alumund or Aluminox</td>
<td>60</td>
<td>H</td>
</tr>
<tr>
<td>Cast iron</td>
<td>Aloxite</td>
<td>367</td>
<td>U</td>
</tr>
<tr>
<td></td>
<td>Carbonundum or</td>
<td>39</td>
<td>G</td>
</tr>
<tr>
<td></td>
<td>Crystolon</td>
<td>36</td>
<td>J</td>
</tr>
<tr>
<td><strong>Disk Grinding</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Thick pieces, wet grinding</td>
<td>Aluminox or Alumund</td>
<td>30</td>
<td>K</td>
</tr>
<tr>
<td>Thin pieces, wet grinding</td>
<td>Aluminox or Alumund</td>
<td>30</td>
<td>J</td>
</tr>
<tr>
<td>High-speed steel, dry grinding</td>
<td>Aluminox or Alumund</td>
<td>60 or 80</td>
<td>H or I</td>
</tr>
<tr>
<td>Washers and similar pieces</td>
<td>Aluminox or Alumund</td>
<td>60</td>
<td>I</td>
</tr>
<tr>
<td><strong>Internal Cylindrical Grinding</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Good all around wheel</td>
<td>Aluminox or Alumund</td>
<td>46</td>
<td>2½ i</td>
</tr>
<tr>
<td>Roughing hardened steel</td>
<td>Aluminox or Alumund</td>
<td>46</td>
<td>J or K</td>
</tr>
<tr>
<td>Finishing hardened steel</td>
<td>Aluminox or Alumund</td>
<td>120</td>
<td>J or K</td>
</tr>
<tr>
<td>Ordinary finishing without roughing</td>
<td>Aluminox or Alumund</td>
<td>80</td>
<td>J or K</td>
</tr>
<tr>
<td>Roughing brass</td>
<td>Crystolon</td>
<td>35</td>
<td>H</td>
</tr>
<tr>
<td>Finishing brass</td>
<td>Crystolon</td>
<td>80</td>
<td>H</td>
</tr>
<tr>
<td>Automobile cylinders</td>
<td>Crystolon</td>
<td>46</td>
<td>K</td>
</tr>
<tr>
<td>Automobile cylinders</td>
<td>Carbonundum or</td>
<td>36</td>
<td>M or P</td>
</tr>
<tr>
<td>Automobile cylinders, roughing or fair finish</td>
<td>Carbolite</td>
<td>36</td>
<td>H or I</td>
</tr>
<tr>
<td>Automobile cylinders, fine finish</td>
<td>Carbolite</td>
<td>60</td>
<td>H</td>
</tr>
<tr>
<td><strong>Sharpening Carbon-Steel Cutters, Dry Grinding</strong></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Milling cutters</td>
<td>Aluminox or Alumund</td>
<td>46 or 80</td>
<td>I</td>
</tr>
<tr>
<td>Formed and gear cutters</td>
<td>Aluminox or Alumund</td>
<td>46</td>
<td>I</td>
</tr>
</tbody>
</table>

References:

©2013 Jurandir Primo
SENAI/Brazil: Processos Mecânicos de Usinagem, 2006
Sandvik: Metal Cutting Technical Guide, Catalogues
Machine Shop Work, by Frederick W. Turner, Oscar E. Perrigo, Howard P. Fairfield, 1918
Indian Institute of Technology, Kharagpur, Study Materials

Links:

http://www.ntma.org/
http://en.wikipedia.org/wiki/Machining
http://www.inderscience.com/