UNIT – 1 Introduction to Machining and Manufacturing

Role of machining in real-world industries (automotive, agriculture, aerospace, medical), Subtractive vs. additive vs. hybrid manufacturing, Types of machine tools: classification and selection, Overview of materials used in machining, Basic machining parameters: cutting speed, feed, depth of cut, MRR

Turning – The Lathe Machine-Construction and working of engine lathe, Work holding and tool holding devices, Operations: facing, straight turning, step turning, taper turning, knurling, thread cutting, parting - off, Cutting tool geometry and selection.

Real-world problems: shaft re-machining, thread repair.

Role of machining in real-world industries

The role of machining is to create precise, finished components by subtractive removing material (cutting, grinding, drilling) from raw stock, enabling the production of complex parts with tight tolerances for industries like aerospace, auto, and medical. It offers high accuracy, versatility with materials (metals, plastics, composites), and flexibility for custom or mass production, ensuring high-quality surfaces and functionality in everything from tiny electronics to large engine parts, acting as a crucial bridge between raw material and functional product.

Material Transformation: It transforms raw stock materials (metals, plastics, ceramics, composites, and more) into usable parts by systematically removing excess material, typically in the form of small chips, using cutting tools.

Achieving Precision and Tight Tolerances: Machining is renowned for its ability to produce parts with extremely high precision and tight dimensional tolerances (often down to microns with advanced CNC systems), which is essential for components that require exact fit and function.

Creating Complex Geometries: Using modern Computer Numerical Control (CNC) machines, particularly 5-axis systems, machinists can create intricate shapes, complex curves, internal cavities, and fine details that would be difficult to manufacture otherwise.

Finishing Other Parts: Components produced by other manufacturing methods like casting or forging often require secondary machining operations (such as grinding or boring) to achieve their final, required dimensions and surface finishes.

Versatility in Production Volumes: Machining is flexible enough to produce everything from one-off prototypes and custom parts to large-scale, mass production runs, making it suitable for a wide range of manufacturing needs.

Common Machining Processes;

Turning: The workpiece rotates while a cutting tool removes material, ideal for creating cylindrical parts like shafts and rods.

Milling: A rotating cutting tool moves across a stationary or moving workpiece to create flat surfaces, slots, and complex 3D contours.

Drilling: Uses a rotating bit to create or refine round holes.

Grinding: Employs an abrasive wheel to remove small amounts of material, primarily for achieving very fine surface finishes or working with hardened materials.

Shaping: This process uses a shaper machine to remove material with a single-point cutting tool that moves back and forth.

Slotting: A single-point tool is held in a ram that moves vertically up and down. The workpiece is mounted on a table that moves horizontally, perpendicular to the tool's motion, to create the desired slot.

Boring: A precise process to enlarge and refine pre-existing holes (cast or drilled) using a single-point cutting tool on a boring machine.

Reaming: The reaming process is a precision machining operation that uses a multi-edged rotary tool to enlarge and finish an existing, pre-drilled hole, significantly improving its dimensional accuracy and surface finish.

Applications Across Industries:

- **Aerospace:** Production of critical, high-strength, lightweight parts like turbine blades, engine mounts, and structural components that must withstand extreme conditions.
- **Automotive:** Mass production of essential engine components, transmission systems, gears, pistons, and brake parts.
- **Agriculture:** creating durable, high-precision parts for farm equipment like tractors, harvesters, and irrigation systems, boosting efficiency, reducing downtime, and enabling smart farming innovations for better yields and sustainability
- **Medical Devices:** Creation of high-precision items such as surgical instruments, orthopedic implants, and prosthetics, where biocompatibility and exact dimensions are vital.
- **Electronics:** Manufacturing of heat sinks, housings, and intricate components for computer hardware and mobile devices.
- **Energy Sector:** Production of valves, large shafts for wind turbines, and components for oil and gas systems that function in high-pressure environments.

Subtractive Vs. Additive Vs. Hybrid Manufacturing:

Subtractive manufacturing removes material from a block (like CNC), additive builds layer-by-layer (like 3D printing), and hybrid combines both in one machine, marrying additive's design freedom with subtractive precision for complex, high-tolerance parts.

Subtractive excels at smooth finishes and mass production but creates waste; additive is great for complex geometries and prototypes with less waste but needs post-processing. Hybrid manufacturing overcomes limitations of each, enabling intricate features and better accuracy in one workflow.

Subtractive Manufacturing

- **Process:** Material is cut, drilled, milled, or ground away from a solid block (stock).
- **Tools:** CNC machines, lathes, mills, laser/waterjet cutters.
- **Pros:** Excellent surface finish, high precision, faster for mass production.
- **Cons:** High material waste (chips), limited design complexity, significant setup.
- **Best for:** Gears, molds, strong metal parts, high-volume production.

Additive Manufacturing (AM)

- **Process:** Material (powder, filament, resin) is added layer by layer, fused by heat or light.
- **Tools:** 3D printers (FDM, SLA, SLS, DMLS).
- **Pros:** Intricate designs, minimal waste, rapid prototyping, material efficiency.
- **Cons:** Slower for bulk, rougher surfaces, requires post-processing, higher cost per part for some methods.
- **Best for:** Prototypes, customized medical devices, complex aerospace parts.

Hybrid Manufacturing

- **Process:** Integrates AM and subtractive methods in a single machine or workflow (e.g., 3D prints a complex shape, then mill critical surfaces).
- **Tools:** Hybrid machines with integrated 3D printing (like Directed Energy Deposition - DED) and CNC milling.
- **Pros:** Combines design freedom with precision, reduces tooling, repairs complex parts, improves final quality (smoothness, tolerance).

- **Cons:** Complex software/programming, higher initial machine cost.
- **Best for:** Repairing high-value components, creating functionally graded materials, parts requiring both complex internal structures and precise external surfaces.

Types of machine tools

Machine tools consist of two or more integrated units, each serving a specific purpose within the same machine. When these components come together to form the structure of a machine tool, they must exhibit certain essential characteristics to ensure optimal performance and functionality:

Rigidity: The overall structure must possess adequate rigidity to withstand the forces generated during machining operations. This rigidity ensures stability and minimizes vibrations, promoting precise and accurate machining.

Functionality: Each unit of the machine tool must serve a defined function that contributes to the machining process. The seamless coordination of these functions is crucial for achieving the desired outcome.

Supplementary Items: Machine tools may incorporate supplementary items, such as tool changers, work holding devices, and coolant systems, which enhance productivity and facilitate efficient operations.

Chip Removal Capability: The machine tool structure must be designed to facilitate the efficient removal of chips or swarf generated during cutting or machining processes. Proper chip evacuation prevents tool damage and maintains a clean working environment.

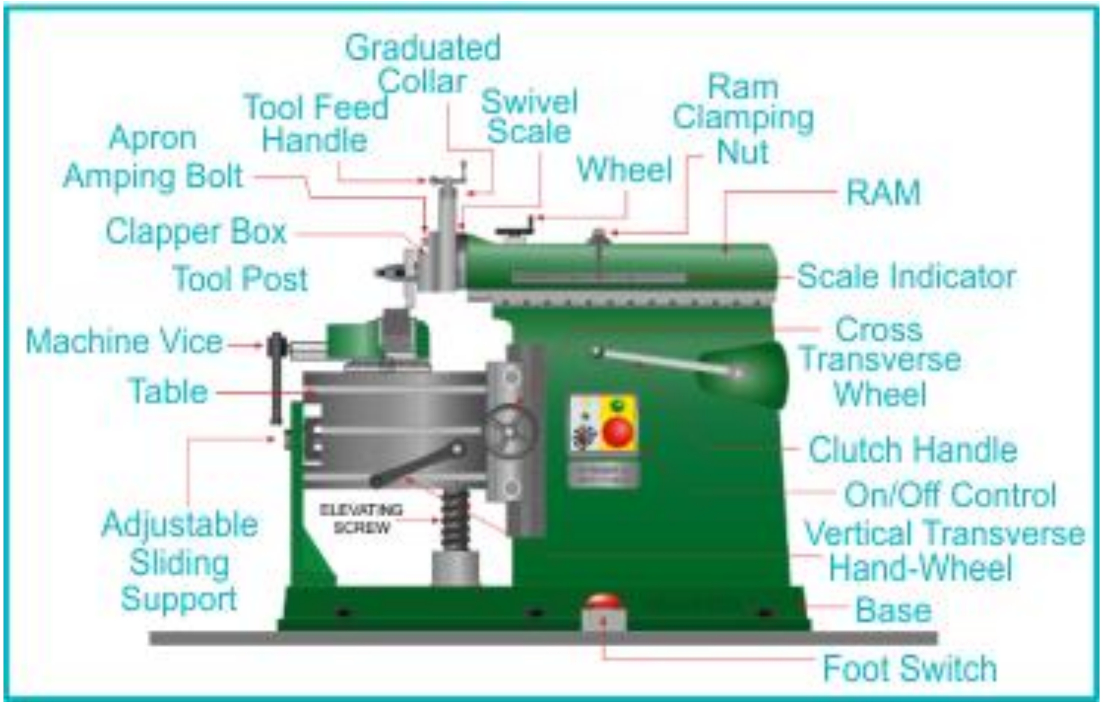
Safety Measures: Ensuring sufficient safety features is paramount to protect operators and maintain a secure working environment. The machine tool design should integrate safety mechanisms such as interlocks, emergency stop buttons, and protective enclosures.

Resistance to Wear: To preserve the accuracy of the machined workpiece, the machine tool structure must exhibit adequate resistance to wear and deterioration over time. This durability ensures consistent performance and extends the machine's service life.

The various types of Machine tools include:

- Shaper
- Planner
- Broach
- Drill
- Hobbing machine
- Lathe
- Milling machine
- Grinding machines

Shaper



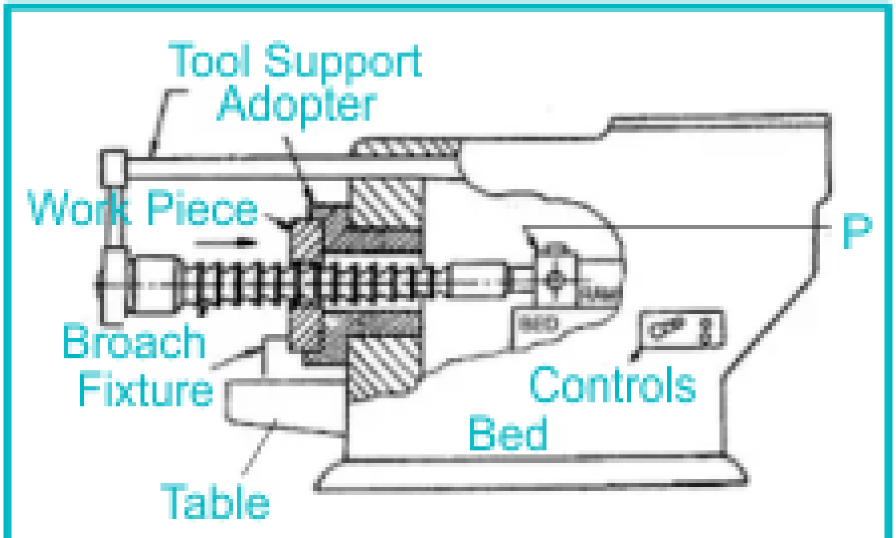
The shaper machine tool is a versatile and classic cutting device used in metalworking and machining industries. Its primary function is to produce flat surfaces and intricate shapes on workpieces by removing material through a reciprocating cutting motion. The shaper consists of a robust base, a vertical column, a horizontal ram, and a tool head, which holds the cutting tool.

Planner

The planner machine tool, also known as a planer, is a significant and powerful machine used in metalworking and manufacturing industries. Its main function is to produce flat and accurate surfaces on large workpieces that are difficult to handle on other machining tools. The planer typically consists of a heavy, sturdy bed, a cross rail, a vertical tool head, and a reciprocating horizontal table.

Broach

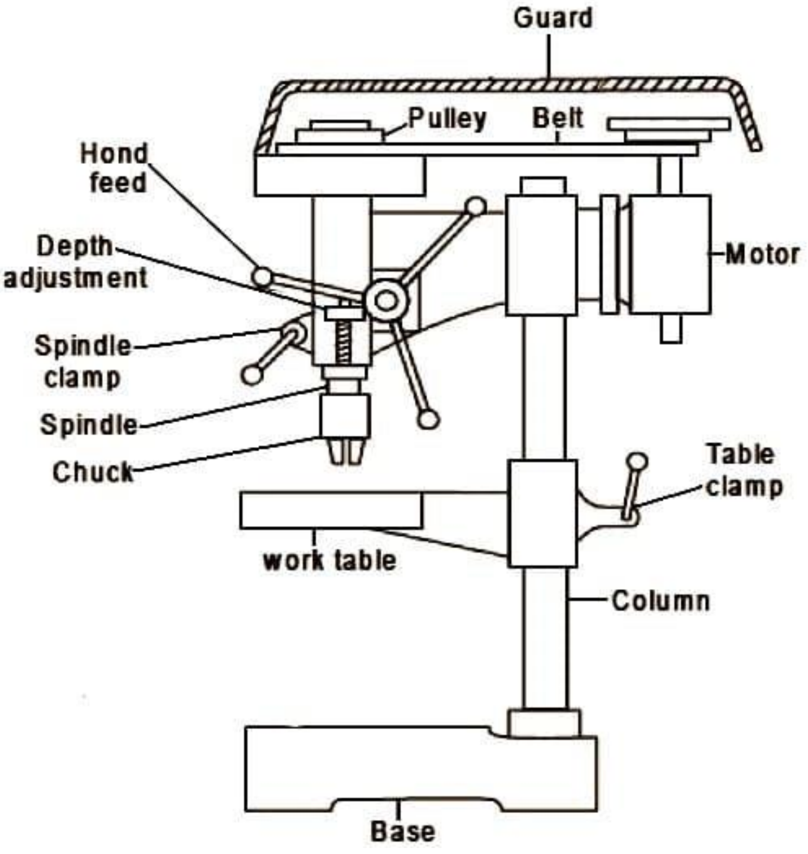
The broach machine tool, commonly known as a broaching machine, is a specialized and highly efficient device used in metalworking to create precise and complex internal or external profiles on workpieces. Broaching is a machining process that involves removing material using a toothed cutting tool called a broach. The broach has a series of progressively larger cutting teeth, and it is pulled or pushed through the workpiece in a linear motion to achieve the desired shape.



Horizontal Broaching Machine

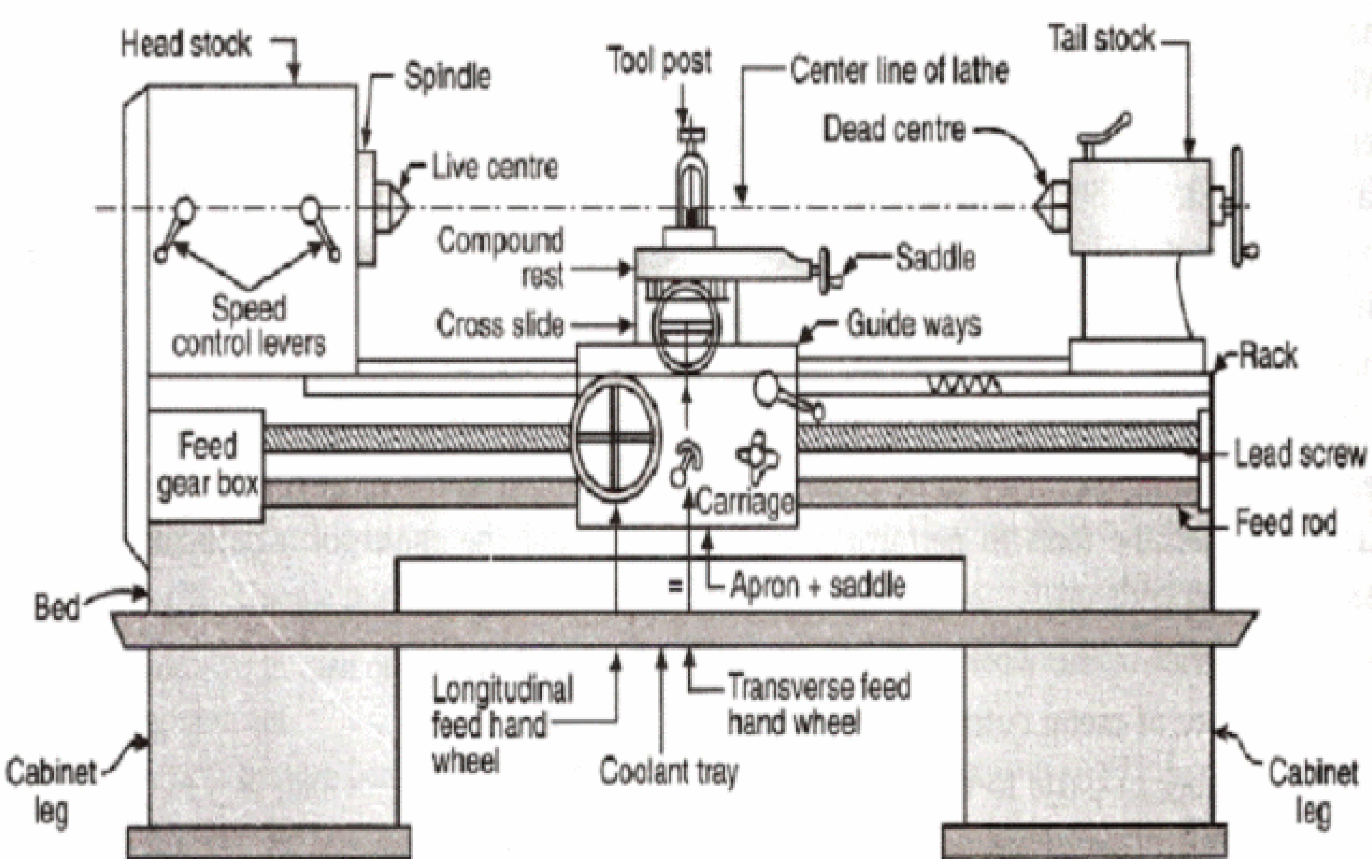
Drill

The drill machine tool, commonly referred to as a drilling machine or drill press, is a fundamental device used in metalworking, woodworking, and construction industries for creating holes in various materials. This versatile machine is designed to deliver accurate and precise drilling operations. The drill machine typically consists of a sturdy base that provides stability, a vertical column that supports the drill head, an adjustable worktable, and a rotating spindle that holds the drill bit.



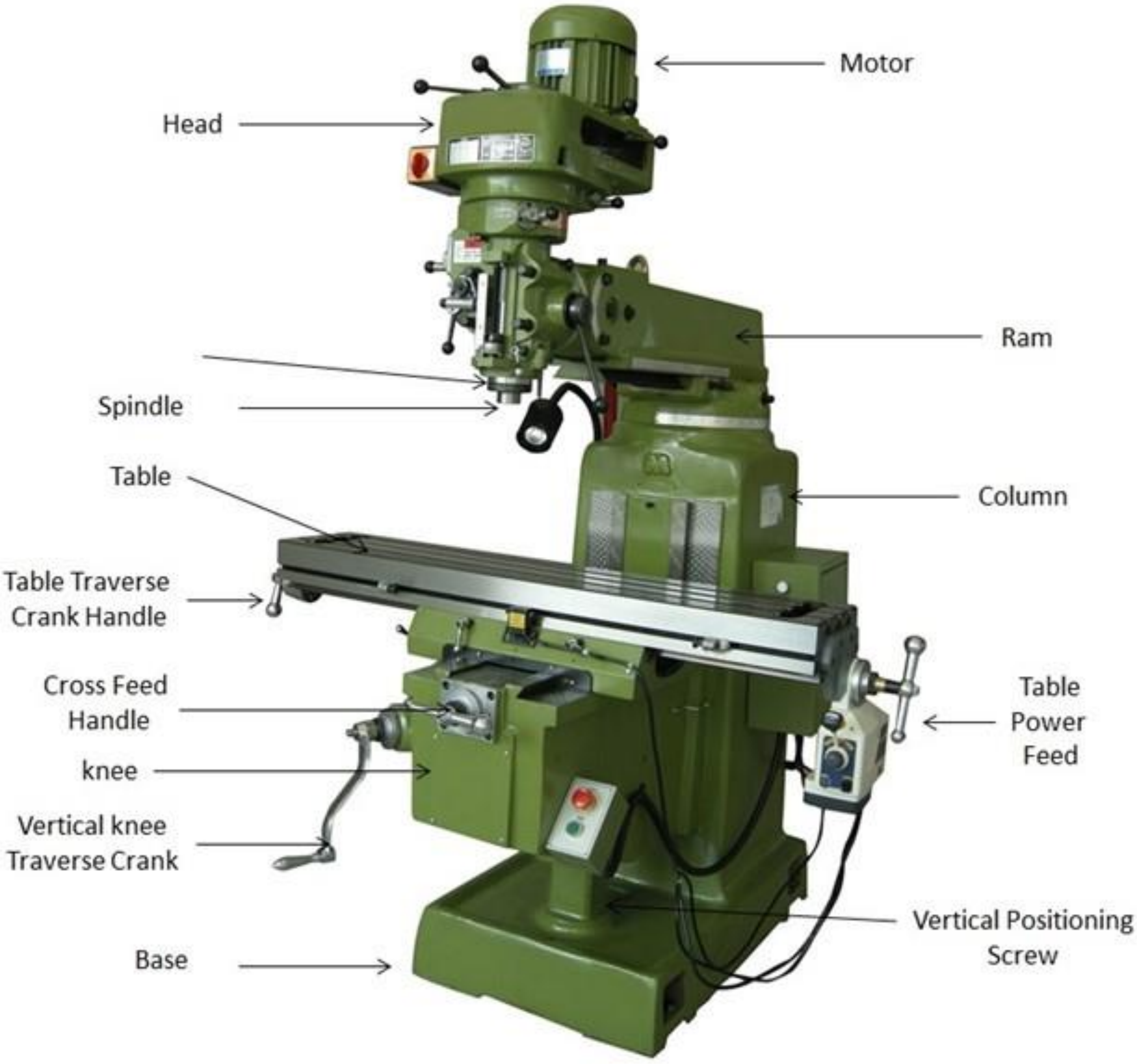
Lathe

The lathe machine tool is a fundamental and versatile device used in metalworking and machining industries. It rotates a workpiece while a cutting tool moves along it to shape, cut, or drill materials with precision. Lathes are widely utilised for various operations, including turning, facing, threading, and tapering, making them essential for creating symmetrical components and precise cylindrical shapes.



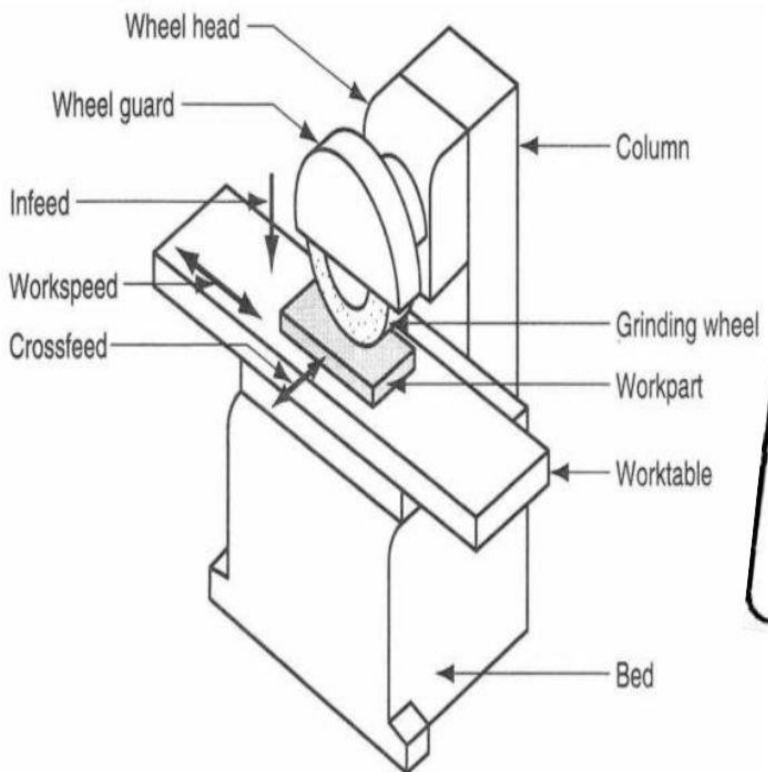
Milling Machine

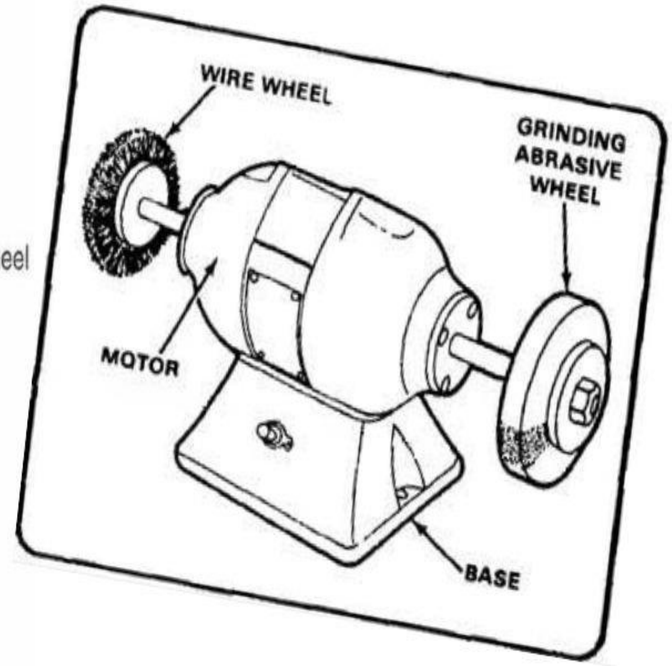
The milling machine tool is a versatile and essential device used in metalworking and machining industries. It employs rotary cutters to remove material from a workpiece, creating various shapes, slots and features with high precision. Milling machines can perform a wide range of operations, such as face milling, end milling, drilling, and slotting.



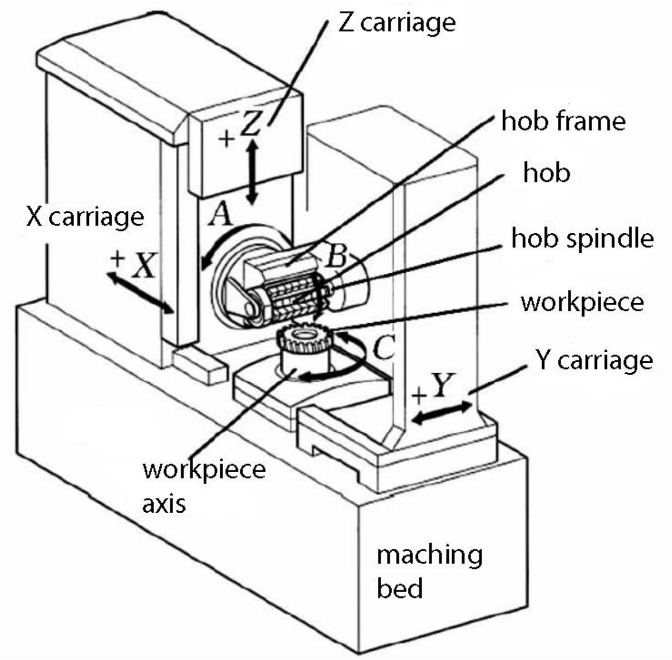
Grinding Machines

Grinding machines are essential machine tools used in the metalworking and manufacturing industries for precision grinding and finishing of surfaces. These machines utilise abrasive wheels to remove material from the workpiece, resulting in a smooth and accurately shaped surface. There are various types of grinding machines, each designed for specific applications. Cylindrical grinders are used for cylindrical or round workpieces, while surface grinders are used for flat surfaces.





Hobbing Machine



The hobbing machine tool, commonly known as a hobbing machine, is a specialised device used in gear manufacturing to create gears with excellent precision and efficiency. It is widely employed in various industries, including automotive, aerospace, and heavy machinery, where gears are essential components for power transmission and motion control.

Advantages of Machine Tools

The various advantages of machine tools include:

Increased productivity: Machine tools enable faster and more efficient production processes, leading to higher output and reduced manufacturing time.

Consistency and accuracy: Machine tools offer precise and repeatable results, ensuring uniformity and consistency in the produced components or products.

Versatility: Many machine tools can perform multiple operations, making them versatile and adaptable to different manufacturing needs.

Reduced labour costs: Automating tasks with machine tools reduces the need for manual labour, leading to cost savings in the long run.

Improved safety: Using machine tools in hazardous or repetitive tasks reduces the risk of accidents and injuries to workers.

Higher quality products: Machine tools produce components with superior surface finishes and tighter tolerances, resulting in better overall product quality.

Flexibility: CNC (Computer Numerical Control) technology allows for quick programming changes, enabling rapid adjustments to meet changing production requirements.

Waste reduction: Efficient material removal and utilisation in machine tools minimise material waste, leading to cost savings and environmental benefits.

Innovation and complexity: Machine tools enable the creation of intricate and complex designs that may not be achievable with manual methods.

Disadvantages of Machine Tools

Some of the noteworthy disadvantages of Machine tools are:

Initial investment cost: Acquiring and setting up machine tools can be expensive, especially for high-end CNC machines, which might pose a significant financial burden for small businesses.

Skilled labour requirement: Operating and maintaining machine tools often require trained and skilled personnel, adding to labour costs and potential difficulties in finding qualified operators.

Downtime and maintenance: Machine tools need regular maintenance, and any breakdown or downtime can disrupt production schedules and lead to lost productivity.

Limited flexibility: Some machine tools are specialised for specific tasks, making them less versatile for handling a wide range of production requirements.

Complexity of programming: CNC machines require complex programming, and mistakes in coding can lead to errors in the final product.

Potential for material waste: Inefficient setup or programming can result in material wastage, which adds to production costs and environmental impact.

Depreciation and obsolescence: Machine tools can become outdated over time, reducing their resale value and necessitating periodic updates or replacements.

Overview of Materials Used in Machining Process:

Machining uses various materials, primarily metals (Aluminum, Steel, Stainless Steel, Copper, Titanium, Brass) for strength, and non-metals (Plastics, Nylon) for electrical/medical uses, chosen based on properties like hardness, machinability, conductivity, and corrosion resistance;

Tool materials must be harder than the work piece (e.g., Carbide, HSS) with coatings for performance, while processes range from traditional cutting to modern non-traditional methods like EDM for diverse applications.

i) Work piece Materials (Materials Being Machined)

Metals:

Aluminum: Lightweight, good machinability, corrosion resistance (aerospace, automotive).

Aluminum alloys possess excellent strength-to-weight ratio, high thermal and electrical conductivity, and natural corrosion protection. They are easy to process and have low batch costs, thus they are often the most economical choice for manufacturing custom metal parts and prototypes. The strength and hardness of aluminum alloys are generally lower than those of steel, but they can undergo anodization to form a hard protective layer on their surface.

Aluminum 6061 is the most common and versatile aluminum alloy, featuring a good strength-to-weight ratio and excellent machining properties. Aluminum 7075 is the most commonly used alloy in aerospace applications, where weight reduction is crucial. It has excellent fatigue properties and can be heat-treated to achieve high strength and hardness comparable to steel. Aluminum 5083 has higher strength than most other aluminum alloys and excellent seawater resistance. Therefore, it is often used in construction and marine applications. It is also the best choice for welding.

Steel/Stainless Steel: Stainless steel alloys have high strength, high ductility, excellent wear resistance, and corrosion resistance, and are easy to weld, process, and polish. They are non-magnetic or magnetic.

Stainless steel 304 is the most common stainless steel alloy, with excellent mechanical properties and good machinability. It can resist most environmental conditions and corrosive media. Stainless steel 316 is another common stainless steel alloy, with mechanical properties similar to 304. Although it has higher corrosion resistance and chemical resistance, especially for salt solutions (such as seawater), it is often the first choice for applications in harsh environments.

Compared with 304, 303 stainless steel has excellent toughness but lower corrosion resistance. Due to its excellent machinability, it is often used in high-volume applications, such as manufacturing nuts and bolts for aerospace applications.

Copper/Brass: Excellent conductivity (electrical, thermal), easy to form (electrical components, fittings).

Titanium: High strength-to-weight, corrosion-resistant, biocompatible (medical, aerospace).

Plastics:

Engineering Plastics: Good for electrical insulation, medical, lower friction.

Other: Ceramics, Composites (also machined, requiring specialized tools/techniques).

ii) Cutting Tool Materials

The characteristic of the ideal cutting tool material are- (a) Hot hardness (b) Wear resistance (c) Toughness (d) Cost and easiness in fabrication.

Hot hardness:

The material must remain hander than the work material at elevated operating temperatures.

Wear resistances:

The material must withstand excessive were even through the relative hardness of the tool-work materials changes.

Toughness:

The term toughness actually implies a combination of strength and ductility. The material must have sufficient toughness to withstand shocks and vibrations and to prevent breakage.

Key Properties Needed: Hardness (30-50% harder than work piece), toughness, wear resistance, high-temperature stability, good thermal conductivity.

Common Materials:

High-Speed Steel (HSS)

The high speed steel tools have faster cutting speeds. This tool is made of cobalt, tungsten, and chromium-alloyed steel. Due to the presence of strong, harder, and temperature-resistant materials in cutting tools, cutting tools made from these materials can operate under (550^{o}C) to (650^{o}C)

Carbides (Tungsten Carbide)

This tool is an alloy made of tungsten, titanium, and other metals. It gives the tool high strength and hardness. Additionally, due to the presence of carbon, it has higher wear resistance and can withstand higher temperatures. The operating temperature of this tool ranges between (900^{o}C) and (1000^{o}C)

Ceramics

The tools made of ceramic are among the hardest tools used in cutting and machining as they can withstand high temperatures. This equipment can function between (1160^{o}C) and (1210^{o}C). Hence, these are used in finishing operations in the lathe or milling machines.

Diamond (for very hard materials)

The cutting tools made of diamonds are currently the strongest material. Diamonds are produced naturally on earth. This gives the tool many benefits in terms of temperature resistance, wear resistance, and shock resistance. These implements are utilized to cut tenacious, robust, and hard materials.

Abrasives: There are a variety of abrasive cutting tools, each of which cuts by rapidly rubbing a hard material against the target object. An abrasive material is hard and coarse, and it wears away whatever it is rubbed against. This equipment is embedded with abrasive material. A grinding wheel is one of the simplest types.

iii) Factors Influencing Material Choice

- **Mechanical Properties:** Hardness, strength, ductility (affects chip formation).
- **Thermal Properties**: Thermal conductivity (affects heat transfer to tool).
- **Chemical Properties:** Corrosion resistance, chemical reactivity.

Cutting Speed, Feed and Depth of Cut:

Cutting speed of a tool can be defined as the rate at which its cutting edge passes over the surface of the work-piece in unit time. It is normally expressed in terms of surface speed in meters per minute.

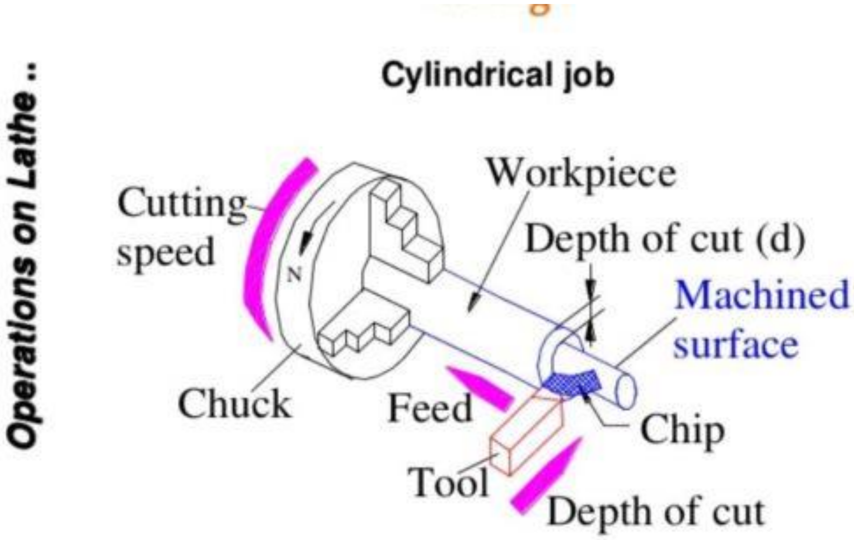
In machining it is important as it considerably affects the tool life and efficiency of machining. Selection of proper cutting speed has to be made very judiciously. If it is too high, the tool gets over heated and its cutting edge may fail, needing regrinding. If it is too low, too much time is consumed in machining and full cutting capacities of the tool and machine are not utilized, resulting in lowering of productivity and increasing the production cost.

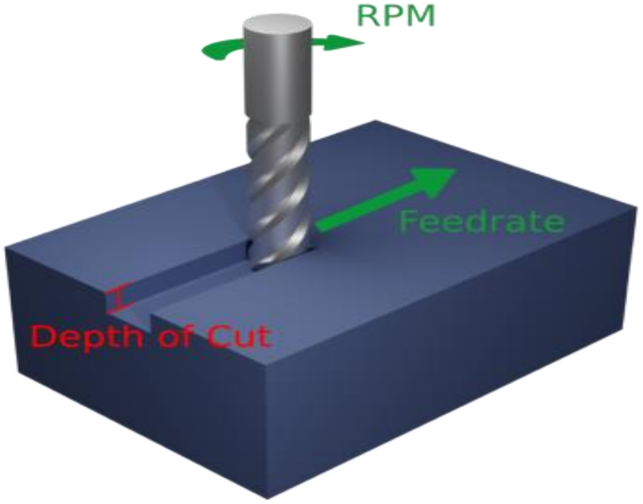
Feed of the cutting tool can be defined as the distance it travels along or in to the work-piece for each pass of its point through a perpendicular position in unit time. In turning operation of lathe, it is equal to the advancement of the tool corresponding to each revolution of work. In planning it is the work, which is fed and not the tool. In milling work, the feed is considered per tooth of the cutter.

The cutting speed and feed of a cutting tool is largely influenced by the following factors:

1. Material being machined.
2. Material of the cutting tool.
3. Geometry of the cutting tool.
4. Required degree of surface finish.
5. Rigidity of the machine tool being used
6. Type of coolant being used

Depth of cut: It is indicative of the penetration of the cutting edge of the tool in to the work piece material in each pass, measured perpendicular to the machined surface i.e. it determines the thickness of metal layer removed by the cutting tool in one pass.





Example: In turning operation on a lathe it is given by

$$D-d/2$$

Where D = Original diameter of the work-piece in mm

d = Diameter obtained after turning in mm in one pass.

Selection of cutting speed, feed & depth of cut:

➢ Hard and strong materials require a lower cutting speed, soft & ductile materials require higher cutting speeds.

➢ For light finishing cut – fine feed & higher speed

➢roughing cut – low feed & lower cutting speed.

➢ Large depth of cut – roughing operation

➢ Small depth of cut – finishing operation

➢ Cemented carbide, ceramics, satellite & HSS – high cutting speed tool

➢ Alloy or carbon steel tools – lower cutting speed.

METAL REMOVAL RATE

Metal Removal Rate (MRR) is the volume of material removed from a work piece per unit of time (e.g., mm^3/min, in^3/min) during machining, indicating process efficiency and productivity. A higher MRR signifies greater productivity, as more material is removed in less time, though it must be balanced with desired surface finish and accuracy.

Let

t - Depth of cut in mm

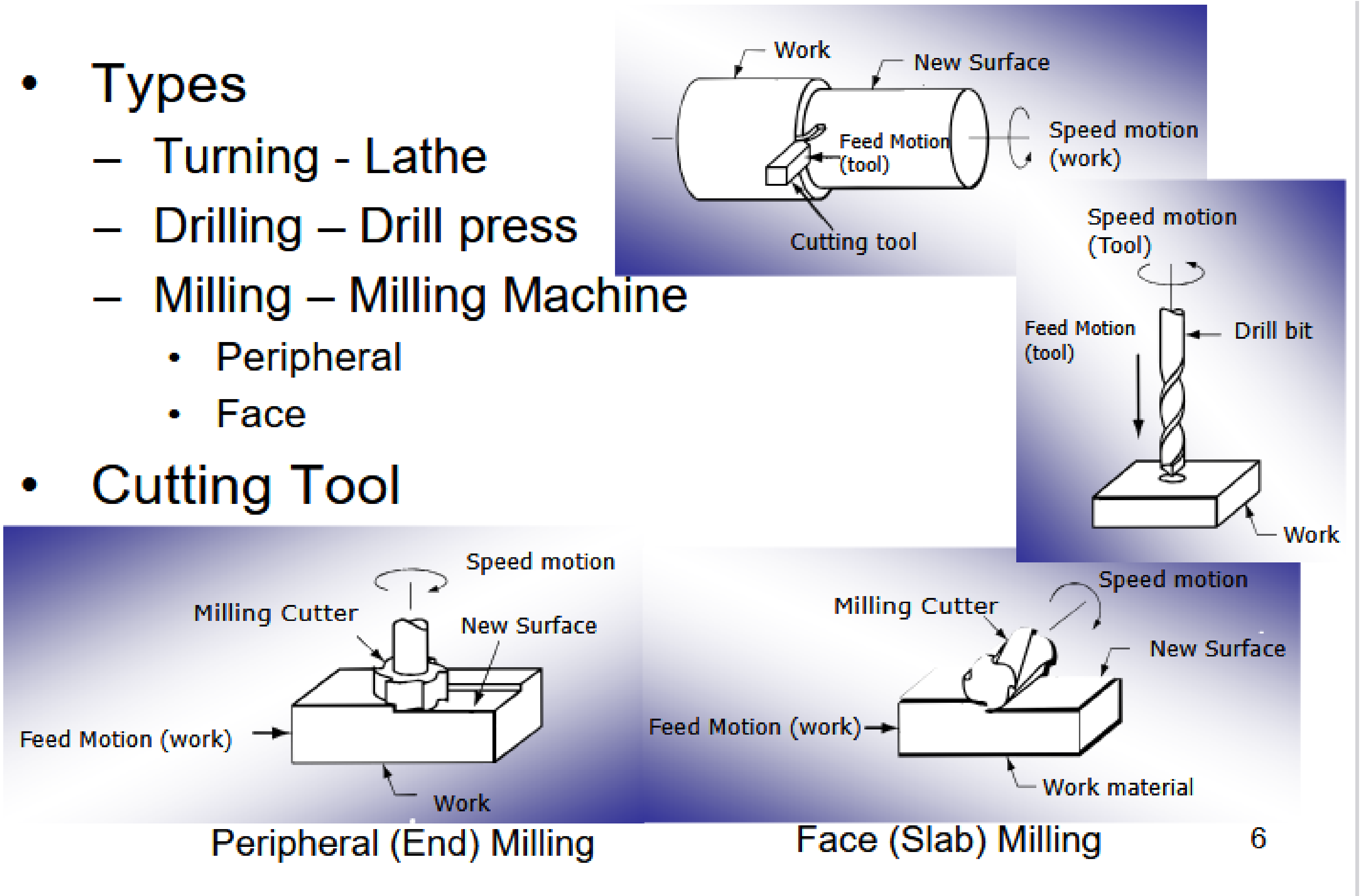
f - Feed/rev. in mm

V - Cutting speed in mm/s

Metal removal rate, MRR = $t \times f \times V$ in mm^3/s

If the metal removal rate is optimum, the machining cost is reduced. To achieve this, the following elements should be considered.

- The cutting tool material should be properly selected
- Cutting tool should be properly ground
- Tool should be supported rigidly and therefore, there should not be any vibration.



- Types
  - Turning - Lathe
  - Drilling – Drill press
  - Milling – Milling Machine
    - Peripheral
    - Face
- Cutting Tool

Peripheral (End) Milling

Face (Slab) Milling

6

For **turning** operation, MRR =f × t × V in mm^3/s

Where t - Depth of cut in mm

f - Feed/rev. in mm

V - Cutting speed in mm/s

For **face and spot milling** operation, MRR = B × t × T in mm^3/s

Where B = Width of cut in mm

T - Table travel in mm/s

For **planning and shaping**, MRR = t × f × L × S in mm^3/s

Where L - Length of work piece

S - Strokes per minute.

LATHE MACHINE

The Lathe Machine:

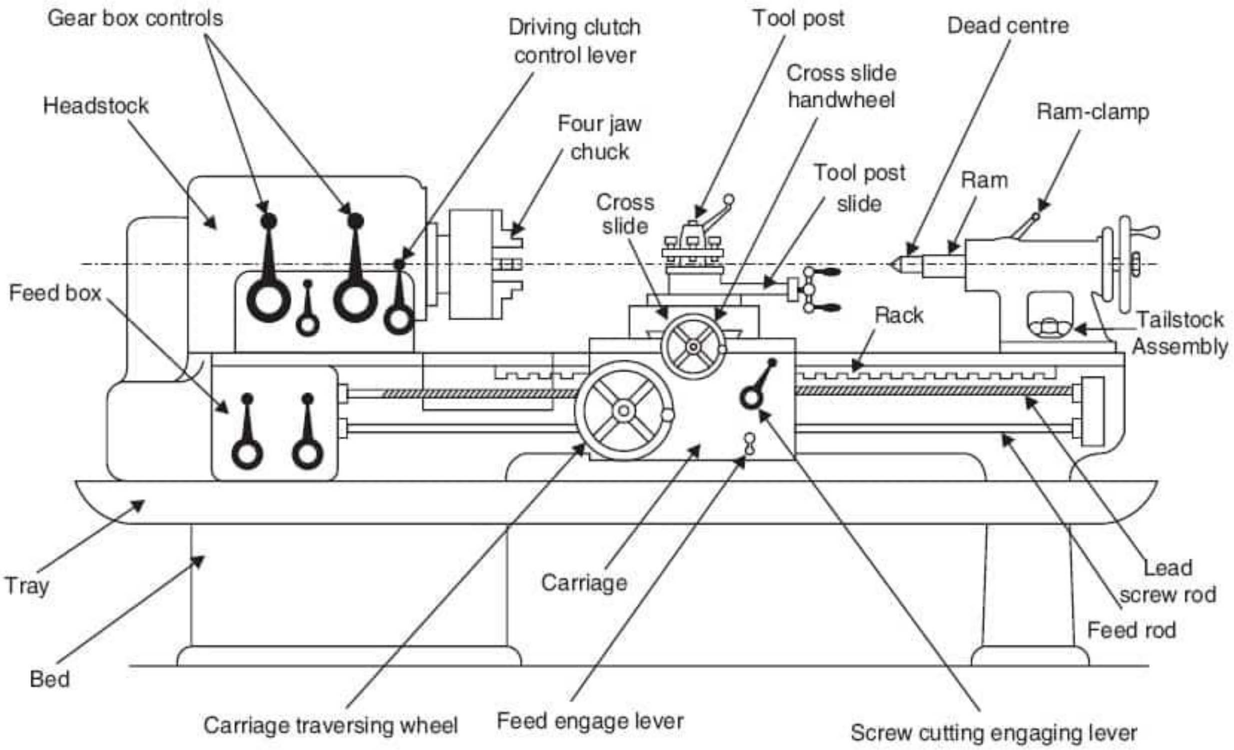
Lathe is a machine, which removes the metal from a piece of work to the required shape and size.

Lathe is one of the most important machine tools in the metal working industry. A lathe operates on the principle of a rotating work piece and a fixed cutting tool.

The cutting tool is feed into the workpiece, which rotates about its own axis causing the workpiece to be formed to the desired shape.

Lathe machine is also known as "the mother/father of the entire tool family"

CONSTRUCTION OF THE LATHE MACHINE



Engine lathe

Main parts of a lathe

Every individual part performs an important task in a lathe. Some important parts of a lathe are listed below:

1. Bed
2. Headstock
3. Spindle
4. Tailstock
5. Carriage
    a. Saddle
    b. Apron
    c. Cross-slide
    d. Compound rest
    e. Compound slide
    f. Tool post
6. Feed mechanism
7. Lead screw
8. Feed rod

1 Bed

Bed is mounted on the legs of the lathe which are bolted to the floor. It forms the base of the machine. It is made of cast iron and its top surface is machined accurately and precisely. Headstock of the lathe is located at the extreme left of the bed and the tailstock at the right extreme. Carriage is positioned in between the headstock and tailstock and slides on the bed guide ways.

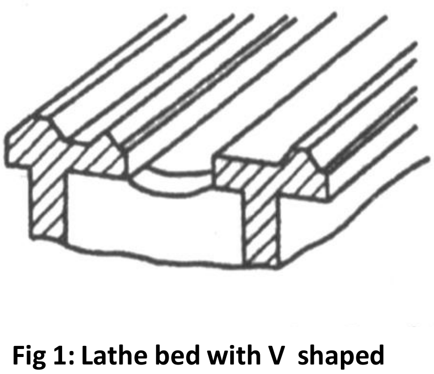


Fig 1: Lathe bed with V shaped guideway

The top of the bed has flat or ‘V’ shaped guide ways. The tailstock and the carriage slides on these guide ways. Inverted ‘V’ shaped guide ways are useful in better guide and accurate alignment of saddle and tailstock. The metal burrs resulting from turning operation automatically fall through. Flat- bed guide ways can be found in older machine tools. It is useful in heavy machines handling large workpieces. But then the accuracy is not high.

2 Headstock

Headstock is mounted permanently on the inner guide ways at the left hand side of the leg bed. The headstock houses a hollow spindle and the mechanism for driving the spindle at multiple speeds. The headstock will have any of the following arrangements for driving and altering the spindle speeds:

(i) Stepped cone pulley drive

(ii) Back gear drive
(iii) All gear drive

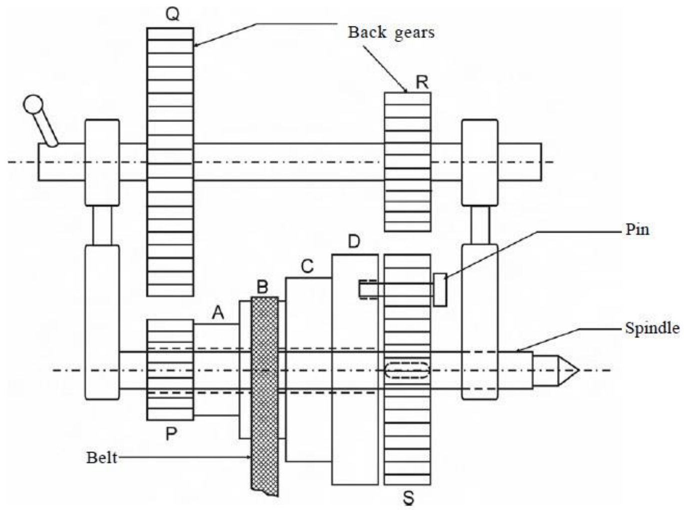


Fig 2: Stepped cone pulley with back gear

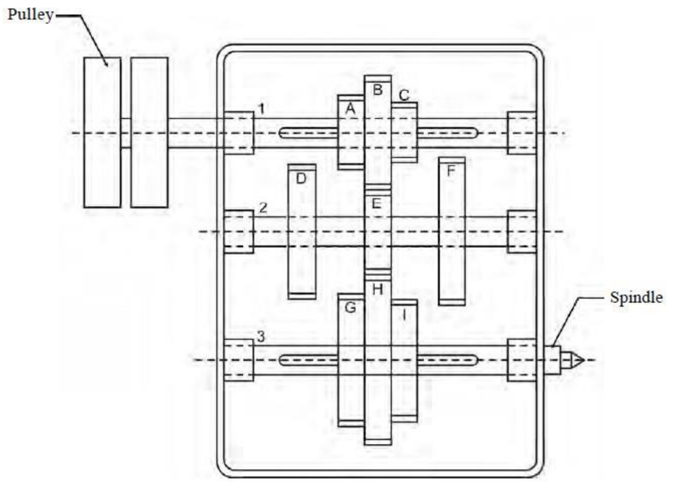


Fig 3: All gear drive

3 Spindle

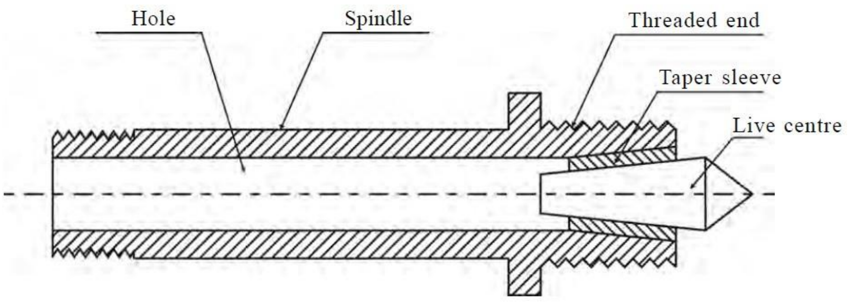


Fig 4: Head stock spindle

The spindle rotates on two large bearings housed on the headstock casting. A hole extends through the spindle so that a long bar stock may be passed through the hole. The front end of the spindle is threaded on which chucks, faceplate, driving plate and catch plate are screwed. The front end of the hole is tapered to receive live centre. which supports the work. On the other side of the spindle, a gear known as a spindle gear is fitted. Through this gear, tumbler gears and a main gear train, the power is transmitted to the gear on the leadscrew.

4 Tailstock

Tailstock is located on the inner guide ways at the right side of the bed opposite to the headstock. The body of the tailstock is bored and houses the tailstock spindle. The spindle moves front and back inside the hole. The spindle has a taper hole to receive the dead centre or shanks of tools like drill or reamer. If the tailstock hand wheel is rotated in the clockwise direction, the spindle advances. The spindle will be withdrawn inside the hole, if the hand wheel is rotated in anti- clockwise direction.

The main uses of tailstock are:

1. It supports the other end of the long workpiece when it is machined between centres.
2. It is useful in holding tools like drills, reamers and taps when performing drilling, reaming and tapping.

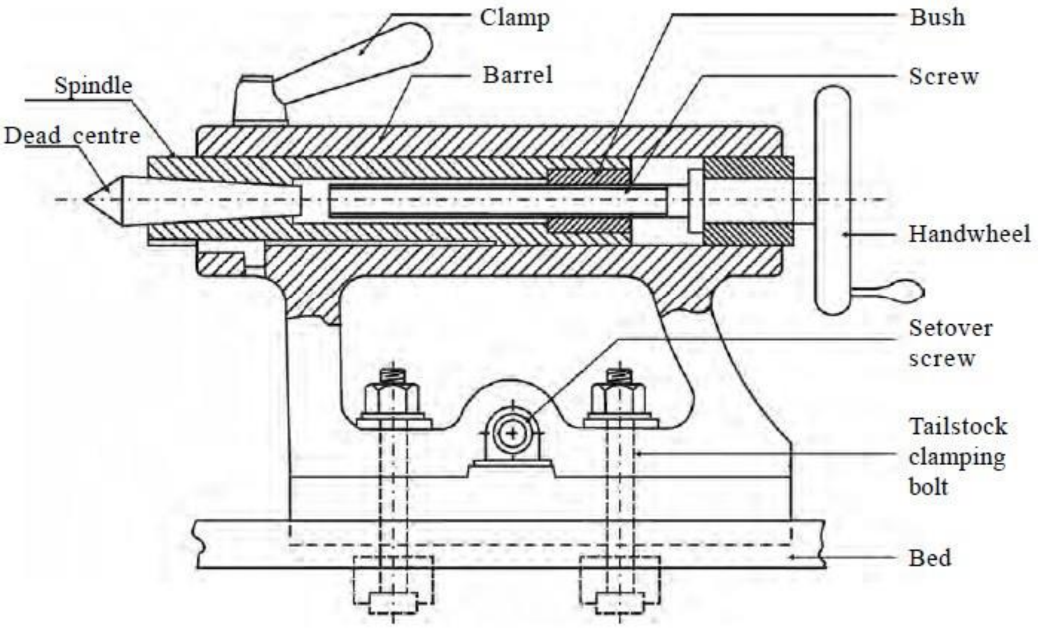


Fig 5: Tail stock

5 Carriage

Carriage is located between the headstock and tailstock on the lathe bed guide ways. It can be moved along the bed either towards or away from the headstock. It has several parts to support, move and control the cutting tool. The parts of the carriage are:

a) saddle
b) apron
c) cross-slide
d) compound rest
e) compound slide
f) tool post

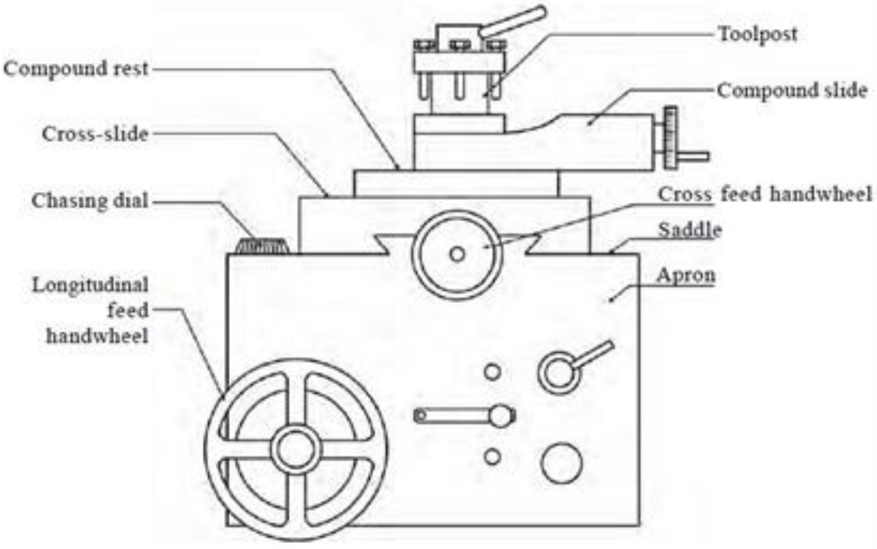


Fig 6: Carriage

Saddle:

It is an “H” shaped casting. It connects the pair of bed guide ways like a bridge. It fits over the bed and slides along the bed between headstock and tailstock. The saddle or the entire carriage can be moved by providing hand feed or automatic feed.

Cross slide:

Cross-slide is situated on the saddle and slides on the dovetail guide ways at right angles to the bed guide ways. It carries compound rest, compound slide and tool post. Cross slide hand wheel is rotated to move it at right angles to the lathe axis. It can also be power driven. The cross slide hand wheel is graduated on its rim to enable to give known amount of feed as accurate as 0.05mm.

Compound rest:

Compound rest is a part which connects cross slide and compound slide. It is mounted on the cross- slide by tongue and groove joint. It has a circular base on which angular graduations are marked. The compound rest can be swivelled to the required angle while turning tapers. A top slide known as compound slide is attached to the compound rest by dove tail joint. The tool post is situated on the compound slide.

Tool post:

This is located on top of the compound slide. It is used to hold the tools rigidly. Tools are selected according to the type of operation and mounted on the tool post and adjusted to a convenient working position. There are different types of tool posts and they are:

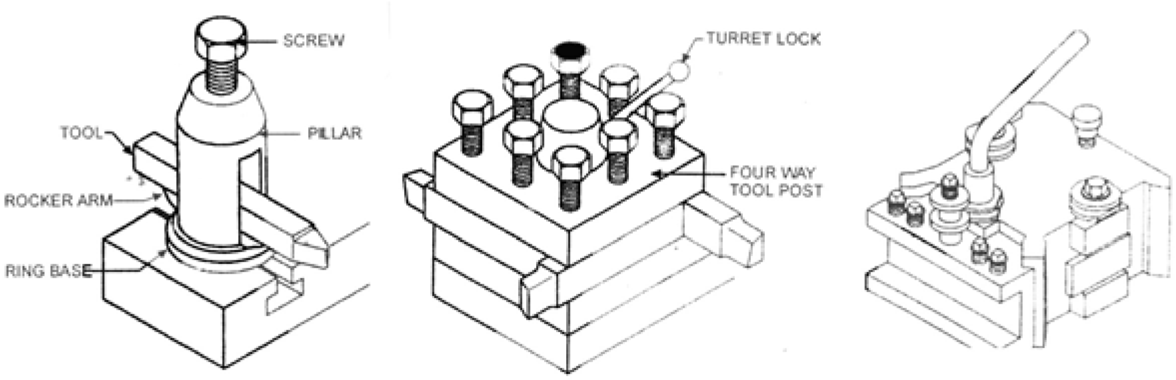
1. Single way tool post
2. Four-way tool post
3. Quick change tool post

Single way tool post

The tool is held by a screw in this tool post. It consists of a round bar with a slotted hole in the center for fixing the tool by means of a setscrew. A concave ring and a convex rocker are used to set the height of the tool point at the right position. The tool fits on the flat top surface of the rocker. The tool post is not rigid enough for heavy works as only one clamping screw is used to clamp the tool.

Four-way tool post

This type of tool post can accommodate four tools at a time on the four open sides of the post. The tools are held in position by separate screws and a locking bolt is located at the center. The required tool may be set for machining by swivelling the tool post. Machining can be completed in a shorter time because the required tools are pre-set.



Single way (Pillar Type) Tool Post

Indexing Type 4 way Tool Post

Quick Change Tool Post

Fig 6: Types of tool posts

6. Lead screw

The leadscrew is a long threaded shaft used as master screw. It is brought into operation during thread cutting to move the carriage to a calculated distance. Mostly leadscrew is Acme threaded.

The leadscrew is held by two bearings on the face of the bed. A gear is attached to the lead screw and it is called as gear on leadscrew. A half nut lever is provided in the apron to engage half nuts with the leadscrew.

7. Feed rod

Feed rod is placed parallel to the leadscrew on the front side of the bed. It is a long shaft which has a keyway along its length. The power is transmitted from the spindle to the feed rod through tumbler gears and a gear train. It is useful in providing feed movement to the carriage except for thread cutting and to move cross-slide. A worm mounted on the feed rod enables the power feed movements.

Types of lathe

Various designs and constructions of lathe have been developed to suit different machining conditions and usage. The following are the different types of lathe:

1. Speed lathe
   a. Woodworking lathe
   b. Centering lathe
   c. Polishing lathe
   d. Metal spinning lathe
3. Bench lathe
5. Semi-automatic lathe
   a. Capstan lathe
   b. Turret lathe
6. Automatic lathe
2. Engine lathe
   a. Belt driven lathe
   b. Individual motor driven lathe
   c. Gear head lathe
4. Tool room lathe
7. Special purpose lathe
   a. Wheel lathe
   b. Gap bed lathe
   c. ‘T’ lathe
   d. Duplicating lathe

Specification of Lathe

The size of a lathe is specified by the following points:

1. The length of the bed
2. Maximum distance between live and dead centres.
3. The height of centres from the bed
4. The swing diameter:

    The swing diameter over bed - It refers to the largest diameter of the work that will be rotated without touching the bed

    The swing diameter over carriage - It is the largest diameter of the work that will revolve over the saddle.

5. The bore diameter of the spindle
6. The width of the bed
7. The type of the bed
8. Pitch value of the lead screw
9. Horse power of the motor
10. Number and range of spindle speeds
11. Number of feeds
12. Spindle nose diameter
13. Floor space required
14. The type of the machine

Working Principle of lathe machine

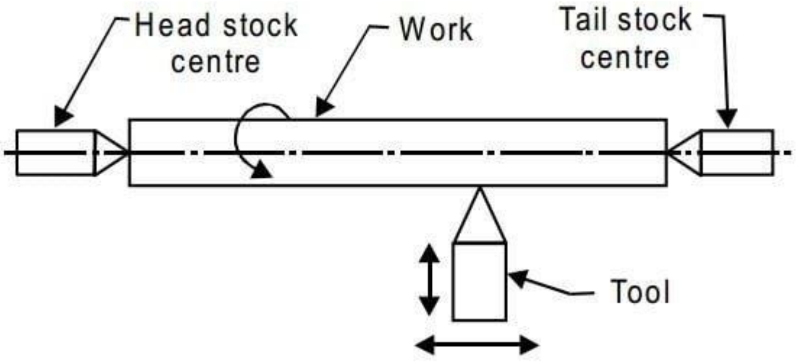
Principle

A lathe machine works on the principle of a rotating workpiece and a fixed cutting tool while a cutting tool removes material from the workpiece to give the desired shape and size.

This rotation is achieved by mounting the workpiece between two supports or in a chuck or faceplate

When the tool is moved parallel to the work-piece then the cylindrical surface is formed.

If the tool is moved inclined to the axis, then it produces a tapered surface and so calls as taper turning.



Principle diagram of lathe

Working

- It holds the work between two supports so call as centers.
- Face plate or Chuck are using for holding the work.
- Face plate or Chuck are mounted on the machine spindle.
- The cutting tool is holding with the help of Tool post.
- The movement of the job is rotating about the spindle axis.
- Against the revolving work, the tool is feed.
- The tool moves either parallel or inclination to the work axis.

Work holding devices used in a lathe (accessories)

The work holding devices are used to hold and rotate the workpieces along with the spindle. Different work holding devices are used according to the shape, length, diameter and weight of the workpiece and the location of turning on the work.

They are:

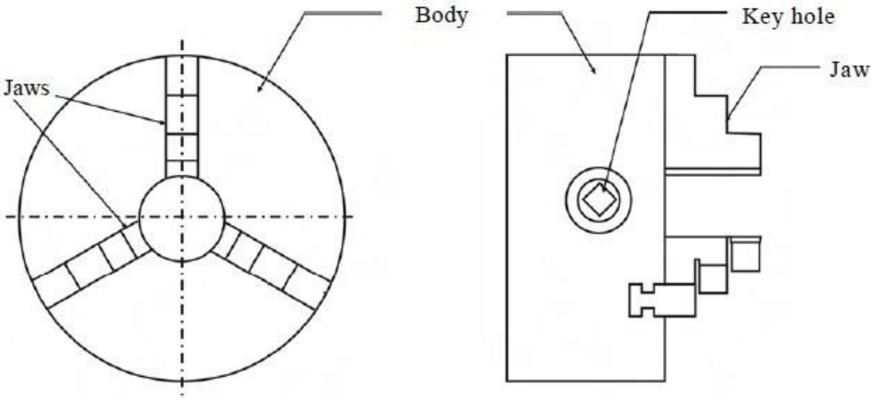
1. Chucks
2. Face plate
3. Driving plate
4. Catch plate
5. Carriers
6. Mandrels
7. Centers
8. Rests

1. Chucks

Workpieces of short length, large diameter and irregular shapes, which cannot be mounted between centers, are held quickly and rigidly in chuck. There are different types of chucks namely, Three jaw universal chuck, Four jaw independent chuck, Magnetic chuck, Collet chuck and Combination chuck.

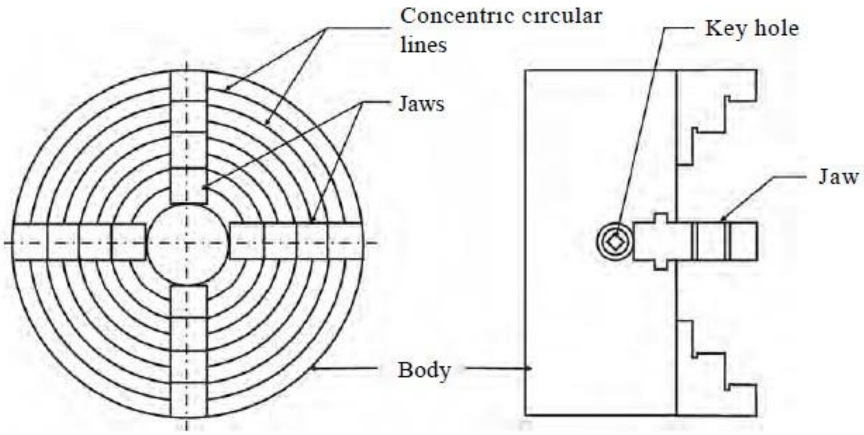
Three jaw self-Centering chuck

The three jaws fitted in the three slots may be made to slide at the same time by an equal amount by rotating any one of the three pinions by a chuck key. This type of chuck is suitable for holding and rotating regular shaped workpieces like round or hexagonal rods about the axis of the lathe. Workpieces of irregular shapes cannot be held by this chuck. The work is held quickly and easily as the three jaws move at the same time.



Four jaw independent chuck

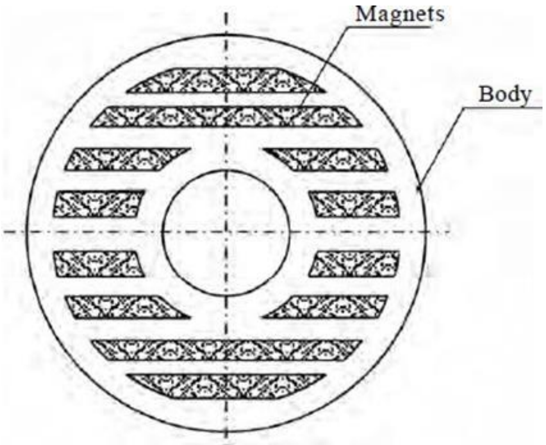
There are four jaws in this chuck. Each jaw is moved independently by rotating a screw with the help of a chuck key. A particular jaw may be moved according to the shape of the work.



Hence this type of chuck can hold woks of irregular shapes. But it requires more time to set the work aligned with the lathe axis. Experienced turners can set the work about the axis quickly. Concentric circles are inscribed on the face of the chuck to enable quick Centering of the workpiece.

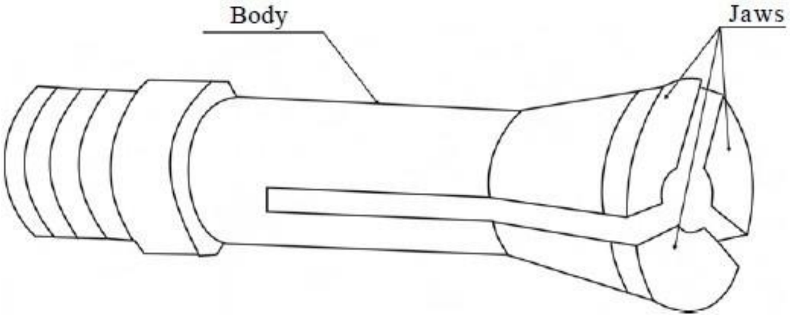
Magnetic chuck

The holding power of this chuck is obtained by the magnetic flux radiating from the electromagnet placed inside the chuck. Magnets are adjusted inside the chuck to hold or release the work. Workpieces made of magnetic material only are held in this chuck. Very small, thin and light works which cannot be held in an ordinary chuck are held in this chuck.



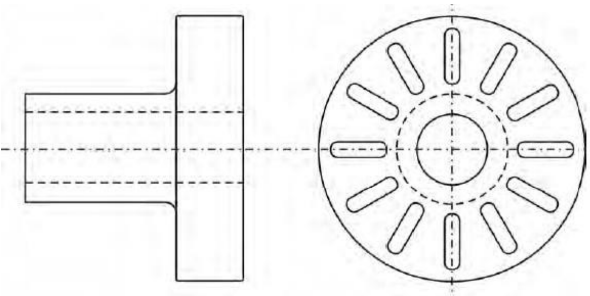
Collet chuck

Collet chuck has a cylindrical bushing known as collet. It is made of spring steel and has slots cut lengthwise on its circumference. So, it holds the work with more grips. Collet chucks are used in capstan lathes and automatic lathes for holding bar stock in production work.



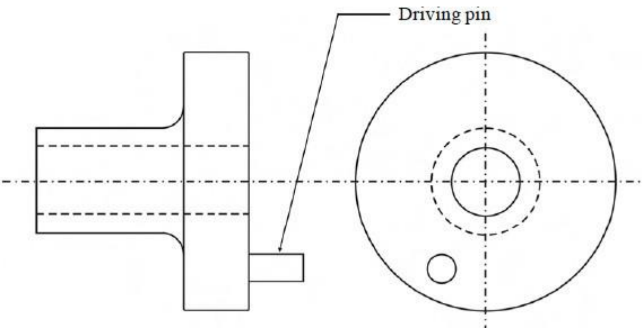
2. Face plate

Faceplate is used to hold large, heavy and irregular shaped workpieces which can not be conveniently held between centres. It is a circular disc bored out and threaded to fit to the nose of the lathe spindle. It is provided with radial plain and 'T' – slots for holding the work by bolts and clamps.



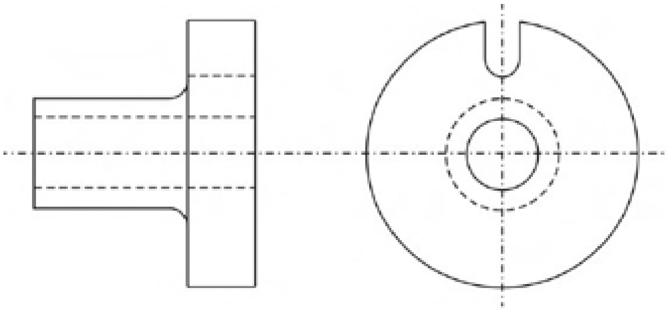
3. Driving plate

The driving plate is used to drive a workpiece when it is held between centres. It is a circular disc screwed to the nose of the lathe spindle. It is provided with small bolts or pins on its face. Workpieces fitted inside straight tail carriers are held and rotated by driving plates.



4. Catch plate

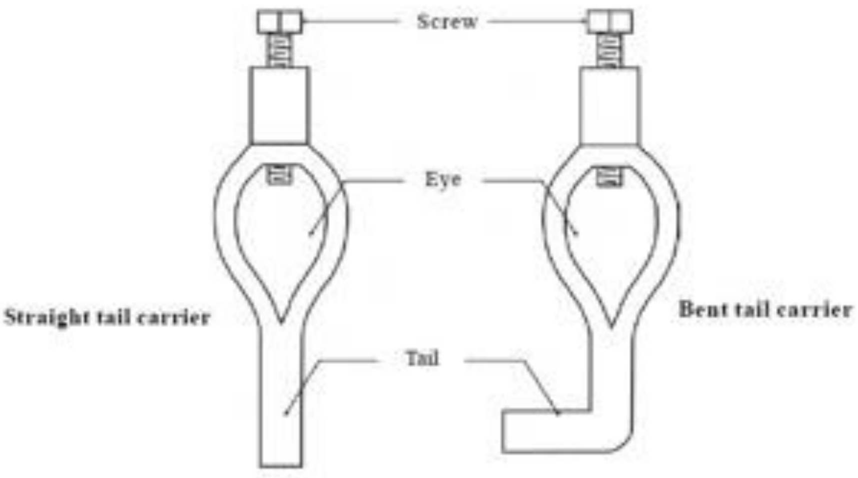
When a workpiece is held between centres, the catch plate is used to drive it. It is a circular disc bored and threaded at the centre. Catch plates are designed with 'U' – slots or elliptical slots to receive the bent tail of the carrier. Positive drive between the lathe spindle and the workpiece is affected when the workpiece fitted with the carrier fits into the slot of the catch plate.



Catch plate

5. Carrier

When a workpiece is held and machined between centres, carriers are useful in transmitting the driving force of the spindle to the work by means of driving plates and catch plates. The work is held inside the eye of the carrier and tightened by a screw. Carriers are of two types and they are:



Carrier |

1. Straight tail carrier
2. Bent tail carrier

Straight tail carrier is used to drive the work by means of the pin provided in the driving plate. The tail of the bent tail carrier fits into the slot of the catch plate to drive the work.

6. Mandrel

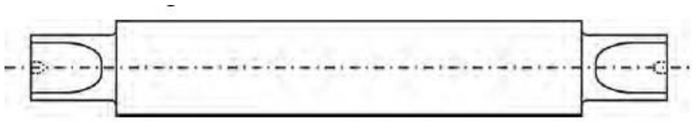
A previously drilled or bored workpiece is held on a mandrel to be driven in a lathe and machined. There are centre holes provided on both faces of the mandrel. The live centre and the dead centre fit into the centre holes. A carrier is attached at the left side of the mandrel. The mandrel gets the drive either through a catch plate or a driving plate. The workpiece rotates along with the mandrel.

There are several types of mandrels and they are:

1. Plain mandrel
2. Step mandrel
3. Gang mandrel
4. Screwed mandrel
5. Cone mandrel
6. Expansion mandrel

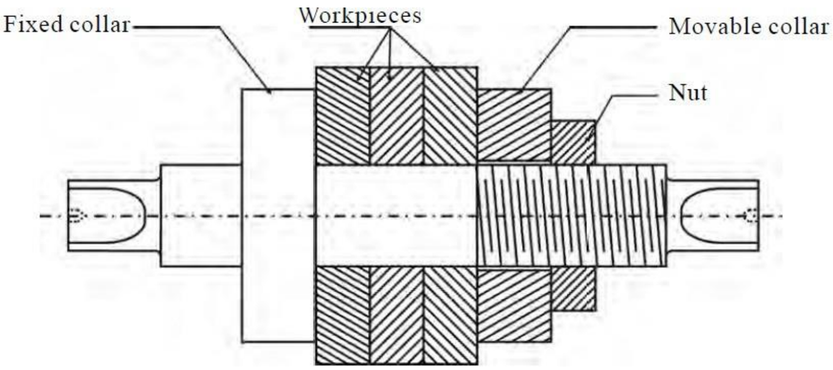
Plain mandrel

The body of the plain mandrel is slightly tapered to provide proper gripping of the workpiece. The taper will be around 1 to 2mm for a length of 100mm. It is also known as solid mandrel. It is the type mostly commonly used and has wide application.



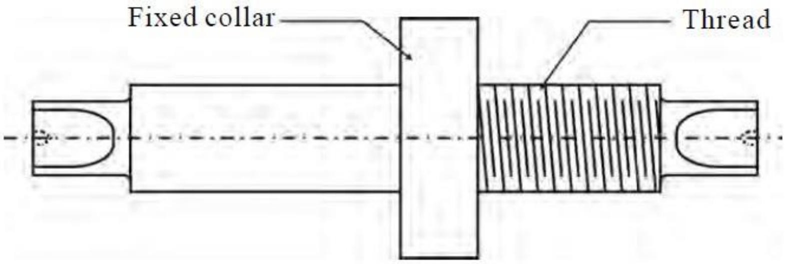
Gang mandrel

It has a fixed collar at one end and a movable collar at the threaded end. This mandrel is used to hold a set of hollow workpieces between the two collars by tightening the nut.



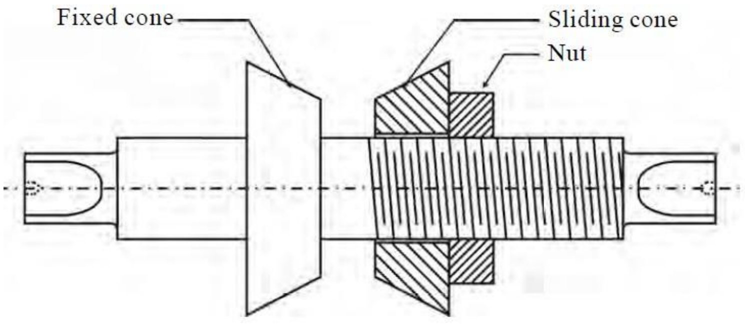
Screwed mandrel

It is threaded at one end and a collar is attached to it. Workpieces having internal threads are screwed on to it against the collar for machining.



Cone mandrel

It consists of a solid cone attached to one end of the body and a sliding cone, which can be adjusted by turning a nut at the threaded end. This type is suitable for driving workpieces having different hole diameters.



7. Centres

Centres are useful in holding the work in a lathe between centres. The shank of a centre has Morse taper on it and the face is conical in shape. There are two types of centres namely

(i) Live centre (ii) Dead centre

The live centre is fitted on the headstock spindle and rotates with the work. The centre fitted on the tailstock spindle is called dead centre. It is useful in supporting the other end of the work. Centres are made of high carbon steel and hardened and then tempered. So the tip of the centres are wear resistant.

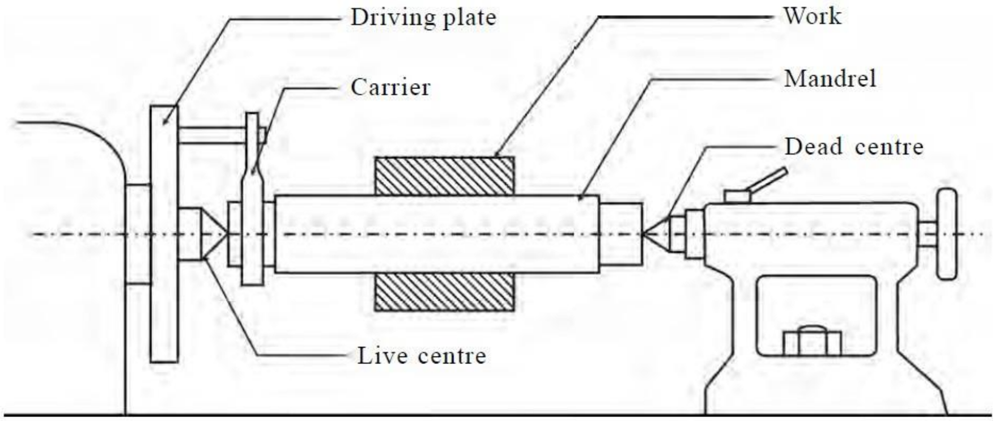


Fig 1.27 Holding a work between centres

Different types of centres are available according to the shape of the work and the operation to be performed. They are:

1. Ordinary centre
2. Ball centre
3. Half centre
4. Tipped centre
5. Pipe centre
6. Revolving centre
7. Inserted type centre

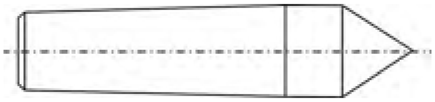
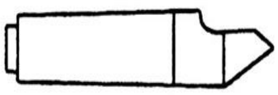
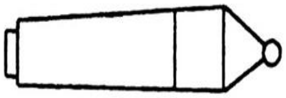
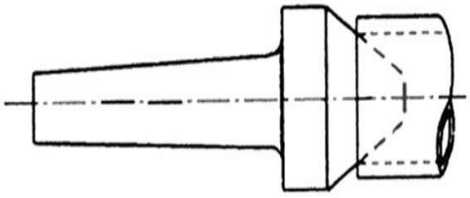
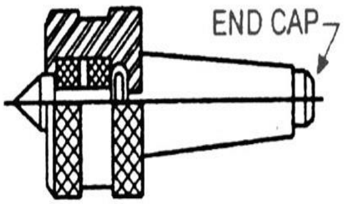


Fig : Ordinary centre

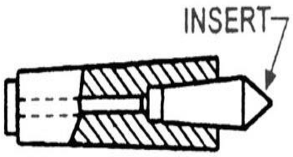

Half centre


Ball centre

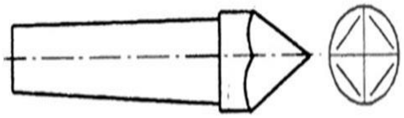

Pipe centre

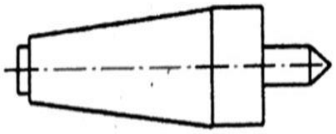


Revolving centre



Insert type centre


Square centre


Stepped centre

OPERATIONS PERFORMED IN A LATHE

Various operations are performed in a lathe other than plain turning. They are:

1. Facing
2. Turning
    a. Straight turning
    b. Step turning
3. Chamfering
4. Grooving
5. Forming
6. Knurling
7. Undercutting
8. Eccentric turning
9. Taper turning
10. Thread cutting
11. Drilling
12. Reaming
13. Boring
14. Tapping
15. Parting-off

Facing:

Facing is the operation of machining the ends of a piece of work to produce flat surface which is square with the axis. The operation involves feeding the tool perpendicular to the axis of rotation of the work.

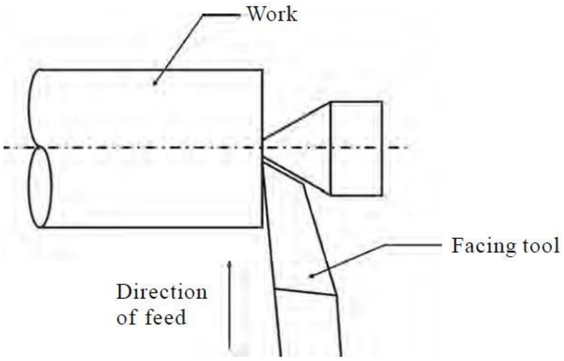


Fig 1.31 Facing

Turning

Turning in a lathe is to remove excess material from the workpiece to produce a cylindrical surface of required shape and size.

Straight turning

The work is turned straight when it is made to rotate about the lathe axis and the tool is fed parallel to the lathe axis. The straight turning produces a cylindrical surface by removing excess metal from the workpieces.

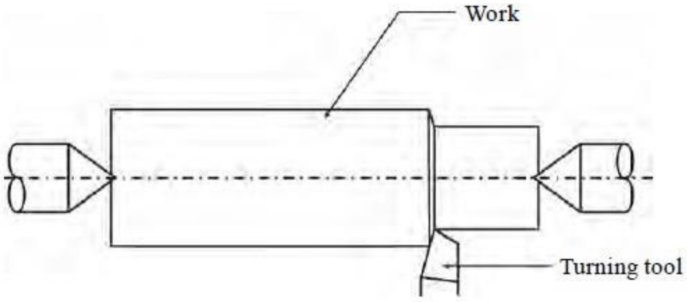
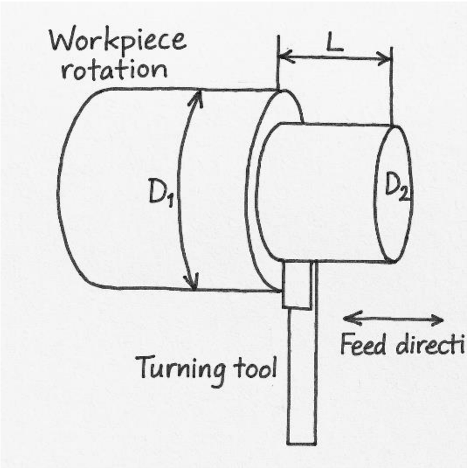


Fig 1.32 Straight turning

Step turning

Step turning is the process of turning different surfaces having different diameters. The work is held between centres and the tool is moved parallel to the axis of the lathe. It is also called shoulder turning.



Chamfering

Chamfering is the operation of bevelling the extreme end of the workpiece. The form tool used for taper turning may be used for this purpose. Chamfering is an essential operation after thread cutting so that the nut may pass freely on the threaded workpiece.

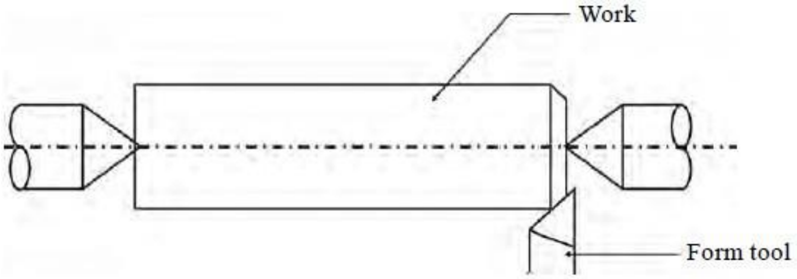
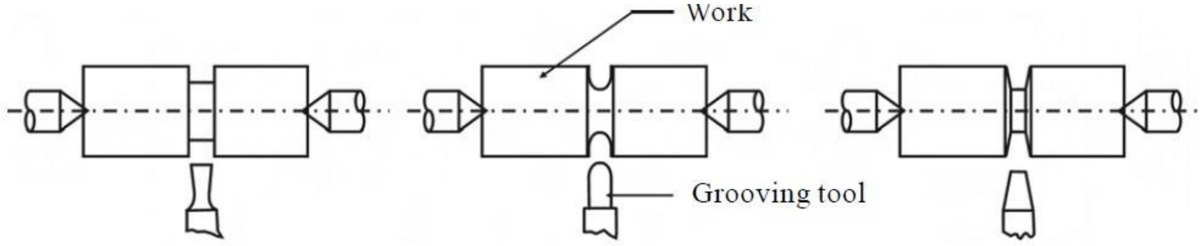


Fig 1.33 Chamfering

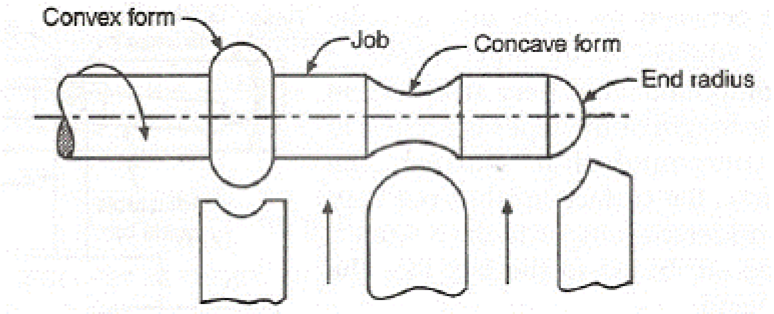
Grooving

Grooving is the process of cutting a narrow groove on the cylindrical surface of the workpiece. It is often done at end of a thread or adjacent to a shoulder to leave a small margin. The groove may be square, radial or bevelled in shape.



Forming

Forming is a process of turning a convex, concave or any irregular shape. For turning a small length formed surface, a forming tool having cutting edges conforming to the shape required is fed straight into the work.

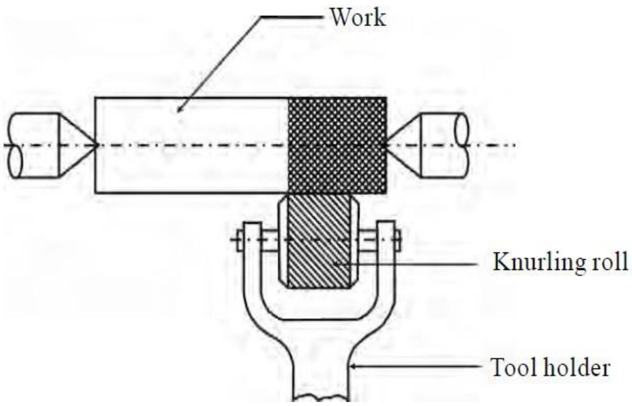


Knurling

Knurling is the process of embossing a diamond shaped pattern on the surface of the workpiece. The knurling tool holder has one or two hardened steel rollers with edges of required pattern. The tool holder is pressed against the rotating work. The rollers emboss the required pattern. The tool holder is fed automatically to the required length. Knurls are available in coarse, medium and fine pitches. The patterns may be straight, inclined or diamond shaped.

The purpose of knurling is

1. To provide an effective gripping surface
2. To provide better appearance to the work
3. To slightly increase the diameter of the work

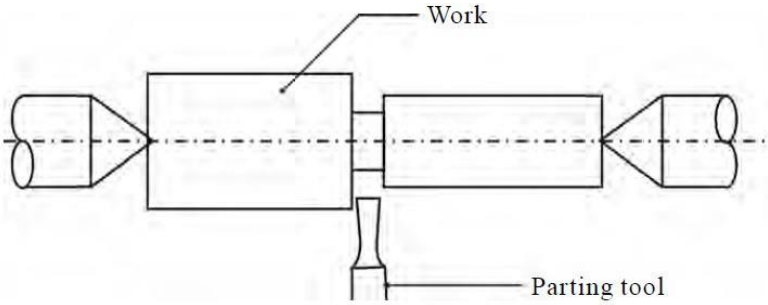


Undercutting

Undercutting is done

(i) at the end of a hole (ii) near the shoulder of stepped cylindrical surfaces

(iii) at the end of the threaded portion in bolts. It is a process of enlarging the diameter if done internally and reducing the diameter if done externally over a short length. It is useful mainly to make fits perfect. Boring tools and parting tools are used for this operation.



Taper turning

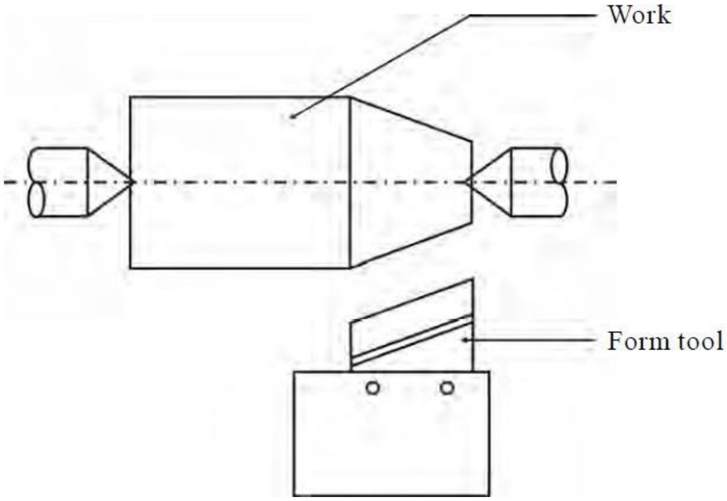
A taper may be defined as a uniform increase or decrease in diameter of a piece of work measured along its length.

Taper turning methods,

1. Form tool method
2. Compound rest method
3. Tailstock set over method
4. Taper turning attachment method
5. Combined feed method

(i) Form tool method

A broad nose tool is ground to the required length and angle. It is set on the work by providing feed to the cross-slide. When the tool is fed into the work at right angles to the lathe axis, a tapered surface is generated. This method is limited to turn short lengths of taper only. The length of the taper is shorter than the length of the cutting edge. Less feed is given as the entire cutting edge will be in contact with the work.



(ii) Compound rest method

The compound rest of the lathe is attached to a circular base graduated in degrees, which may be swivelled and clamped at any desired angle. The angle of taper is calculated using the formula:

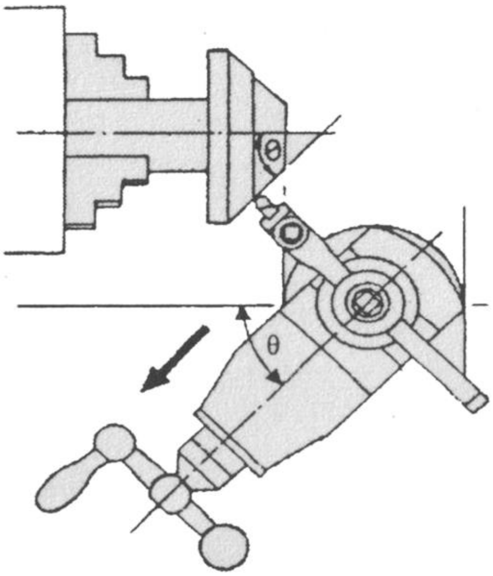
where D = Larger diameter

d = Smaller diameter

l = Length of the taper

θ = Half taper angle

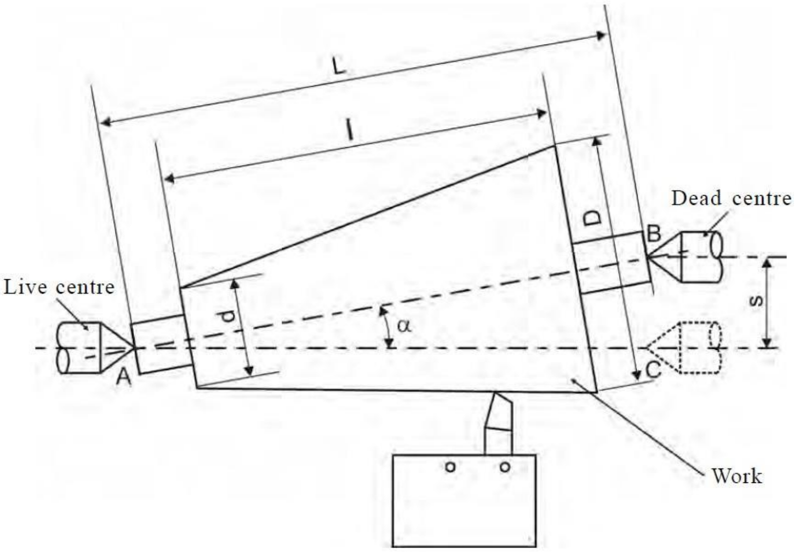
The compound rest is swivelled to the angle calculated as above and clamped. Feed is given to the compound slide to generate the required taper.



Taper turning by swivelling the compound rest

(iii) Tailstock set over method

Turning taper by the set over method is done by shifting the axis of rotation of the workpiece at an angle to the lathe axis and feeding the tool parallel to the lathe axis. The construction of tailstock is designed to have two parts namely the base and the body. The base is fitted on the bed guide ways and the body having the dead centre can be moved at cross to shift the lathe axis.



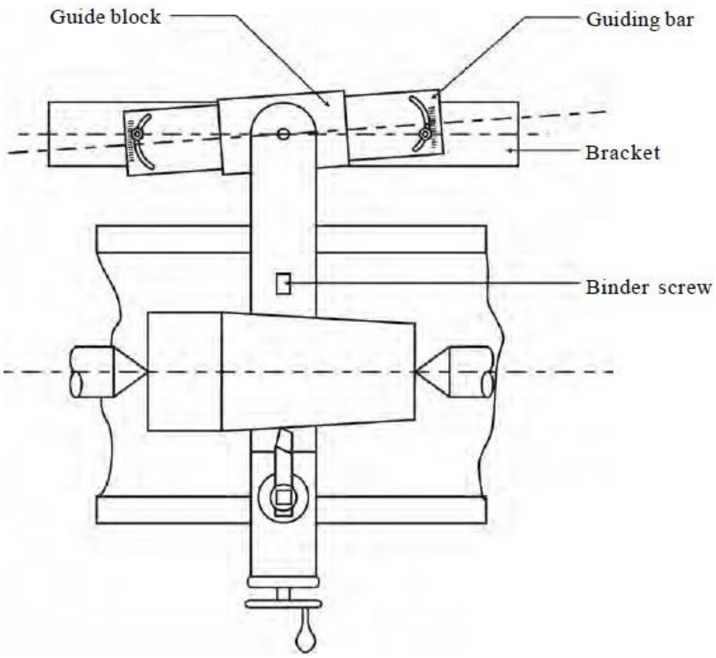
The amount of set over (s) can be calculated as follows:

$$setover, s = L * \frac{D - d}{2l}$$

The dead centre is suitably shifted from its original position to the calculated distance. The work is held between centres and longitudinal feed is given by the carriage to generate the taper. The advantage of this method is that the taper can be turned to the entire length of the work. Taper threads can also be cut by this method. The amount of set over being limited, this method is suitable for turning small tapers (approx. upto 8°). Internal tapers cannot be done by this method.

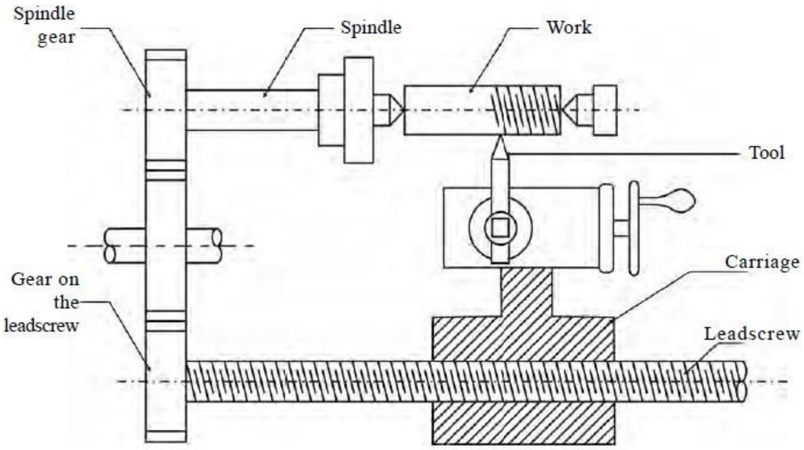
(iv) Taper turning by an attachment

The taper attachment consists of a bracket which is attached to the rear end of the lathe bed. It supports a guide bar pivoted at the centre. The bar having graduation in degrees may be swivelled on either side of the zero graduation and set at the desired angle to the lathe axis. A guide block is mounted on the guide bar and slides on it. The cross slide is made free from its screw by removing the binder screw. The rear end of the cross slide is tightened with the guide block by means of a bolt. When the longitudinal feed is engaged, the tool mounted on the cross slide will follow the angular path as the guide block will slide on the guide bar set at an angle of the lathe axis. The depth of cut is provided by the compound slide which is set parallel to the cross-slide. The advantage of this method is that long tapers can be machined. As power feed can be employed, the work is completed at a shorter time. The disadvantage of this method is that internal tapers cannot be machined.



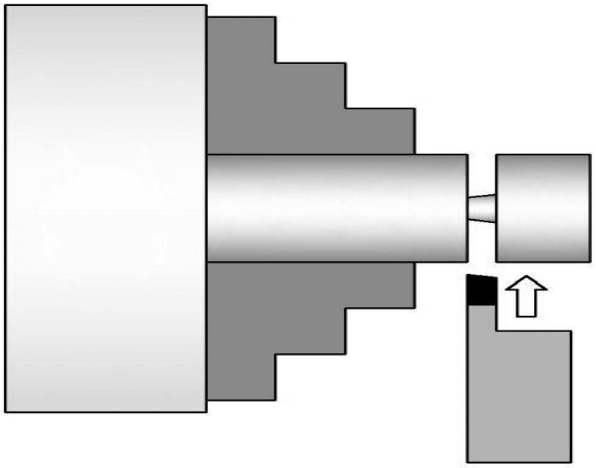
Thread cutting

Thread cutting is one of the most important operations performed in a lathe. The process of thread cutting is to produce a helical groove on a cylindrical surface by feeding



the tool longitudinally. The job is revolved between centres or by a chuck. The longitudinal feed should be equal to the pitch of the thread to be cut per revolution of the work piece.

Parting, Parting Off



Parting, also known as parting off, is a lathe machining operation used to cut off a workpiece from a larger stock material. The primary purpose of parting is to separate a finished or semi-finished component from the rest of the material, creating individual pieces with specific lengths. This process is essential for producing individual components from long bars of material.

Tools used in a lathe

Tools used in a lathe are classified as follows

A. According to the construction, the lathe tools are classified into three types
  1. Solid tool
  2. Brazed tipped tool
  3. Tool bit and tool holders

B. According to the operation to be performed, the cutting tools are classified as
  1. Turning tool
  2. Thread cutting tool
  3. Facing tool
  4. Forming tool
  5. Parting tool
  6. Grooving tool
  7. Boring tool
  8. Internal thread cutting tool
  9. Knurling tool

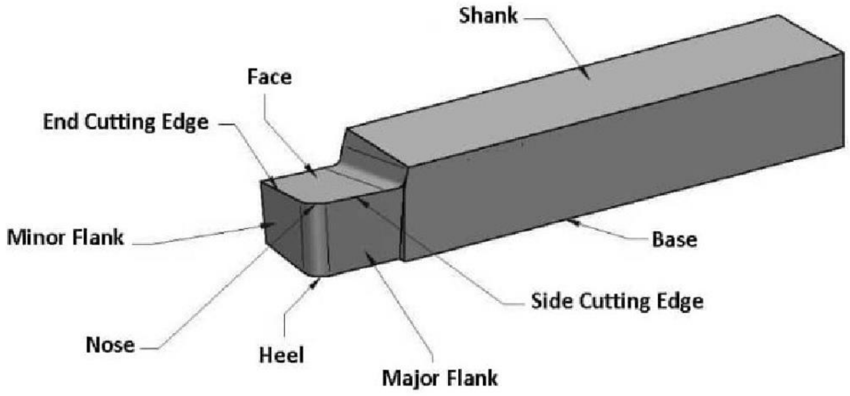
C. According to the direction of feed movement, the following tools are used

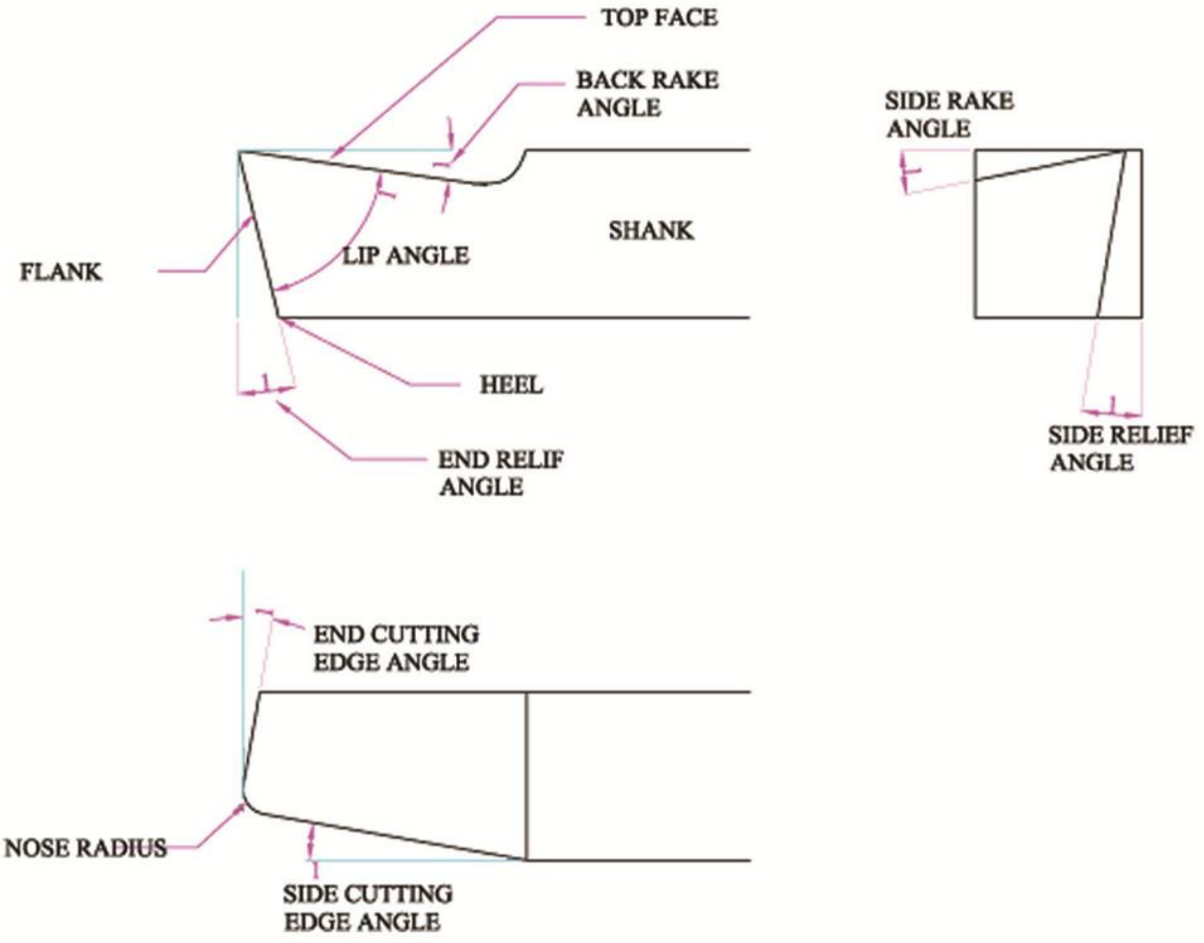
1. Right hand tool
2. Left hand tool
3. Round nose tool

CUTTING TOOL GEOMETRY AND SELECTION:

Tool Nomenclature

It means the systematic naming of the various parts and angles of a cutting tool.





Shank

The tool holder on the machine recognizes the shank, which is the main body of the single point cutting tool. In terms of volume, it makes up a bigger portion of the tool.

Face

The face of the tool is where chips move upward and outward during the machining process.

Base

In single-point cutting tools, the bottom surface on which the tool stands is called the base.

Nose or Cutting Point

There is a point in front of the tool where the side and end cutting edges meet, known as a tool nose. It is also referred to as the tool’s cutting point.

Nose Radius

The radius of the nose is known as the nose radius, and it contributes to better surface finishing and longer tool life.

Heel

The heel of a single-point cutting tool is defined as the point at which the flank and base surfaces meet.

Flank

The major flank and minor flank are the two flank surfaces found on cutting tools. A vertical surface next to the side cutting edge is the major flank. A minor flank is a vertical surface that lies close to the cutting edge at the end. However, neither surface is actually vertical; rather, they both have a slight incline toward the base. The cutting face is also referred to as the flank.

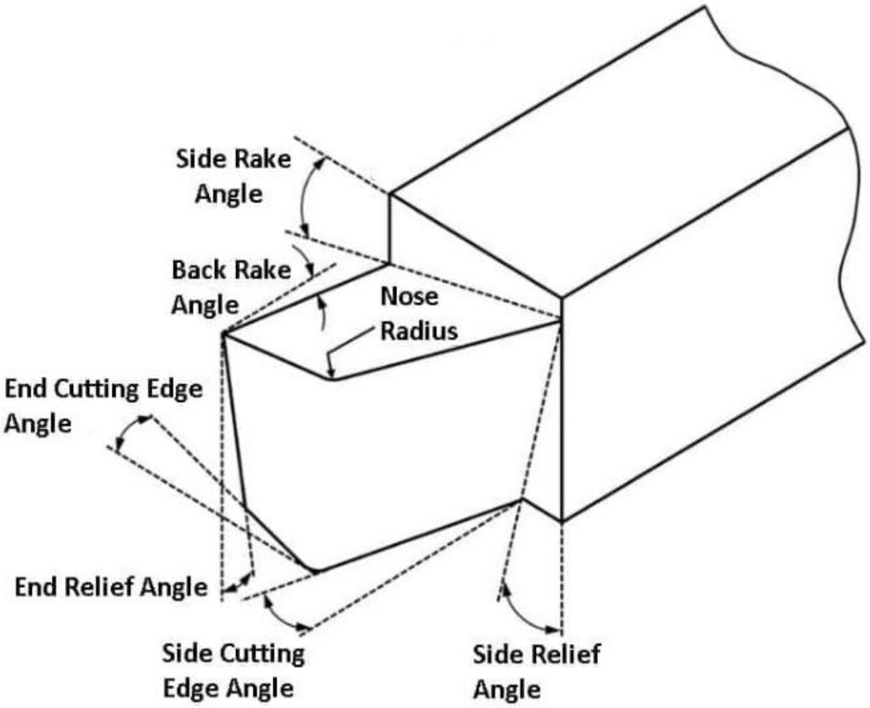
Cutting Edge

Cutting edges are those that are used to remove material during the machining process. Cutting Edges are produced by faces and cutting surfaces. Single-point cutting tools have two cutting edges: a side cutting edge and an end cutting edge.

1. **Side cutting edge**: Edge or line formed by the face and a minor flank or cutting surface.
2. **End cutting edge**: The face and the major flank or major cutting surface forms the end cutting edge, just like the side cutting edge.

Single Point Cutting Tool Angels

1. Rake angle
    - Back rake angle
    - Side rake angle
2. Relief angle
    - End relief angle
    - Side relief angle
3. Cutting edge angle
    - Side cutting edge angle
    - End cutting edge angle
4. Clearance Angle



1 Rake Angle

An essential geometric component of a single point cutting tool is the rake angle. The angle formed by the cutting edge and a line perpendicular to the surface of the workpiece is known as the rake angle.

a) Back Rake Angle

The angle formed between a single-point cutting tool's face and a horizontal line parallel to the tool's baseline is known as the back rake angle. If the side cutting edge slopes away from the point and toward the shank, the back rack angle is positive. If the slope is reversed, the back rack angle is negative.

b) Side Rake Angle

The side rake angle is measured in a plane perpendicular to the base and the side cutting edge of the tool. It is the angle formed between the tool's face and a line parallel to the tool's base If the slope is toward the cutting edge, the side rack angle is negative. Also, if the slope is away from the cutting edge, the side rack angle is positive.

2 Relief Angle

The flank and a perpendicular to the machined surface together make up the relief angle. It makes sure that the tool does not make contact with the machined surface while cutting.

a) End Relief Angle

The angle that forms a right angle with a minor flank or minor cutting face and a perpendicular line to the base is known as an end relief angle.

b) Side Relief Angle

The angle that forms a right angle with the side between the major flank or major cutting face and the perpendicular plane to the base is known as the side relief angle.

3 Cutting Edge Angle

The angle created by the cutting edge and a line perpendicular to the machined surface is known as the cutting edge angle. It is the standard angle for turning operations is 90 degrees.

a) Side Cutting Edge Angle

It is the angle made by a perpendicular line to the machined surface and the side cutting edge. It has an impact on the chip's width during cutting.

b) End Cutting Edge Angle

It is the angle made by a perpendicular line to the machined surface and the cutting edge's end. It affects the chip's thickness when cutting.

4 Clearance Angle

The clearance angle is the angle formed by a line tangent to the machined surface and the flank. It enables the tool to move freely over the surface without bumping into it.

FACTORS TO CONSIDER WHILE SELECTING CUTTING TOOL

1. **Workpiece Material:**

   Hardness: Harder materials need harder, tougher tools (carbide, ceramic).

   Type: Compatibility with steels, aluminum, composites, etc..

2. **Machining Operation & Requirements:**

   Operation Type: Roughing (material removal rate) vs. Finishing (surface finish, precision).

   Cut Quality: Finish, tolerance, burr formation.

   Chip Control: Tool geometry affects chip breaking and removal.

3. **Tool Characteristics**:

   Tool Material: High-Speed Steel (HSS) for general, Carbide for harder, Cobalt for heat, Ceramic for high speed.

   Geometry: Rake angle, clearance, edge sharpness, number of flutes.

   Coating: PVD/CVD coatings for wear resistance, heat barrier, reduced friction (e.g., TiN, AlTiN).

4. **Machine & Process Parameters:**

   Cutting Speed & Feed Rate: Must match tool and workpiece.

   Depth of Cut: Affects cutting forces and heat.

   Machine Rigidity & Spindle Speed: Influences tool stability and RPM.

   Coolant/Lubrication: Critical for temperature control and chip evacuation.

5.Economic & Practical Aspects:

Cost-Effectiveness: Tool life, regrinding ease, initial cost vs. overall savings.

Tool Life: Longevity and durability.

Ease of Use/Maintenance: Installation, adjustment, maintenance.

Tool Selection

1. **Cutting speed**

Cutting speed is an important factor in tool selection because it affects the heat generated during machining, as well as the tool wear and deformation.

Higher cutting speeds require harder and more durable tools. This is because as the cutting speed increases, the temperature of the cutting tool and the workpiece also increases, which can cause the tool to wear more quickly.

The cutting speed is influenced by several factors, including the type of material being machined, the desired surface finish, and the machining process being used.

By selecting the appropriate cutting speed, machinists can optimize machining performance, improve surface finish, and minimize tool wear and breakage.

2.Feed rate

A higher feed rate can generate more heat due to increased friction between the cutting tool and the workpiece material. This can cause tool wear and deformation, resulting in poor surface finish, dimensional inaccuracies, and even tool breakage.

On the other hand, a slower feed rate can reduce heat buildup and improve cutting tool life, but it can also slow down the machining process and reduce productivity.

The optimal feed rate depends on several factors, including the material properties of the workpiece, the type of cutting tool, the cutting speed, and the desired surface finish.

3.Chip removal

When a cutting tool is in contact with the workpiece, it removes material by cutting a chip. The chip must be removed from the cutting zone to prevent damage to the tool or the workpiece.

In cutting operations such as turning, the chip is formed continuously, and it is typically curled around the tool's cutting edge.

In these operations, chip breaking is essential to prevent the chip from wrapping around the workpiece or the tool, which can cause damage or result in poor surface finish. The design of the cutting tool and the selection of the proper feed and speed rates can affect chip formation and breakage.

During cutting operations that are interrupted, such as drilling, the chip is formed in discrete segments. The design of the drill bit must be able to break the chips into small pieces and remove them from the hole.

Inadequate chip removal can cause clogging, tool wear, and poor hole quality.

4.Machining process

Machining process refers to a broad range of manufacturing techniques used to remove material from a workpiece to create the desired shape, size, and surface finish. The most common machining processes include turning, milling and drilling. Each process requires a specific type of cutting tool with a particular geometry, coating, and other features to achieve optimal results.

The selection of these factors depends on the specific requirements of the machining process, such as the type of material, the required tolerances, and the desired surface finish.

5. Machine capability

Machining capability refers to the ability of the machining equipment to perform the required operations accurately and efficiently.

The capability of the machine tool depends on several factors, including the type of machine, its size, power and precision.

The selection of the cutting tool must consider the capability of the machine tool to perform the desired operation. If the machine tool does not have the power to handle a particular cutting tool or if its rigidity is insufficient, it may not be possible to achieve the required results.

It is also essential to consider the machining capability of the equipment when selecting the cutting tool. The tool's size, shape, and geometry should be compatible with the machine's spindle, tool holder, and other accessories

6.Surface finish

Surface finish refers to the final texture and appearance of a machined surface.

It is an important consideration in machining operations, as it can affect the function, aesthetics, and durability of the finished product. The desired surface finish can vary depending on the application, and may range from a rough, textured surface to a mirror-like polished finish.

The choice of cutting tool, cutting parameters, and machining technique can all affect the surface finish. A rough surface may be desirable for certain applications, such as for better adhesion of paint or coatings.

A smooth and polished surface may be preferred for applications that require minimal friction, such as in bearings or sliding surfaces.

Achieving the desired surface finish requires careful selection of cutting tools, appropriate machining parameters (such as cutting speed, feed rate, and depth of cut), and proper machining techniques.

7.Tolerance

Tolerance refers to the acceptable range of deviation from a specified dimension or measurement of a workpiece. It is the difference between the maximum and minimum limits of a specified dimension. In machining operations, parts must be manufactured to precise tolerances to ensure that they meet the required specifications.

The required tolerance depends on the application and the functional requirements of the part. Tighter tolerances require more precise machining, which can affect tool selection, machining process, and cost.

The tolerance of the final dimensions of the workpiece can also be affected by other factors, such as the material properties, cutting tools, machining parameters, and environmental conditions.

Machining to tighter tolerances may require more precise machining equipment and higher precision cutting tools, which can increase the cost of the machining process.

8.Volume

Volume refers to the quantity of parts that need to be produced. The volume of production is an important consideration in tool selection because it can influence the choice of tool materials and the tool's wear resistance.

If a large volume of parts needs to be produced, then a tool that can withstand high production rates and has a longer life will be more suitable.

However if only a small volume of parts is required, a less durable tool may be sufficient, and the cost of the tool may be a more significant consideration.

9.Coolant

Coolant is a liquid or gas that is used in machining operations to reduce heat generated by friction and remove chips from the cutting zone. Coolant helps to prolong tool life, improve surface finish, and prevent workpiece deformation or warping. There are several types of coolants, including water-based, oil-based, and synthetic-based.

The choice of coolant depends on several factors, including the type of material being machined, the cutting tool being used, and the machining process. Water-based coolants are typically used for low-speed machining operations, while oil-based coolants are used for high-speed machining operations. Synthetic-based coolants are often used in high-precision machining operations that require excellent surface finish and dimensional accuracy.

10. Environment

The machining environment is an important factor to consider when selecting a cutting tool. Temperature, humidity, and dust levels can all impact the performance of the tool and the quality of the finished product. High temperatures can cause tools to wear out more quickly, while high humidity can lead to rust and corrosion. Dust levels can also affect tool performance, as excessive dust can clog the cutting edge and reduce cutting efficiency.

To ensure optimal tool performance, it is important to choose a cutting tool that is suitable for the specific machining environment, if the environment is particularly dusty, a tool with a coating that resists clogging may be a good choice. If the environment is particularly humid, a tool with corrosion-resistant properties may be preferred