NMCR – 06
LATHE MACHINE TRAINING

BASIC LATHE MACHINE OPERATION

TRAINING DAYS

4 DAYS COURSE

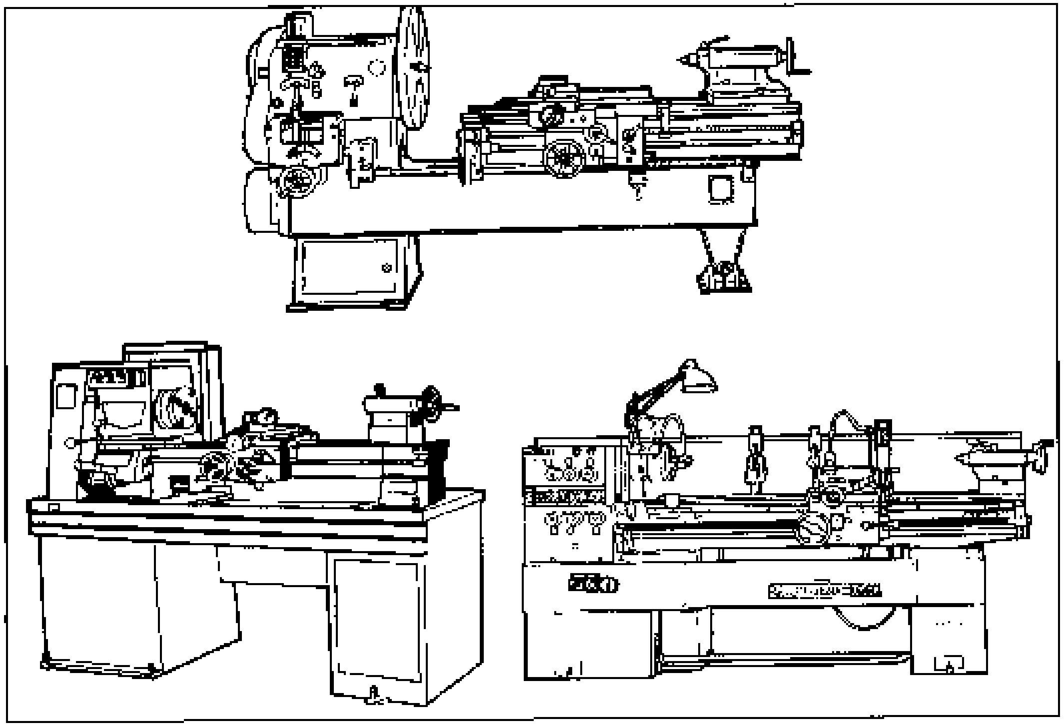
TARGET TRAINEES

- FITTER / MACHINIST
- NO. 1 OILER
- OILER
- WIPER

OBJECTIVES

- ✓ How to operate lathe machine safely.
- ✓ Procedure to carry out maintenance on lathe machine and related tools.
- ✓ Various different operations, which can be performed on lathe machine.

INTRODUCTION



HISTORICAL BACKGROUND

- The lathe is an ancient tool, dating at least to the Egyptians and, known and used in Assyria, Greece, the Roman and Byzantine Empires.
- The origin of turning dates to around 1300BC when the Egyptians first developed a two-person lathe. One person would turn the wood work piece with a rope while the other used a sharp tool to cut shapes in the wood.

- The Romans improved the Egyptian design with the addition of a turning bow. Early bow lathes were also developed and used in Germany, France and Britain.

- In the middle ages a pedal replaced hand-operated turning, freeing both the craftsman's hands to hold the woodturning tools. The pedal was usually connected to a pole, often a straight-grained sapling. The system today is called the "spring pole" lathe (see Pole lathe). Spring pole lathes were in common use into the early 20th Century.

- A two-person lathe, called a "great lathe", allowed a piece to turn continuously (like today's power lathes). A master would cut the wood while an apprentice turned the crank.

- During the industrial revolution the lathe was motorized, allowing wooden turned items to be created in less time and allowing the working of metal on a lathe. The motor also produced a greater rotational speed, making it easier to quickly produce high quality work.

- ***Henry Maudslay*** designed and builds the most efficient lathe construction. The outstanding Maudslay’s lathe is a lead screw geared to the spindle of the lathe. This invention made possible the advancement of the tool at a constant rate of speed and distance of travel.

- Today most commercial lathes are computer-operated allowing for mass-production that can be created with accurate precision and without the cost of employing craftsmen.

SIZE / CAPACITY OF LATHE

The size of a lathe is determined by the largest piece of stock that can be machined. Before machining a workpiece, the following measurements must be considered:

- **SWING**
- **LENGTH BETWEEN CENTERS**

- ***SWING*** **-** refers to the diameter of the work that can be rotated in the lathe.

- ***LENGTH of BED*** **-** maximum distance between centers of the chuck and tailstock when the tailstock is moved to the far end of the lathe.

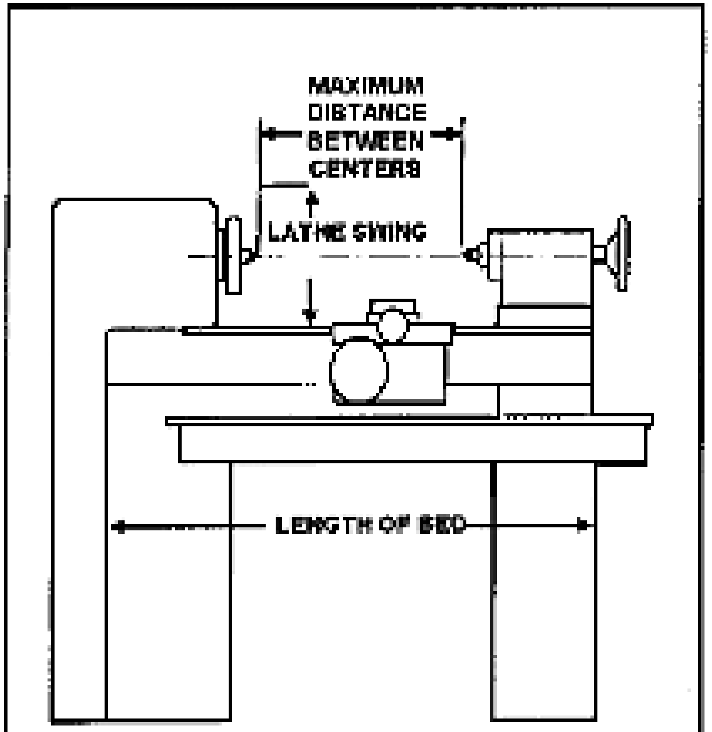


Figure 7-1. Engine lathe.

OPERATING PRINCIPLE

The lathe makes parts by spinning the part and having a cutting tool take material off of the diameter of the stock. A tool is placed in the toolholder. The part is placed in a chuck that is attached to the spindle. As the part spins, dials on the carriage and compound move the tool along the part. You can also create center drilled holes in your part by placing a drill bit in the tailstock and moving the tailstock into your part.

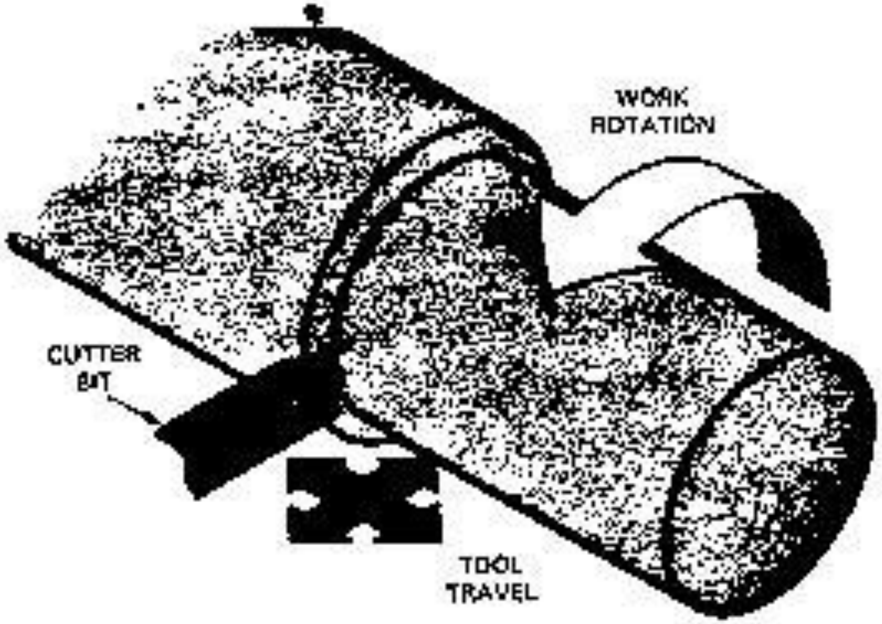
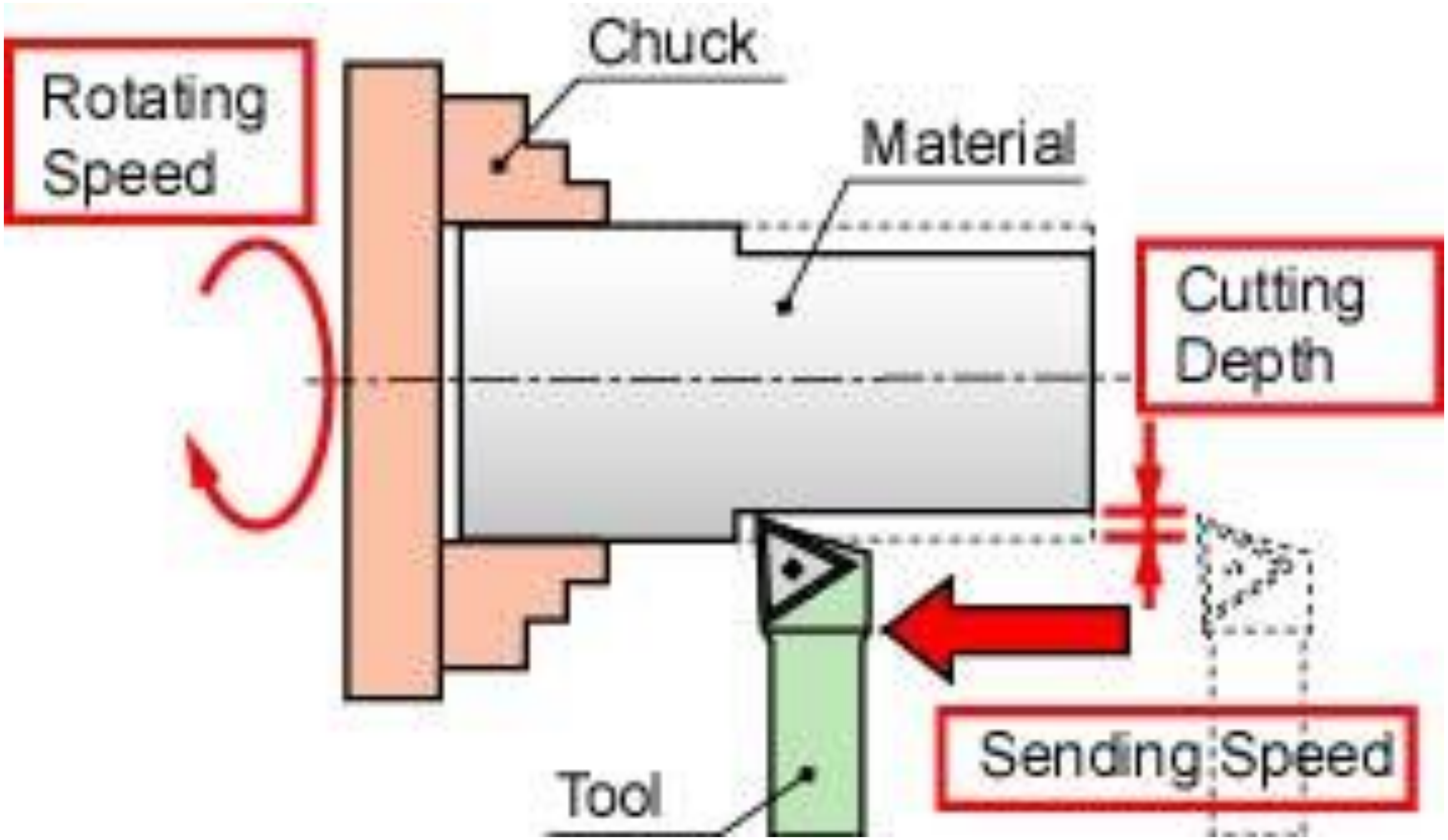


Fig. 10-2. Note operating principle of the lathe

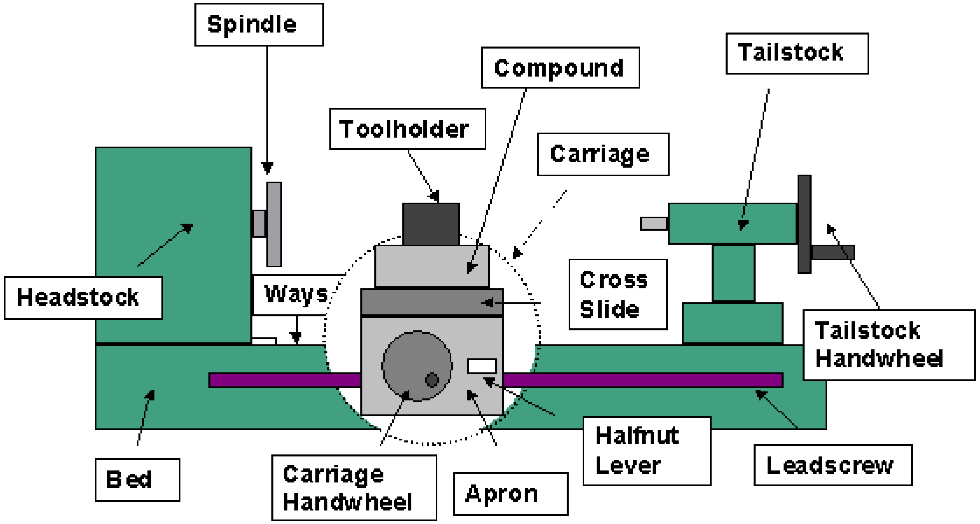
Important Elements

In order to get an efficient process and beautiful surface at the lathe machining, it is important to adjust the ***rotating speed, cutting depth*** and ***cutting speed***. Please note that the important elements can not decide easily, because these suitable values are quiet different by materials, size and shapes of the part.


Chuck
Rotating Speed
Material
Cutting Depth
Sending Speed
Tool

Rotating speed is defined as the speed at which the work moves with respect to the tool (usually measured in feet per minute). ***Feed rate*** is defined as the distance the tool travels during one revolution of the part. Cutting speed and feed rate determines the surface finish, power requirements, and material removal rate. The primary factor in choosing feed and speed is the material to be cut. However, one should also consider material of the tool, rigidity of the workpiece, size and condition of the lathe, and ***depth of cut***.

STRUCTURE AND OPERATION



MAJOR PARTS

The chief function of any lathe, no matter how complex it may appear to be, is to rotate the work against a controllable cutting tool. Each of the lathe parts in falls into one of the following categories:

- **Driving the lathe.**
- **Holding and rotating the work.**
- **Holding, moving and guiding the cutting tool.**

- ***Driving the lathe***

  Power is transmitted to the various drive mechanisms by belt drive and/or gear train.

- ***Holding and rotating the work***

  The *headstock* contains the spindle to which the various work holding attachments are fitted. The *spindle* revolves in heavy/duty bearings and is rotated by belts, gears or a combination of both. It is hollow with the front tapered internally to receive tools and attachments with taper shanks. the hole permits long stock to be turned without dangerous overhang.

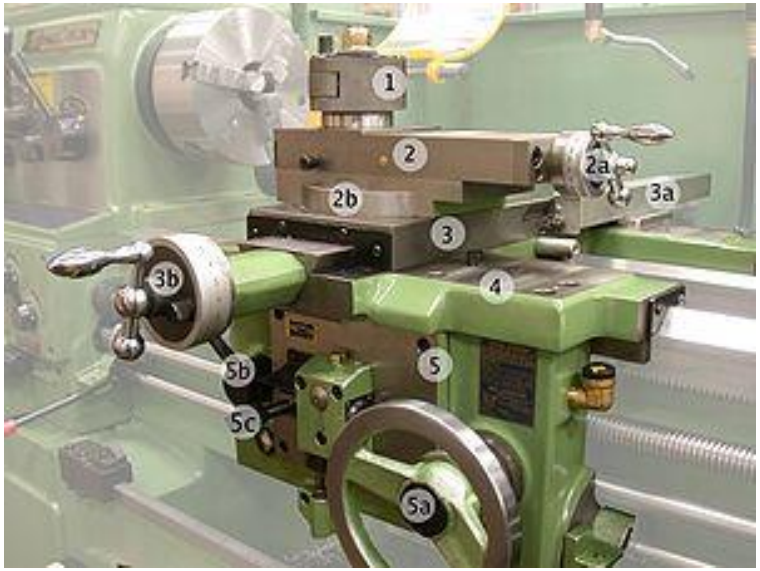

H1
HWACHEON
H2
H3
H4
H5
H6
H7
H8
H9
H10

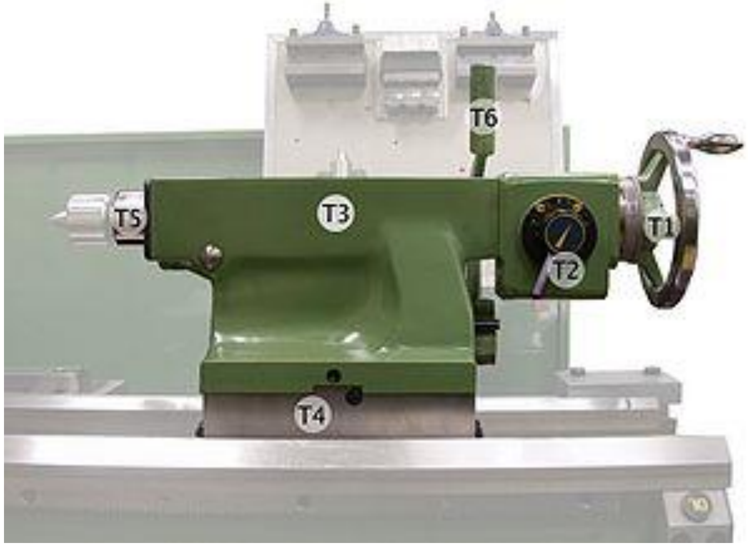
Work is held in the lathe by a chuck, faceplate, collet or between centers.

The outer end of the work is often supported by the tailstock. It can be adjusted along the ways to accommodate different lengths of work. The tailstock mounts the "dead" center, and can be fitted with tools for drilling, reaming and threading. It can also be offset for taper turning.

- ***Holding, moving and guiding the cutting tool***

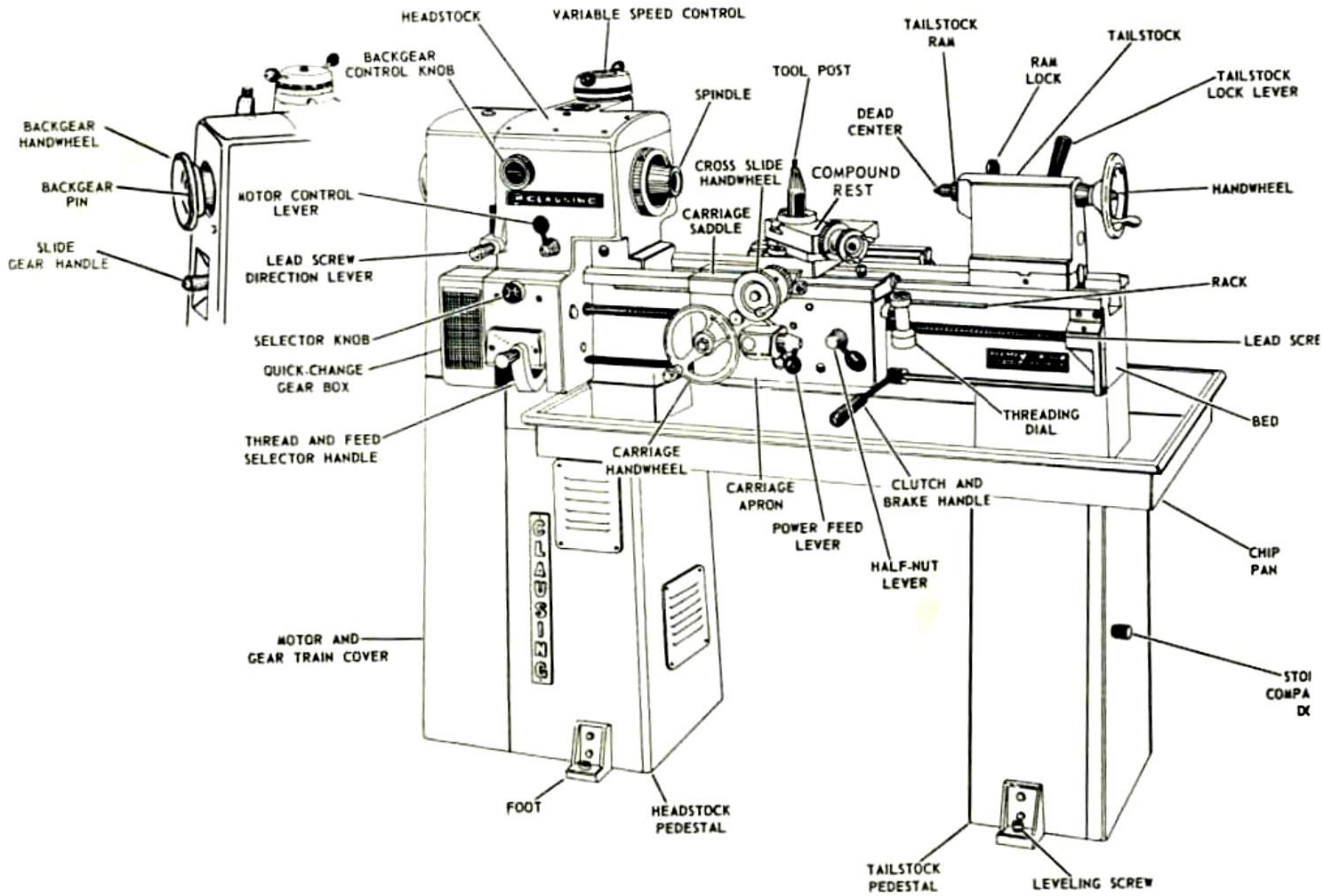
  The *bed* is the foundation of a lathe. All other parts are fitted to it. Ways are integral with the bed. The V-shape maintains precise alignment of the headstock and tailstock, and serves as rails to guide the travel of the carriage. The *cutting tool* is mounted on the carriage.


1
2
2a
2b
3a
3
3b
4
5
5b
5c
5a


T6
T5
T3
T1
T2
T4

The carriage controls and supports the cutting tool and is composed of:

- The *saddle* is fitted to and slides along the ways.
- The *apron* contains the drive mechanism to move the carriage along the ways using hand or power feed.
- The *cross slide* permits *transverse* tool movement (movement toward or away from the operator).
- The *compound rest* permits angular tool movement.
- The *tool rest* mounts the cutting tool.


HEADSTOCK
VARIABLE SPEED CONTROL
TAILSTOCK RAM
TAILSTOCK
BACKGEAR CONTROL KNOB
TOOL POST
RAM LOCK
TAILSTOCK LOCK LEVER
SPINDLE
DEAD CENTER
BACKGEAR HANDWHEEL
BACKGEAR PIN
CROSS SLIDE HANDWHEEL
COMPOUND REST
HANDWHEEL
MOTOR CONTROL LEVER
CARRIAGE SADDLE
SLIDE GEAR HANDLE
LEAD SCREW DIRECTION LEVER
RACK
SELECTOR KNOB
LEAD SCRE
QUICK-CHANGE GEAR BOX
THREADING DIAL
BED
THREAD AND FEED SELECTOR HANDLE
CARRIAGE HANDWHEEL
CARRIAGE APRON
CLUTCH AND BRAKE HANDLE
POWER FEED LEVER
HALF-NUT LEVER
CHIP PAN
MOTOR AND GEAR TRAIN COVER
CLAUSING
FOOT
HEADSTOCK PEDESTAL
TAILSTOCK PEDESTAL
LEVELING SCREW

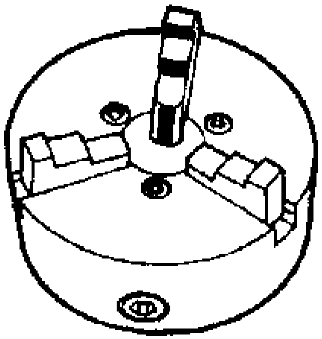
CHUCKS AND FACEPLATES

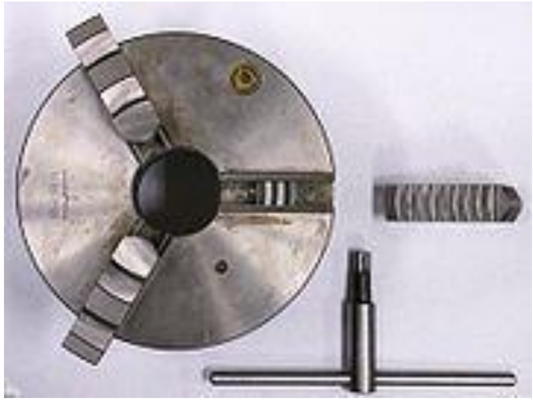
- ***Universal Scroll Chuck***

  The universal scroll chuck usually has three jaws which move in unison as an adjusting pinion is rotated. The advantage of the universal scroll chuck is its ease of operation in centering work for concentric turning. This chuck is not as accurate as the independent chuck, but when in good condition it will center work within 0.002 to 0.003 inches of runout.

The jaws are moved simultaneously within the chuck by a scroll or spiral-threaded plate. The jaws are threaded to the scroll and move an equal distance inward or outward as the scroll is rotated by the adjusting pinion. Since the jaws are individually aligned on the scroll, the jaws cannot usually be reversed. Some manufactures supply two sets of jaws, one for internal work and one for external work. Other manufactures make the jaws in two pieces so the outside, or gripping surface may be reversed. which can be interchanged.

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.


UNIVERSAL SCROLL CHUCK



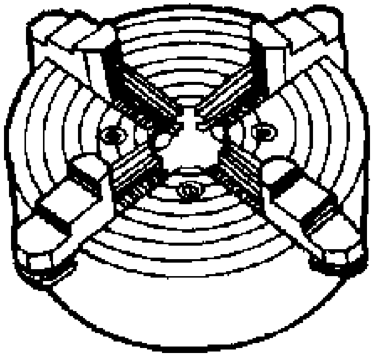
- ***Independent Chuck***

  The independent chuck generally has four jaws which are adjusted individually on the chuck face by means of adjusting screws. The chuck face is scribed with concentric circles which are used for rough alignment of the jaws when chucking round workpieces. The final adjustment is made by turning the workpiece slowly by hand and using a dial indicator to determine it's concentricity. The jaws are then readjusted as necessary to align the workpiece within the desired tolerances.

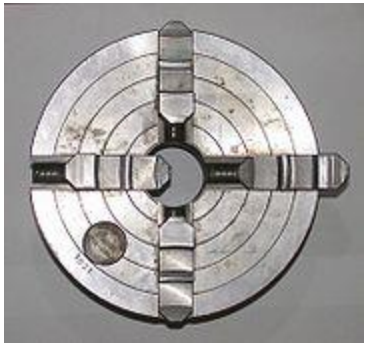
The jaws of the independent chuck may be used as illustrated or may be reversed so that the steps face in the opposite direction; thus workpieces can be gripped either externally or internally. The independent chuck can be used to hold square, round, octagonal, or irregularly shaped workpieces in either a concentric or eccentric position due to the independent operation of each jaw.

Because of its versatility and capacity for fine adjustment, the independent chuck is commonly used for mounting odd-shaped workpieces which must be held with extreme accuracy.

A combination chuck combines the features of the independent chuck and the universal scroll chuck and can have either three or four jaws. The jaws can be moved in unison on a scroll for automatic centering or can be moved individually if desired by separate adjusting screws.



INDEPENDANT CHUCK



- ***Faceplates***

  A lathe faceplate is a flat, round plate that threads to the headstock spindle of the lathe. It 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. Radial T-slots in the faceplate surface facilitate mounting workpieces.

The faceplate is valuable for mounting workpieces in which an eccentric hole or projection is to be machined. The number of applications of the faceplates depends upon the ingenuity of the machinist. A small faceplate known as a driving faceplate is used to drive the lathe dog for workpieces mounted between centers. The driving faceplate usually has fewer T-slots than the larger faceplates. When the workpiece is supported between centers, a lathe dog is fastened to the workpiece and engaged in a slot of the driving faceplate.

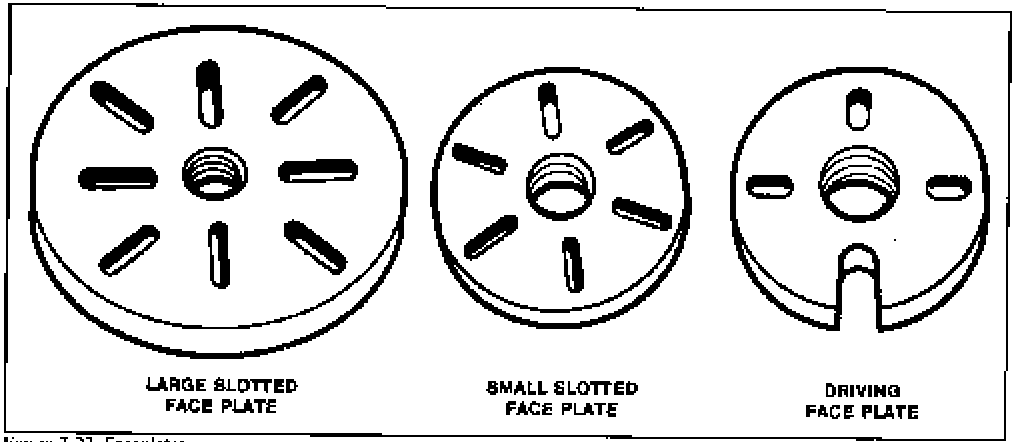
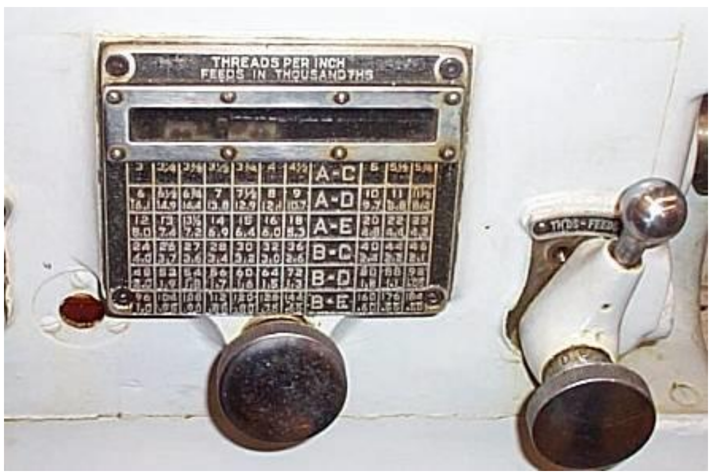
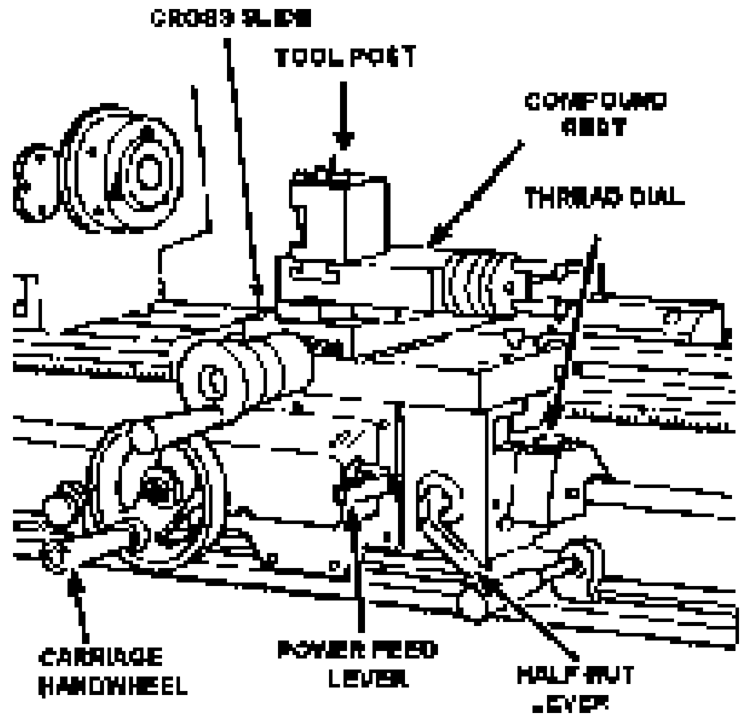


Figure 7-23. Faceplates.

FEEDING AND THREADING MECHANISM

The feed mechanism transmits power through a train of gears to the ***QUICK CHANGE GEAR BOX***, which regulates the amount of tool movement per revolution of the spindle. The feed mechanism also contains gears for reversing tool travel. Lettering on the ***INDEX PLATE*** tells how to position the levers for various thread cutting and feed combinations. The ***LEAD SCREW*** transmits the power to the carriage through a gearing and clutch arrangement in the apron. The ***FEED CHANGE LEVERS*** on the apron control the operation of power feeds and, when placed in neutral, permit half-nuts to be engaged for threading operations.


THREADS PER INCH
FEEDS IN THOUSANDTHS


CROSS SLIDE
TOOL POST
COMPOUND REST
THREAD DIAL
CARRIAGE HANDWHEEL
POWER FEED LEVER
HALF NUT LEVER

SAFE AND CORRECT OPERATION

- ***CARE AND MAINTENANCE OF LATHES***

Lathes are highly accurate machine tools designed to operate around the clock if properly operated and maintained. Lathes must be lubricated and checked for adjustment before operation. Improper lubrication or loose nuts and bolts can cause excessive wear and dangerous operating conditions.

LATHE MACHINE OPERATIONS

The lathe ways are precision ground surfaces and must not be used as tables for other tools and should be kept clean of grit and dirt. The lead screw and gears should be checked frequently for any metal chips that could be lodged in the gearing mechanisms. Check each lathe prior to operation for any missing parts or broken shear pins. Refer to the operator's instructions before attempting to lift any lathe

. Newly installed lathes or lathes that are transported in mobile vehicles should be properly leveled before any operation to prevent vibration and wobble. Any lathes that are transported out of a normal shop environment should be protected from dust, excessive heat, and very cold conditions. Change the lubricant frequently if working in dusty conditions. In hot working areas, use care to avoid overheating the motor or damaging any seals. Operate the lathe at slower speeds than normal when working in cold environments.

PREPARING THE LATHE FOR OPERATION

- ✓ *Clean and lubricate the lathe. Use the lubricants specified by the manufacturer.*
- ✓ *Turn the spindle by hand to make sure it is not locked in back gear. Set the drive mechanism to the desired speed and feed.*
- ✓ *Place all guards in position.*
- ✓ *Move the carriage along the ways; there should be no binding.*
- ✓ *Inspect the cross-feed and compound rest slides. Adjust the gibs if there is too much play.*

- ✓ *Do not permit excessive overhang of the compound rest.*
- ✓ *Inspect the tailstock if it is to be used for any portion of the operation. Check it for alignment and use a smooth dead center.*
- ✓ *Place the proper work holding attachment on the headstock spindle. Clean the threads and apply a drop of oil.*
- ✓ *Sharpen the cutter bit. Clamp it in the appropriate tool holder and mount it in the tool post.*

- ***SAFETY***

  All lathe operators must be constantly aware of the safety hazards that are associated with using the lathe and must know all safety precautions to avoid accidents and injuries. Carelessness and ignorance are two great menaces to personal safety. Other hazards can be mechanically related to working with the lathe, such as proper machine maintenance and setup. Some important safety precautions to follow when using lathes are:

- ✓ *Correct dress is important, remove rings and watches, roll sleeves above elbows.*
- ✓ *Always stop the lathe before making adjustments.*
- ✓ *Do not change spindle speeds until the lathe comes to a complete stop.*
- ✓ *Handle sharp cutters, centers, and drills with care.*
- ✓ *Remove chuck keys and wrenches before operating*
- ✓ *Always wear protective eye protection.*
- ✓ *Handle heavy chucks with care and protect the lathe ways with a block of wood when installing a chuck.*

- ✓ *Know where the emergency stop is before operating the lathe.*
- ✓ *Use pliers or a brush to remove chips and swarf, never your hands.*
- ✓ *Never lean on the lathe.*
- ✓ *Never lay tools directly on the lathe ways. If a separate table is not available, use a wide board with a cleat on each side to lay on the ways.*
- ✓ *Keep tools overhang as short as possible.*

- ✓ *Never attempt to measure work while it is turning.*
- ✓ *Never file lathe work unless the file has a handle.*
- ✓ *File left-handed if possible.*
- ✓ *Protect the lathe ways when grinding or filing.*
- ✓ *Use two hands when sanding the workpiece. Do not wrap sand paper or emory cloth around the workpiece.*

- ***HOW TO CLEAN THE LATHE***

A lathe should be cleaned after each work period. Remove chips with a paint brush – ***NOT YOUR HAND***. Wipe all painted surfaces with a soft cloth. To complete the job, move the tailstock to the extreme right and use a soft cloth to wipe the remaining oil, chips and dirt from the machined surfaces. ***DO NOT USE COMPRESSED AIR TO REMOVE THE CHIPS***. The flying chips are dangerous.

The lead screw needs an occasional cleaning too. This may be done by adjusting the lathe to rotate at a slow speed and using a piece of cord. Permit the cord to feed along the threads. ***DO NOT WRAP THE CORD AROUND YOUR HAND*** because ***THE CORD MIGHT CATCH ON THE LEAD SCREW AND CAUSE SERIOUS INJURY!***

CUTTING TOOLS AND TOOL HOLDERS

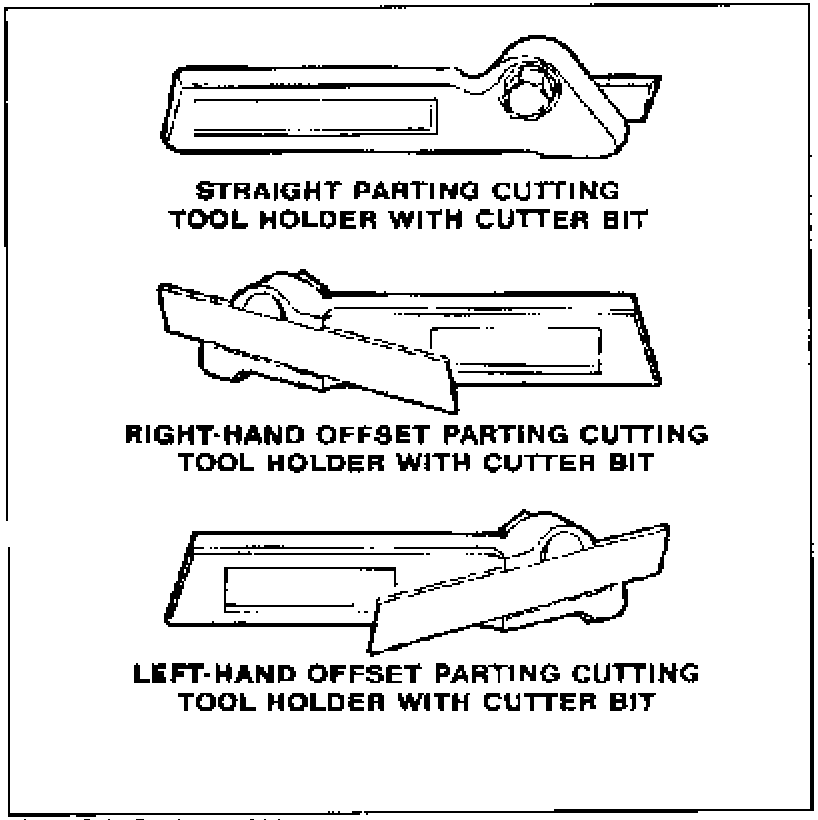


Figure 7-6. Parting tool bits.

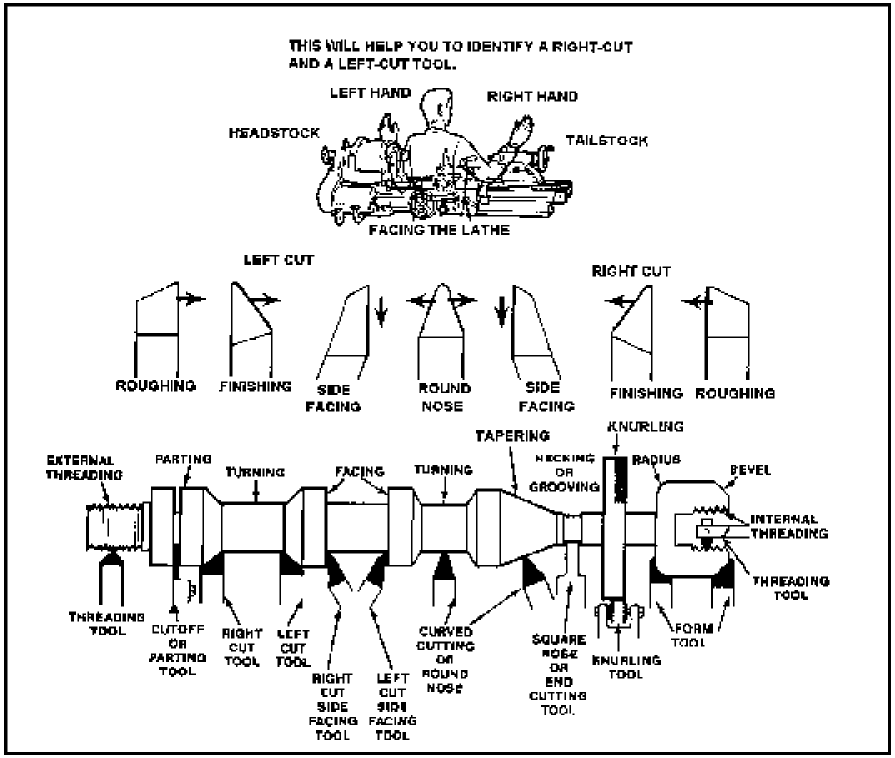


Figure 7-5. Tool bit shapes.

TOOL HOLDERS

Lathe tool holders are designed to securely and rigidly hold the tool bit at a fixed angle for properly machining a workpiece. Tool holders are designed to work in conjunction with various lathe tool posts, onto which the tool holders are mounted. Tool holders for high speed steel tool bits come in various types for different uses. These tool holders are designed to be used with the standard round tool post that usually is supplied with each lathe.

Standard tool holders for high-speed steel cutting tools have a square slot made to fit a standard size tool bit shank. Some standard tool holders for steel tool bits are the ***straight tool holder, right*** and ***left offset tool holder,*** and the ***zero rake tool holder*** designed for special carbide tool bits. 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.***

CUTTING TOOLS / TOOL BITS

The lathe cutting tool or tool bit 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. These tool bits 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 can handle the high heat that is generated during cutting and are not changed after cooling.

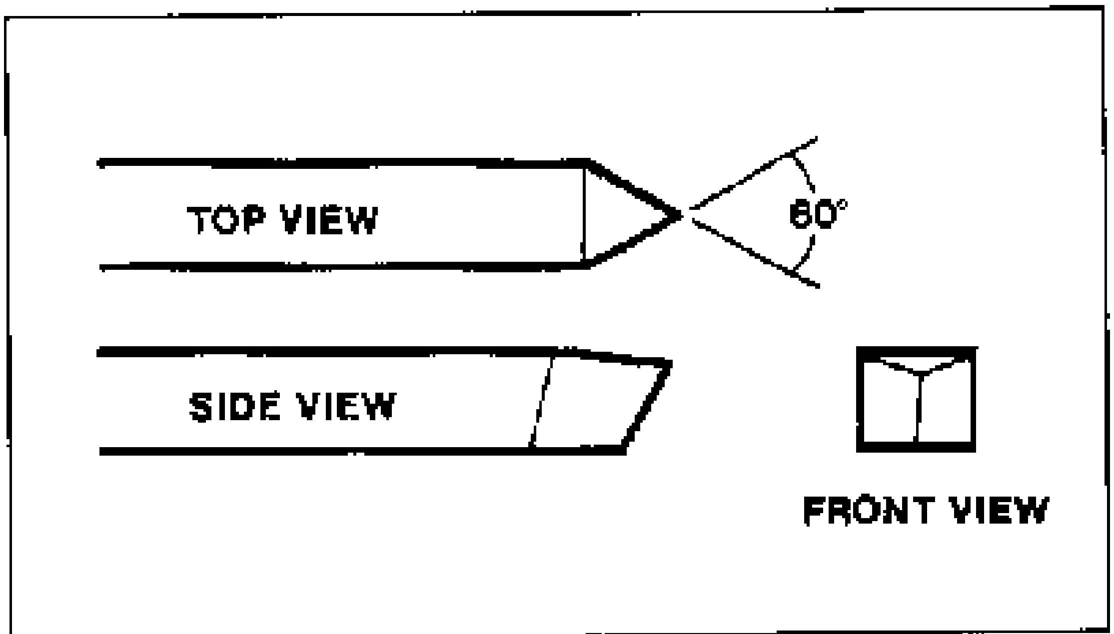


Figure 7-7. Thread cutting tool bit.

Tool bits made from special materials such as carbides, ceramics, diamonds, cast alloys are able to machine workpieces 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. These tool bits are used for turning, facing, boring and other lathe operations.

SINGLE POINT TOOL BITS

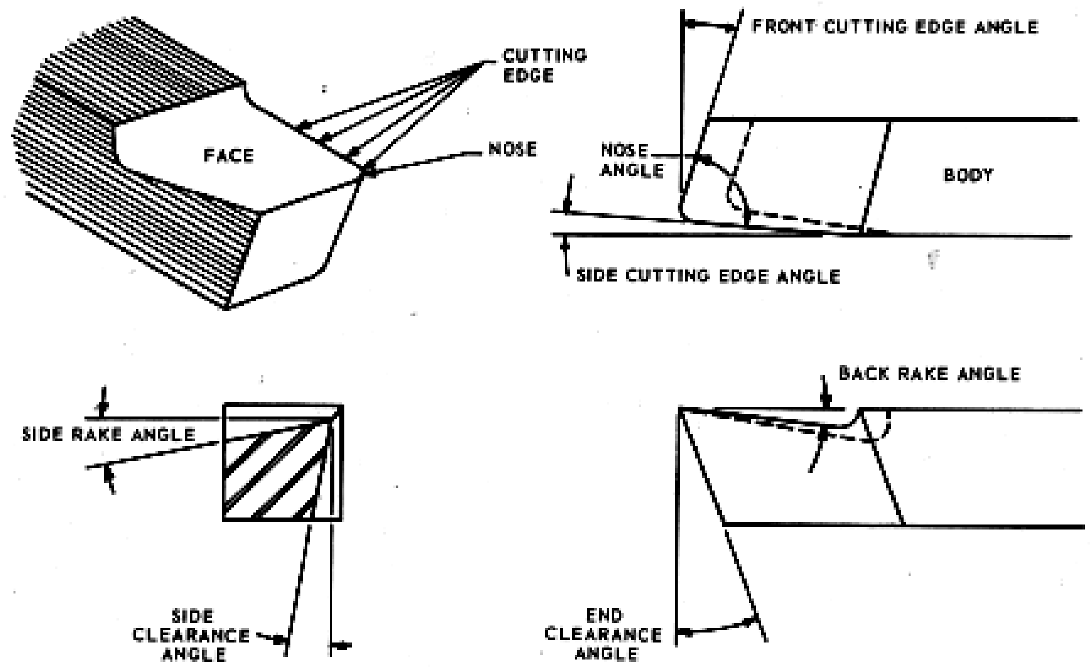
Single point tool bits can be one end of a high-speed steel tool bit or one edge of a carbide or ceramic cutting tool or insert. Basically, a single point cutter bit is a tool that has only one cutting action proceeding at a time. A machinist or machine operator should know the various terms applied to the single point tool bit to properly identify and grind different tool bits

- 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. The nose radius is 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.

ANGLES OF TOOL BITS

The successful operation of the lathe and the quality of work that may be achieved depend largely on the angles that form the cutting edge of the tool bit. Most tools are hand ground to the desired shape on a bench or pedestal grinder. The cutting tool geometry for the rake and relief angles must be properly ground, but the overall shape of the tool bit is determined by the preference of the machinist or machine operator.


CUTTING EDGE
FACE
NOSE
FRONT CUTTING EDGE ANGLE
NOSE ANGLE
BODY
SIDE CUTTING EDGE ANGLE
BACK RAKE ANGLE
SIDE RAKE ANGLE
SIDE CLEARANCE ANGLE
END CLEARANCE ANGLE

Lathe tool bit shapes can be pointed, rounded, squared off, or irregular in shape and still cut quite well as long as the tool bit angles are properly ground for the type of material being machined. The angles are the side and back rake angles, the side and end cutting edge angles, and the side and end relief angles. Other angles to be considered are the radius on the end of the tool bit and the angle of the tool holder. After knowing how the angles affect the cutting action, some recommended cutting tool shapes can be considered.

Rake angle pertains to the top surface of the tool bit. There are two types of rake angles, the side and back rake angles. The rake angle can be positive, negative, or have no rake angle at all. The tool holder can have an angle, known as the tool holder angle, which averages about 15°, depending on the model of tool holder selected. The tool holder angle combines with the back rake angle to provide clearance for the heel of the tool bit from the workpiece and to facilitate chip removal. The side rake angle is measured back from the cutting edge and can be a positive rake angle or have no rake at all.

Rake angles cannot be too great or the cutting edge will lose strength to support the cutting action. The side rake angle determines the type and size of chip produced during the cutting action and the direction that the chip travels when leaving the cutting tool. Chip breakers can be included in the side rake angle to ensure that the chips break up and do not become a safety hazard.

Side and relief angles, or clearance angles, are the angles formed behind and beneath the cutting edge that provide clearance or relief to the cutting action of the tool. There are two types of relief angles, side relief and end relief. Side relief is the angle ground into the tool bit, under the side of the cutting edge, to provide clearance in the direction of tool bit travel. End relief is the angle ground into the tool bit to provide front clearance to keep the tool bit heel from rubbing. The end relief angle is supplemented by the tool holder angle and makes up the effective relief angle for the end of the tool bit.

Side and cutting edge angles are the angles formed by the cutting edge with the end of the tool bit (the end cutting edge angle), or with the side of the tool bit (the side cutting edge angle). The end cutting edge angle permits the nose of the tool bit to make contact with the work and aids in feeding the tool bit into the work. The side cutting edge angle reduces the pressure on the tool bit as it begins to cut. The side rake angle and the side relief angle combine to form the wedge angle (or lip angle) of the tool bit that provides for the cutting action.

A radius ground onto the nose of the tool bit can help strengthen the tool bit and provide for a smooth cutting action.

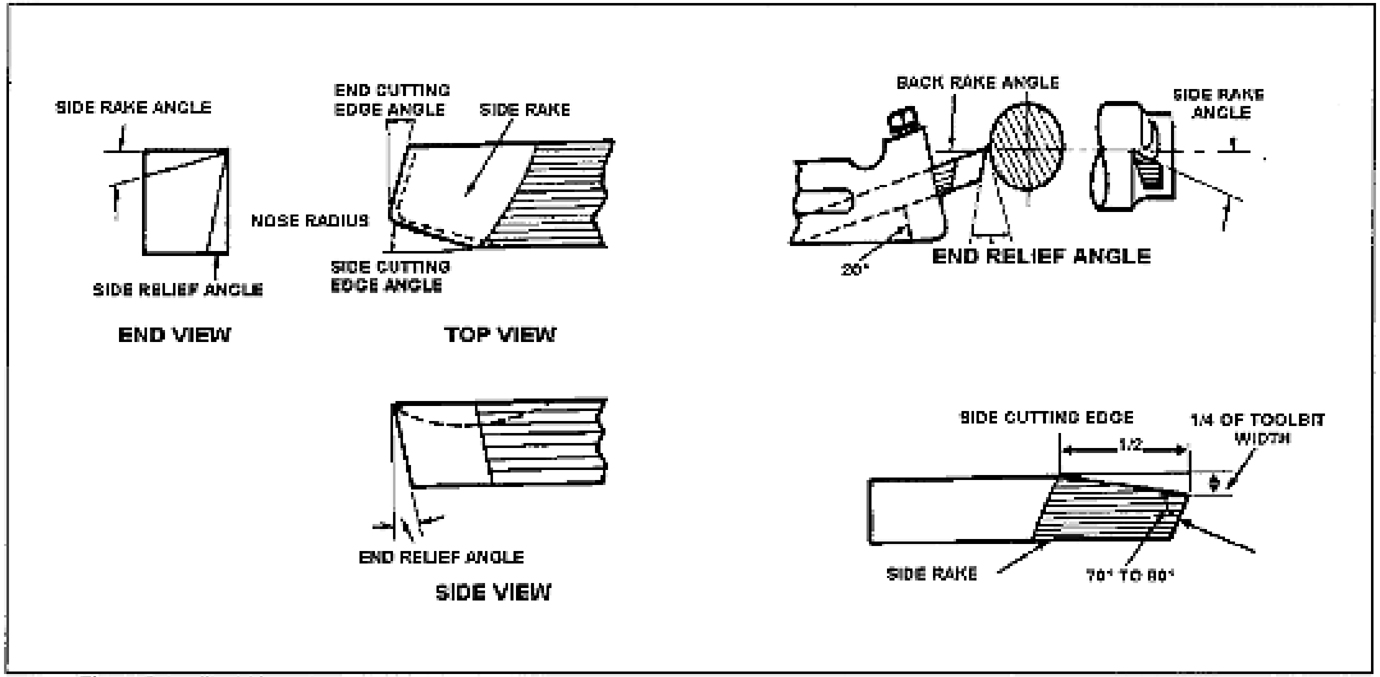
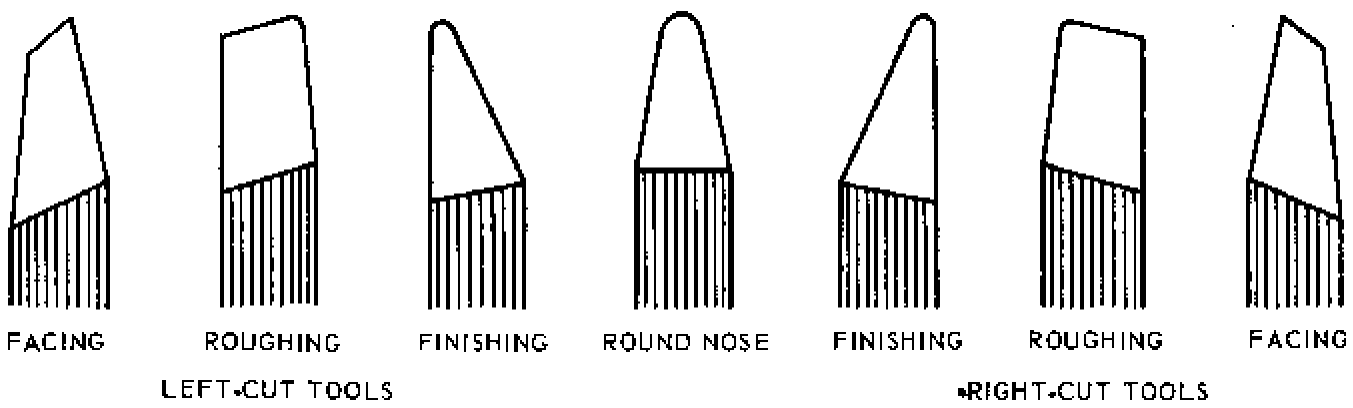


Figure 7-4. Tool bit angles.

SHAPES OF TOOL BITS

The overall shape of the lathe tool bits can be rounded, squared, or another shape as long as the proper angles are included. Tool bits are identified by the function they perform, such as turning or facing. They can also be identified as roughing tools or finishing tools. Generally, a roughing tool has a radius ground onto the nose of the tool bit that is smaller than the radius for a finishing or general-purpose tool bit. Experienced machinists have found the following shapes to be useful for different lathe operations.


FACING
ROUGHING
FINISHING
ROUND NOSE
FINISHING
ROUGHING
FACING
LEFT-CUT TOOLS
RIGHT-CUT TOOLS

A right-hand turning tool bit is shaped to be fed from right to left. 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.

A left-hand turning tool bit is the opposite of the right-hand turning tool bit, designed to cut when fed from left to right. This tool bit is used mainly for machining close in to a right shoulder.

The round-nose turning tool bit is very versatile and can be used to turn in either direction for roughing and finishing cuts. No side rake angle is ground into the top face when used to cut in either direction, but a small back rake angle may be needed for chip removal. The nose radius is usually ground in the shape of a half-circle with a diameter of about 1/32 inch.

The right-hand facing tool bit is intended for facing on right-hand side shoulders and the right end of a workpiece. The cutting edge is on the left-hand side of the bit, and the nose is ground very sharp for machining into a square corner. The direction of feed for this tool bit should be away from the center axis of the work, not going into the center axis.

A left-hand facing tool bit is the opposite of the right-hand facing tool bit and is intend to machine and face the left sides of shoulders.

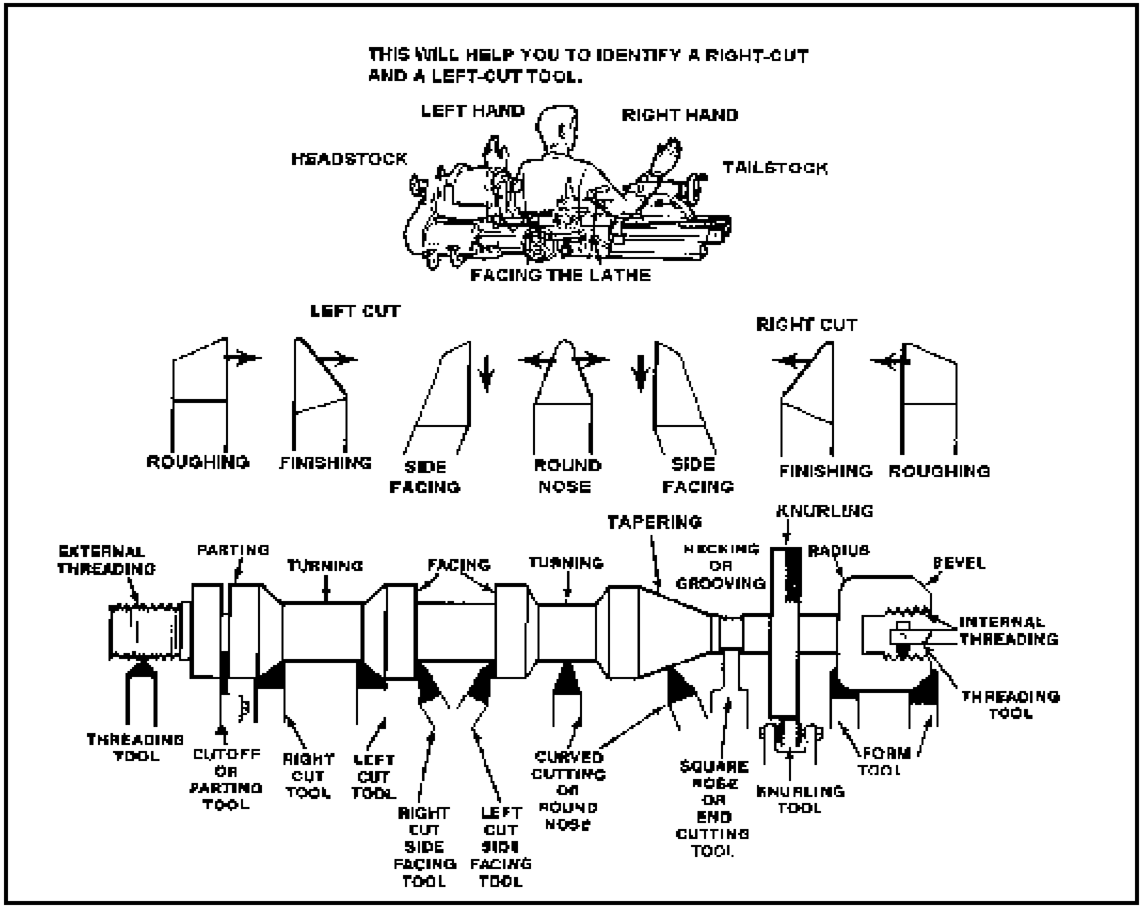


Figure 7-5. Tool bit shapes.

The parting tool bit is also known as the cutoff tool bit. 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.

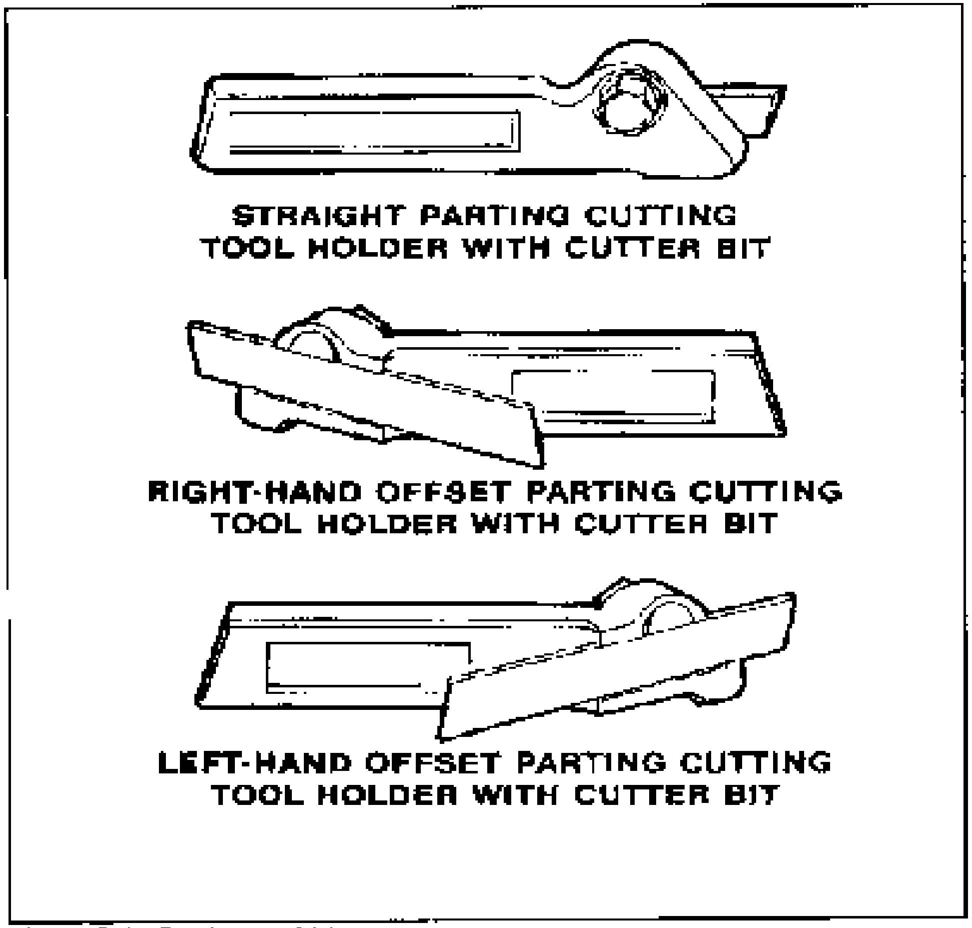
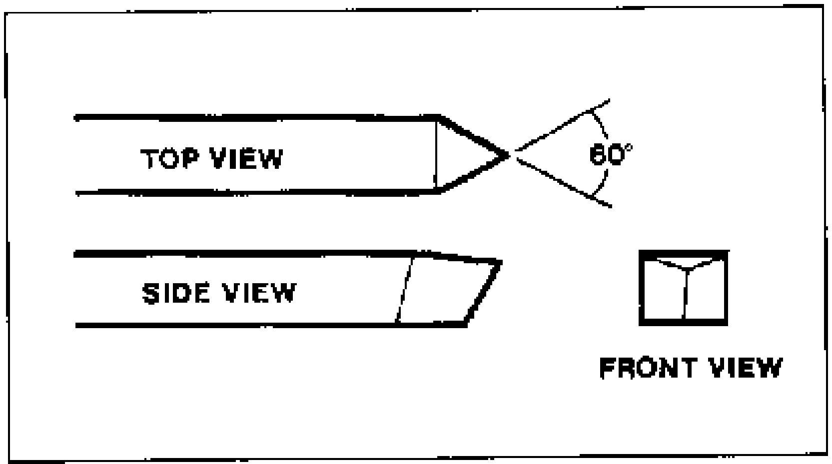


Figure 7-6. Parting tool bits.

Thread-cutting tool bits are ground to cut the type and style of threads desired. 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



igure 7-7. Thread cutting tool bit.

There is no set procedure to grinding lathe tool bit angles and shapes, but there are general guidelines that should be followed. Do not attempt to use the bench or pedestal grinder without becoming fully educated as to its safety, operation, and capabilities. In order to effectively grind a tool bit, the grinding wheel must have a true and clean face and be of the appropriate material for the cutting tool to be ground. Carbide tool bits must be ground on a silicon carbide grinding wheel to remove the very hard metal.

High-speed steel tool bits are the only tool bits that can effectively be ground on the bench or pedestal grinder when equipped with the aluminum oxide grinding wheel which is standard for most field and maintenance shops. Before grinding, shaping, or sharpening a high-speed steel tool bit, inspect the entire grinder for a safe setup and adjust the tool rests and guards as needed for tool bit grinding

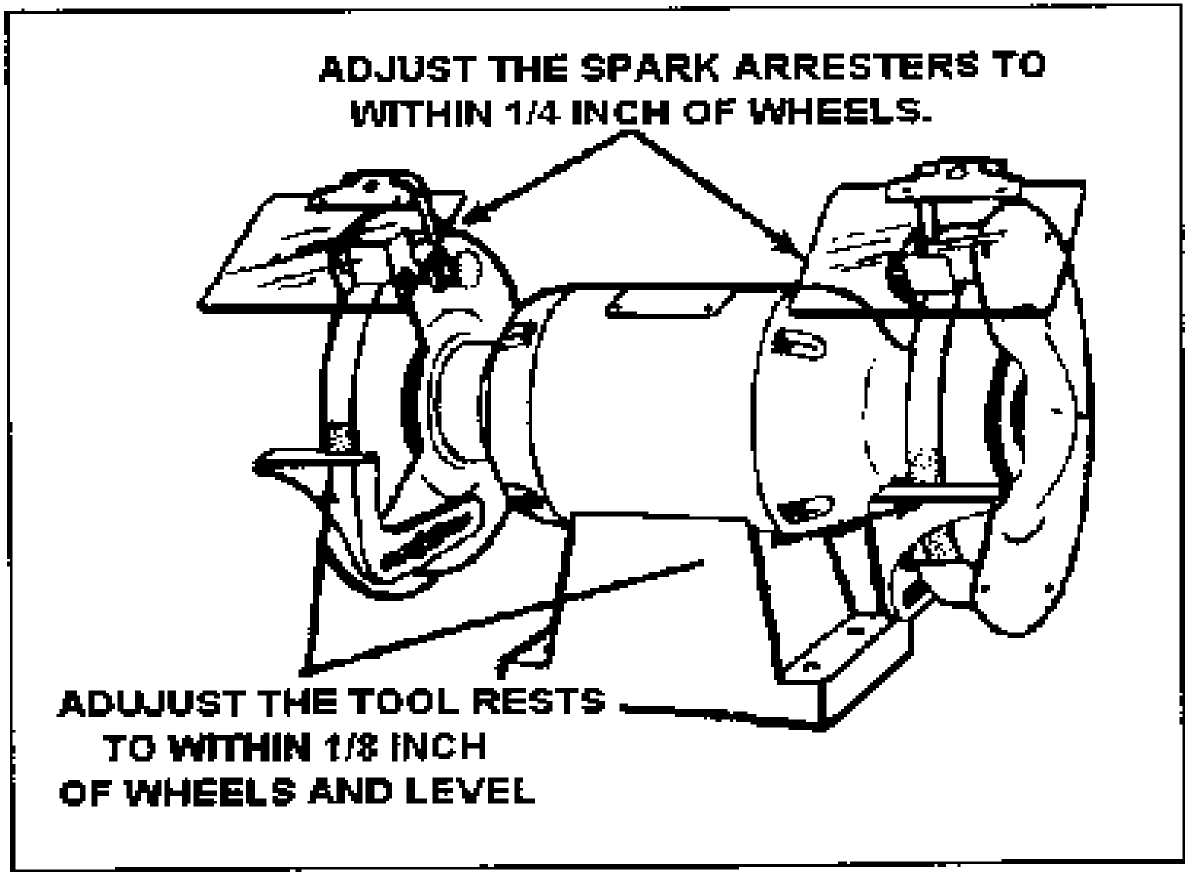


Figure 7-12. Grinder setup for lathe tool bit grinding.

Set the tool rest 1/8 inch or less from the wheel, and adjust the spark arrestor 1/4 inch or less.

Each grinder is usually equipped with a coarse-grained wheel for rough grinding and a fine-grained wheel for fine and finish grinding. Dress the face of the grinding wheels as needed to keep a smooth, flat grinding surface for the tool bit.

When grinding the side and back rake angles, ensure the grinding wheel has a sharp corner for shaping the angle.

Dip the tool bit in water occasionally while grinding to keep the tool bit cool enough to handle and to avoid changing the property of the metal by overheating.

Frequently inspect the tool bit angles with a protractor or special grinding gage.

Grind the tool bit to the recommended angles in the reference for tool bit geometry.

After grinding to the finished shape, the tool bit should be honed lightly on an oilstone to remove any burrs or irregular high spots. The smoother the finish on the cutting tool, the smoother the finish on the work.

Figure shows the steps involved in grinding a round nose tool bit to be used for turning in either direction. As a safety note, never use the side of the grinding wheel to grind a tool bit, as this could weaken the bonding of the wheel and cause it to crack and explode.

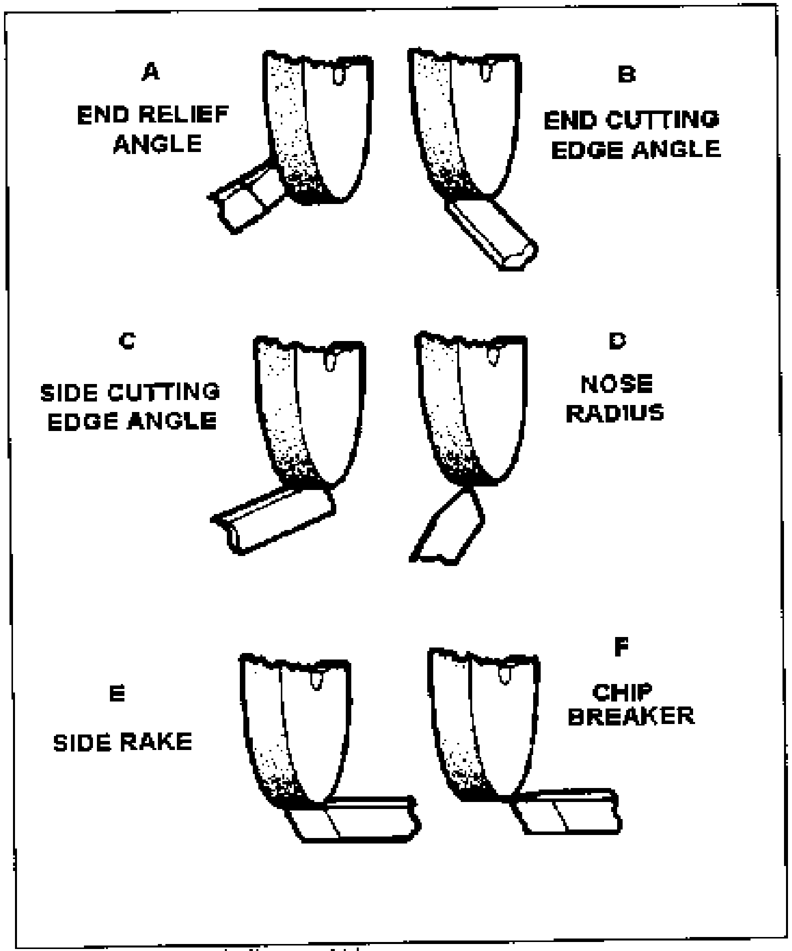


Figure 7-13. Grinding tool bits.

SETTING CUTTING TOOL

- ***Installing a Cutting Tool***

Lathe cutting tools are held by tool holders. To install a tool, first clean the holder then tighten the bolts.

The toolpost is secured to the compound with a T-bolt. The tool holder is secured to the tool post using a quick release lever.

- **Positioning the Tool**

In order to move the cutting tool, the lathe saddle and cross slide can be moved by hand.

A third axis of motion is provided by the compound. The angle of the compound can be adjusted to allow tapers to be cut at any desired angle.

First, loosen the bolts securing the compound to the saddle. Then rotate the compound to the desired angle referencing the dial indicator at the base of the compound. Retighten the bolts. Now the tool can be hand fed along the desired angle. No power feed is available for the compound. If a fine finish is required, use both hands to achieve a smoother feed rate.

The cross slide and compound have a micrometer dial to allow accurate positioning, but the saddle doesn't. To position the saddle accurately, you may use a dial indicator mounted to the saddle. The dial indicator presses against a stop.

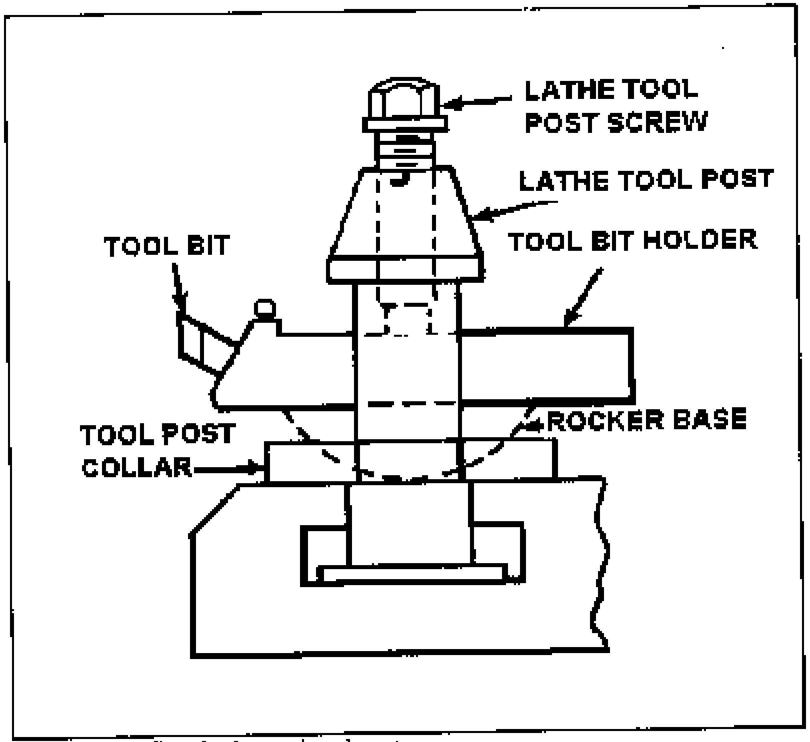
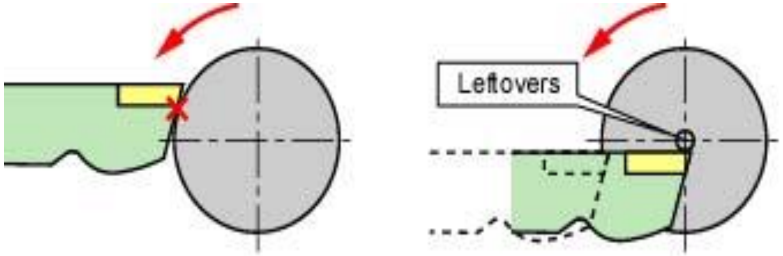


Figure 7-15. Standard round tool post.

CHECKING CUTTING TOOL SETTING

When the cutting tool edge is higher than the center of material, the edge of a blade does not hit the material, and it cannot cut at all. Conversely, if the edge is low, it becomes impossible to cut the center of material. Moreover, the scale of a handle does not have correct value, and then accurate processing becomes impossible.

Though the height of the cutting tool is adjusted in careful, we cannot unite with the center of material completely. Therefore, we have to set the tool to the direction, that the edge is easy to touch the material. The general cutting tool and the parting tool have to be set a few low position. The boring bar has to set a few high position.



(a) Higher Edge (b) Lower Edge

FACTORS AFFECTING CUTTING SPEED

- ✓ material to be cut
- ✓ material of the tool
- ✓ rigidity of the workpiece
- ✓ size and condition of the lathe
- ✓ *depth of cut*.

Cutting Speed for Lathe using High-Speed Tool without Coolant

Turning and Boring			
Material	Cutting		
	Heavy	Finishing	Screw
Low-Carbon Mild Steel	90	100	35
Tool steel, Annealed	50	70	20
Cast Iron, Soft Gray	50	80	25
Brass	150	300	50
Aluminum	200	300	50
Bronze	90	100	25

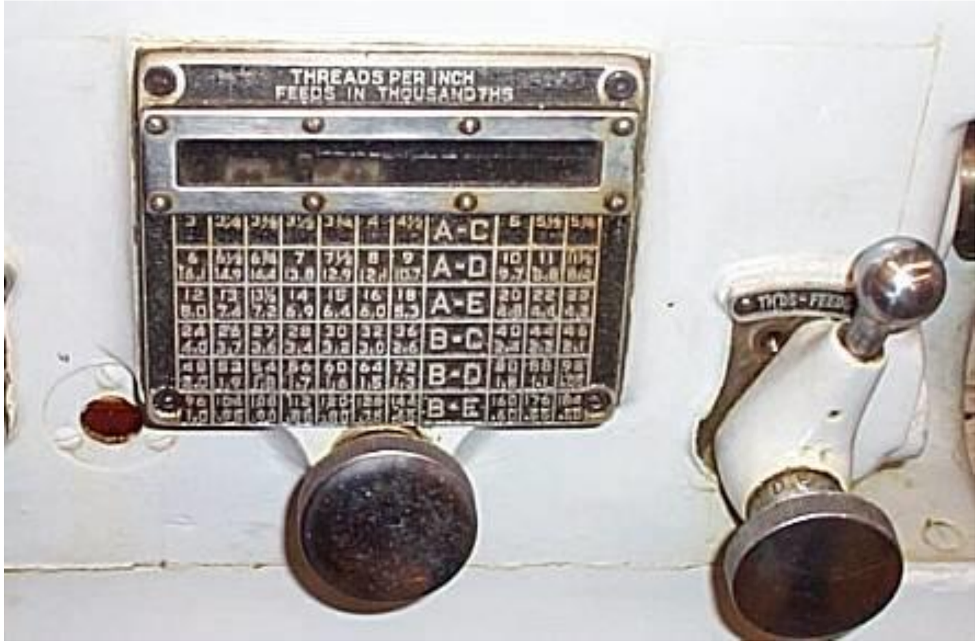
CALCULATING CUTTING SPEED

Formula for finding Cutting Speed :

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

CS	Cutting Speed
π	3.1416
rpm	revolution per minute

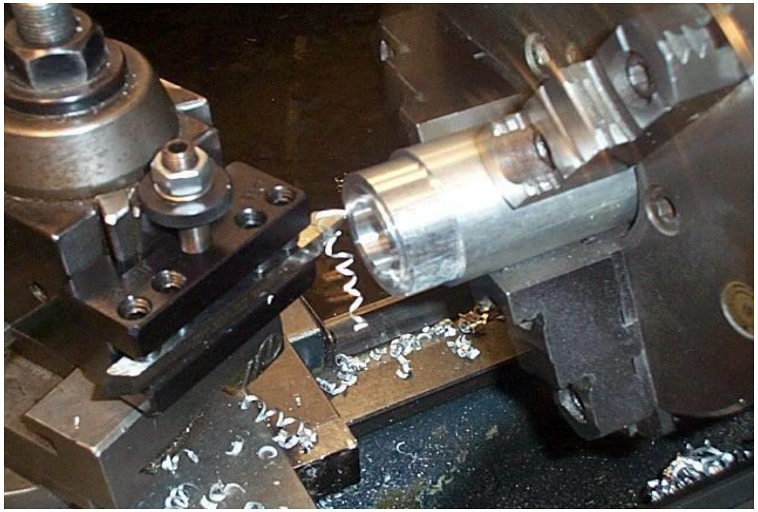
The ***Feed Rate*** (Cutting Speed) of a lathe tool cutting a workpiece is determined by the settings in the ***Quick Change Gearbox***. The gearbox settings are changed through moving either knobs or levers, depending on the particular machine in the Student Shop. Each machine in the shop has a chart on the gearbox, that shows the feed rates for a particular setting of the knobs. The feed rates are expressed in thousandths of an inch per revolution of the spindle.

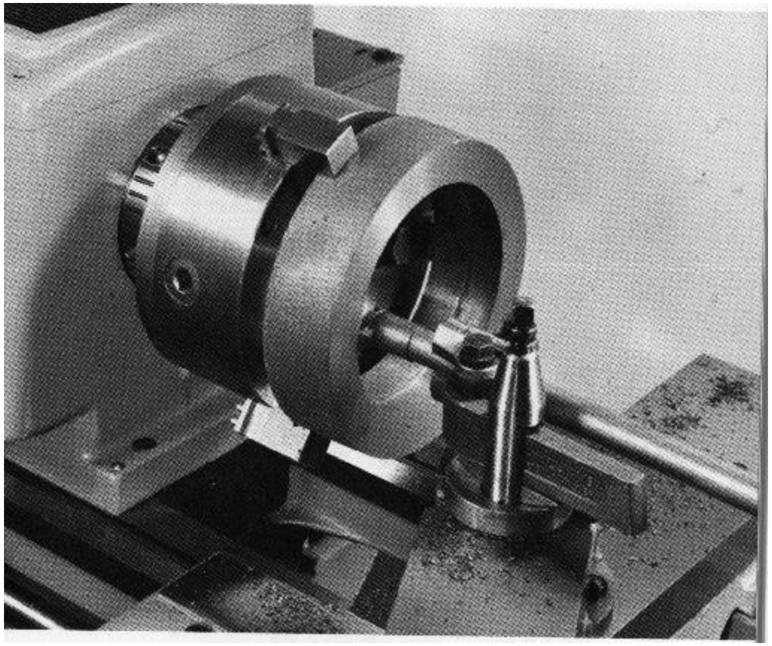

THREADS PER INCH
FEEDS IN THOUSANDTHS
A-C
A-D
A-E
B-C
B-D
B-E

To set feed rates:

- The Spindle must not be rotating while attempting to change gears! Make sure the power is off while carrying out this operation.
- Move one lever or knob at time, making sure that it fully engages into its proper detente or position. If it resists moving into position, it can help to move the spindle by hand, back and forth, while moving the levers.
- check the feed rate by engaging the feed with the spindle running, but the tool not making contact with the workpiece.

MACHINING WORKS





TURNING STOCK

1. Mount and secure the work in the lathe chuck. Check work if centered by using dial indicator or by chalk method.
2. Select cutting tool and install cutting tool in the tool holder with point of the cutting tool projecting about 15mm.
3. Mount cutting tool in the tool post. Adjust cutting tool to the desired position of the point of the cutting tool and then secure.

4. Set spindle speed at low rotating speed. High rotating speed is used for fine finishing of the work.
5. Position cutting tool in reference with desired depth of cut using the cross feed knob. Usually, at the beginning of work, low depth cut is use for prevention from overfeeding.
6. Start the lathe machine. Advance carriage using apron hand wheel cutting about 3mm in the work. Take away point of the cutting tool from the work using apron hand wheel and then stop the lathe machine.

7. Make adjustment according to desired measurement. Start lathe machine and continue advancing little by little to prevent overfeeding until the point of the tool reaches approximately 15mm away from the jaws of the lathe chuck.
8. Remove the point of the cutting tool from the work and stop the lathe machine. Slightly loosen the chuck jaws to release the work.
9. Turnover the work and then mount and secure the work in the lathe chuck making sure work is accurately centered as of the previous turned side of the work.

10. Start lathe machine and continue with the work until unturned side has been done in accordance with the measurement desired. Remove point of the cutting tool and stop lathe machine.
11. Set spindle speed to high rotating speed for fine finishing. Start lathe machine and then advance cutting tool until point of the cutting tool is about 15mm away from the chuck jaws. Remove point of the cutting tool from the work and then the lathe.

12. Slightly release chuck jaws to remove the work. Turnover the work to the unfinished side of the work. Use copper strip between chuck jaws and finished side of the work preventing damage from the finished side.
13. Mount and secure the work to the lathe chuck taking measure of centering the work as of the previous operation.
14. Complete the fine finish of the work.
15. Stop the lathe machine when work is finished and then clean and secure the lathe machine.

TURNING TAPER

- ***SET OVER METHOD***

1. Find the center of the stock and prepare it for mounting between centers.
2. Turn the stock to desired cylinder size and then remove the stock from between centers.
3. Calculate the amount of set over for the tailstock.

4. Loosen both the clamping nut and the tailstock. This screw should be drawn out the distance required such as ¼”. Then, screw in the tailstock set over screw on the opposite side of the tailstock until the tailstock is set off center to the distance desired. This can be checked by means of witness mark on the base of the tailstock.
5. Fasten the tailstock in the position by tightening the clamping nut. The amount of the tailstock that has been set over may be tested by moving the tailstock close to the headstock and measure with a rule the difference in alignment of the points of the two centers.

6. Mount the stock between centers. Be sure to apply center lubricant on the dead center.
7. Select cutting tool, mount in the tool holder with point of tool just even with a line through the center of the work.
8. With the apron hand wheel, draw the tool forward until it is in position to make a cut. Then with the cross feed knob, advance the tool so that it will take a fairly heavy cut.
9. Start the lathe and feed the tool towards the left in the usual manner.

10. Continue advancing the tool with the hand wheel until cut runs out. Withdraw the cutting tool and move the carriage back to the starting point.
11. Measure the length on the taper for example, 1” or 2”. At this point, measure the large and the small end of the taper. Calculate the amount of taper in the unit of length. Compare this with the amount of taper required.
12. Make adjustment in taper if necessary.
13. With repeated cuts, reduce the stock to approximately the size desired.

14. Set the caliper to the dimension then with a light finishing cut; reduce the stock to the size desired.
15. When the job has been completed, set the tailstock back in normal position.

- ***TAPER ATTACHMENT METHOD***

1. Disengaged the cross feed screw, removing the screw that holds the cross feed control nut on the saddle.

2. Attach the connecting slide arm to the cross feed by tightening taper attachment clamp.
3. Engage the taper attachment on the ways and fasten it with the set screws.
4. Set the taper slide bar at the angle desired and clamp it in position with the screw nut.
5. Make the cut as ordinary turning

CUTTING THREADS

1. Determine the number of threads to be cut per inch.
2. Adjust the quick change gear to obtain the ratio desired.
3. Mount the work on the lathe chuck. Be sure work is centered and secured at the lathe chuck.

   If thread will end in the same intermediate point on the surface of a straight cylinder, put a pencil line at the point desired. If permissible, cut a neck or at the thread to aid in extracting the tool at the proper point.

4. Set the compound rest at 29 degrees.

   Accurate adjustment of the threading tool is easier if it is needed to remove the tool before the cutting of thread is completed. It also makes possible removal of most of the stock from one side of the thread only. Furthermore, for better finish, the tool should drag slightly on the right hand side of the thread.

5. Select a sharp and correct ground threading tool. The inclusive angle of the point of the tool should be 60 degrees, with each cutting edge having the same degree of angularity. An end relief and a side relief of 8 to 10 degrees should be provided on both sides. No side rake is required.

6. Insert the tool in a tool holder with only the ground portions of the bit projecting beyond the nose of the holder.
7. Set the point of the threading tool even with the center of the work.
8. Hold the center gauge against the side of the work then advance the threading tool and set it square with the work. Tighten the set screw which holds the tool post and again test it if it perpendicular with the work.
9. Test the height at which the point of the tool is set.

10. Advance the threading tool until the point touches the work, then adjust the micrometer collar on the cross-feed screw to zero.
11. Draw the tool back pass the right end of the work and then advance the point of the tool about 0.002 by turning the compound rest screw towards right. Adjustment of the depth of cut is usually made by advancing the compound rest, thereby causing the tool to cut on one side of the thread groove. However, if desired, the depth of cut may be adjusted by means of the cross feed thereby causing the tool to cut on both sides of the thread groove.

12. Start the lathe machine. Engage half-nut. The use of the thread dial indicator makes it unnecessary to reverse the lathe spindle to return the carriage to the starting point to begin each successive cut. The dial is graduated and numbered to indicate when the half-nut should be closed on the feed screw.

    For even numbered thread, engage the half-nut when the datum line coincides with any line on the dial.

    For odd numbered thread, engage the half-nut on any odd numbered line.

- When the threading tool reaches the terminus of the thread, quick draw the tool away from the work by giving the cross-feed screw one complete revolution towards the left and at the same tome, immediately disengages the half-nut.
- Draw the carriage back to the starting point with the apron hand wheel, then advance the tool slightly by turning the compound rest screw so as to advance the tool to about 0.002.
- Turn the cross-feed screw one complete revolution towards the right, and then engage the half-nut. Withdraw the tool at the terminus of the thread by giving the cross-feed screw one complete revolution towards left. Stop on the cross-feed saves time.

- END -