Ed's

NOTE BOOK

MACHINING

2006

MACHINE SHOP NOTE BOOK

By

Edouard Forcier, III

INTRODUCTION

The Notebook's objective is to provide the student with basic knowledge regarding fundamental machine tool operations. This Notebook is not meant to replace a machine theory book, but only to serve as an introduction.

Basically, the Notebook covers basic blueprint reading terminology and application, measuring, lathe and milling machine work, surface grinding, heat treating and speeds and feeds.

Included are a few simple and practical projects that follow the process. After many years of trial and error, the hammer project is used as the basis of this Notebook. When machining the hammer, the student will find that the Notebook follows the machining process. In the end, the student will have a practical and useful tool that was made by him or herself..

DRAWING or BLUEPRINT READING

It is said that a drawing, a.k.a. as a blueprint, is putting ideas down on paper so they can be further explored. However, it is still and idea. The shop make these ideas into a tangible product hat can be touched, measured, examined and tested. The drawing is the vehicle where the idea is transferred into a tangible product. Drawings are made to American National Standards Institute (ANSI) standards. Because of the standard, drawings are made to rules and thus are its own "language". It is the universal language of engineers, draftsmen, machinists, welders, inspectors and any other individual involved in manufacturing.

There are different types of drawings a terms that you need to know. Here are some of them.

Assembly Drawing

Show how all the parts of a component fit with each other. Let's you visualize where components go.

Detail drawing

A drawing of a single part showing, shape, size all dimensions, tolerances, material used and any specific process necessary, such as heat treating.

Sectional Drawing or a View

A "cut out of the outside" of a complex part to show the "inside". Important if certain features or shapes are necessary to achieve.

Drawings are made of lines that signify many things. It is important that these explained. Standardized, they have a specific meaning and use.

Object

Solid lines that define the object's shape or feature.

Hidden

Dashed lines that represent shape that can't be seen because they are hidden by a view. They unseen surfaces must be represented.

Center

A dashed line followed by a space, then by a long line pattern to show the center of circles.

Dimension

A line with arrowheads at each end. The arrows indicate how long a feature is.

Extension

A line leading from the objects end to its other end. Between it is the dimension line showing the features size.

A couple of other words.

Feature

A feature is a diameter, hole, length, slot, rectangle, shape or the like.

Views

A drawing has several views attached to it. Basically, as you would unfold a toothpaste box, you'd see that these sides are attached, as they are when the box is folded. A drawing is exactly the same. As the views unfold, they unfold to the respective position with the object. The three major views are the FRONT, RIGHT and TOP views. The tough part of this is to "see" in your head how the views all fit together so you can "visualize" a three dimensional picture of the object in your brain.

FITS, DIMENSIONS and TIOLERANCES

Manufactured parts are most always used in conjunction with other parts. In order for the parts to function as designed, they must assemble in a certain fashion. This is known as "fit". In order for parts to assemble, they must be made to a certain size or dimension. It is impossible to make parts exact and perfect consistently, size variations will occur. How much the size can vary and still meet the functional requirements, is known as "tolerances".

Fit determines how parts will assemble together such as a nut sliding onto a bolt, a .5mm lead sliding into your mechanical pencil or how the knobs slide onto your stereo system.

Basically, there are four types of fits. They are;

Sliding
These allow two parts to slide by each other without binding. Putting a key into a lock is an example.

Running
These are close fitting "sliding" fits such as those found in crankshaft or ball bearing assemblies.

Force
Force fits occur when one part is pressed intro another and they won't come apart.

Shrink
Occurs when one part is heated (or frozen), it expands (or shrinks), and is assembled with its mate. As the part cools (or warms) to room temperature, it contracts (or expands) to its mate and forms a very strong assembly.

Refer to the manufacturer's bible, the latest edition of the Machinery's Handbook regarding fits or any other technical information you need. It's a must.

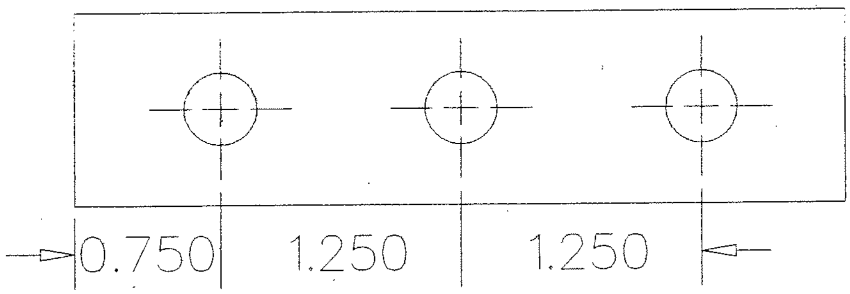
Dimensions are numbers that show how long, how wide, how thick or what a hole diameter is. These are known as "size" dimensions. Dimensions that show how far a hole's center is from each edge is known as a "location" dimension.

In American manufacture, dimensions are expressed in inches only. In Metric manufacture, dimensions are expressed in millimeters (mm's) only. Partial inches are used and are written as 1/64, 1/32, 1/16 and 1/8. Note, that the inch symbol, ", is never recorded with the value. A length such a 1' 6 ½" is represented as 18.5", 18.50" or 18.500". Notice the dimension has up to three decimal values. They all measure the same length, but the number of decimal places is significant as it represents how "close" the dimension must be made to (tolerance). More on that later. We could just have use 18 ½", but fractions are generally never used with the main reason being is that precision measuring tools measure in decimal equivalents.

INCREMENTAL and BASELINE DIMENSIONING

Incremental

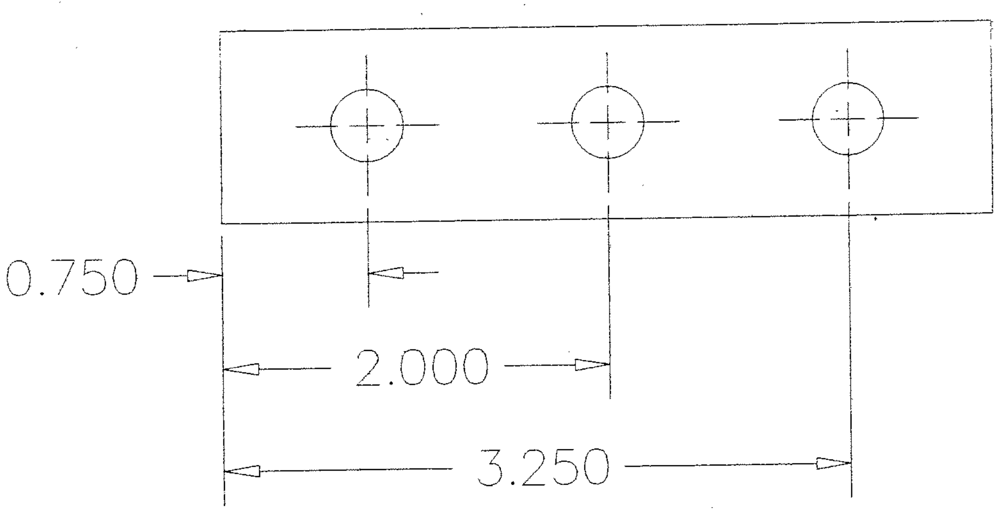
There are basically two types of drawing dimension styles. The first is Incremental or Chain dimensioning. In this system, the location or distance of the second dimension or feature is dependent upon the actual location of the first feature.



Each dimension has a tolerance applied to it. Theoretically speaking, the sum total of all the plus tolerances could make the length exceed the overall length of the part. However, modern manufacturing methods and machine tools control these variables to keep the part within design intentions.

Baseline

The second method is absolute or baseline dimensioning. In this system, all dimensions begin at a common feature, known as a datum. Using a baseline or datum to locate from, dimensions are individually controlled and their true position is not affected by the position of the previous feature. This helps to make precise parts more easily.



TOLERANCES

Tolerances are the amount a dimension can be permitted to deviate or vary from its basic size. Let's examine how tolerances are arrived at.

The design requires that a shaft with a diameter of 2.250" fit into a hole with a clearance of.001". The designer arrives at this .001" number from a number of places, generally from fits and allowance tables. See the Machinery's Handbook for an explanation. The .001" clearance occurs when the shaft is at 2.250" and the hole is at 2.251". This is known as Maximum Material Condition. The designer wants a maximum clearance of .005" between both parts at worst case. This occurs when the shaft is at its smallest diameter and the hole is at its biggest size. This condition is known as Least Material Condition. Subtract .001 from .005 and your tolerance is .004". According to the tables, the 004" tolerance could be divided 50-50 between the shaft and hole or 60-40, 30-70 or any combination the designer needs equaling .004". The designer selects the following;

Hole diameter	2.251	to	2.253	equals	.002" tolerance
Shaft Diameter	2.250	to	2.248	equals	.002" tolerance
Min clr	. 001	Max clr	.005		.004" tolerance

Fortunately for the machininst, all we care about is what the numbers are and how to machine the part to those values.

Tolerances are expressed in many different styles. These include,

Bi-lateral — Equal tolerances expressed from the basic such as 2.252 +/-.001.

Unilateral — Expressed in either the + or – direction from the basic 2.250 $^{+.002}_{-.000}$ or 2.249 $^{+.000}_{-.002}$

Limits — Expressed as a maximum over a minimum tolerance

$$\frac{2.253}{2.251} \qquad \frac{2.250}{2.248}$$

Baseline — An NIS (Nippon Industrial Standard, a Japanese system) from which the size is taken from a basic size, such as 2.250.

2.250 +.001, +.003 (2.251 to 2.253)
2.250 -.000, -.002 (2.250 to 2.248)

A few definitions.

Tolerance is the allowable variation is size. This is needed because dimensions cannot be consistently machined to an exact and precise consistently. This occurs because of tool wear, machine wear, temperature variations and the like.

Allowance is the intentional clearance in size between two mating parts so they can fit together. In the previous example, it is .001".

Fit or clearance is the real space between mating parts. Using the previous example, it could be small as .001" to as much a .004", or anywhere in between. The fit could be negative for press or interference fits.

Enough of theory.

BLUEPRINT (or Print) READING

The blueprint, drawing or print is the absolute or final authority in manufacturing. It must contain all of the necessary information to manufacture this part to the standard. There is a certain procedure in reading a print. Prints follow a certain format. Let's examine the Hammer print.

Title Block	Contains the company name, part name, part material, designer, date of design, tolerances, special processes and parts number.
Revisions	A block that identifies, lists and details made to the part.
Notes	Records processes or items needing caution or special attention.
Tolerance Block	Lists tolerances for dimensions not having a tolerance attached to it. This is known as a "general tolerance". All dimensions must have a tolerance attached to it somehow.

Let's look and analyze the Hammer print.

In looking at the Hammer print, we find;

1. The part is only one view.
2. The material is mild steel (1018), ¾" diameter and easy to machine.
3. Most dimensions are two-place decimals, thus a +/-.010" tolerance, except as noted.
4. We have to turn, groove, knurl and thread.
5. The part is small enough to hold in your hand. We don't need a crane to pick it up.

Looking at the Head print, we see;

1. We have a top and front view.
2. Options on the head style.
3. A ¾" square tool steel material (4140) that needs to be heat-treated.
4. A drilled and tapped drill hole that is also counter drilled, and a drilled pin hole.
5. Notes needing our atteention.
6. It is a small and lightweight part.

A few print procedures. The main thing in any craft endeavor is to be consistent in methods of work. If you are consistent, making things will come easier to you. In print reading here are some guidelines.

1. Scan the drawing to get an overall feel of what's on it.
2. Look at the title block. Pick out the company, material, material size, part number and such.
3. Look at all of the views
4. Start focussing on details, such as features, shapes, hidden views, and details like that.
5. Focus on size details. Do you need a crane to move this around or can you a hold thousand or so in your hands? (Did you ever notice the screws in your eyeglasses? Somebody has to make those and you can hold a few thousand in you hand!)
6. Start to form a 3-D picture of the object in your mind. "Rotate" it, go "over" or "under" it so your can clearly "see" it in your head.
7. When a machinist studies the print, his/her mind starts to make a list of machining operations, such as facing, turning, boring, threading and most importantly, work holding.
8. Focus on dimensions. Certain tolerances will dictate machining operations such as drilling only versus drilling and reaming

Once the print is thoroughly understood can the machining process begin.

Methods and Routings

Methods

Manufacturing requires a precise and careful control of the process to promote efficiency, costs and quality. This is important. Remember, in the shop, the machinist has only ONE Chance to make the part right. If the print is wrong, out comes the eraser and the problem is fixed. If the machinist errs, the part, worth from a few pennies to thousands of dollars, is SCRAP. Useless. No value. JUNK.

Large companies have manufacturing engineering departments to break down the process, arrange them in a logical and efficient order and devise time values, determine proper machines to machine the part etc. This is needed to verify that all print requirements and standards are met.

Routings

Routing sheets provide a narrative description of the process. Each machine process is described, sequentially listed, has an estimated time value for set up and cycle time and a cost value. The routing for your hammer is typical of the process.

The routing sheet contains the part name, part number, machine or product it is used on. There are few significant things about all this paperwork.

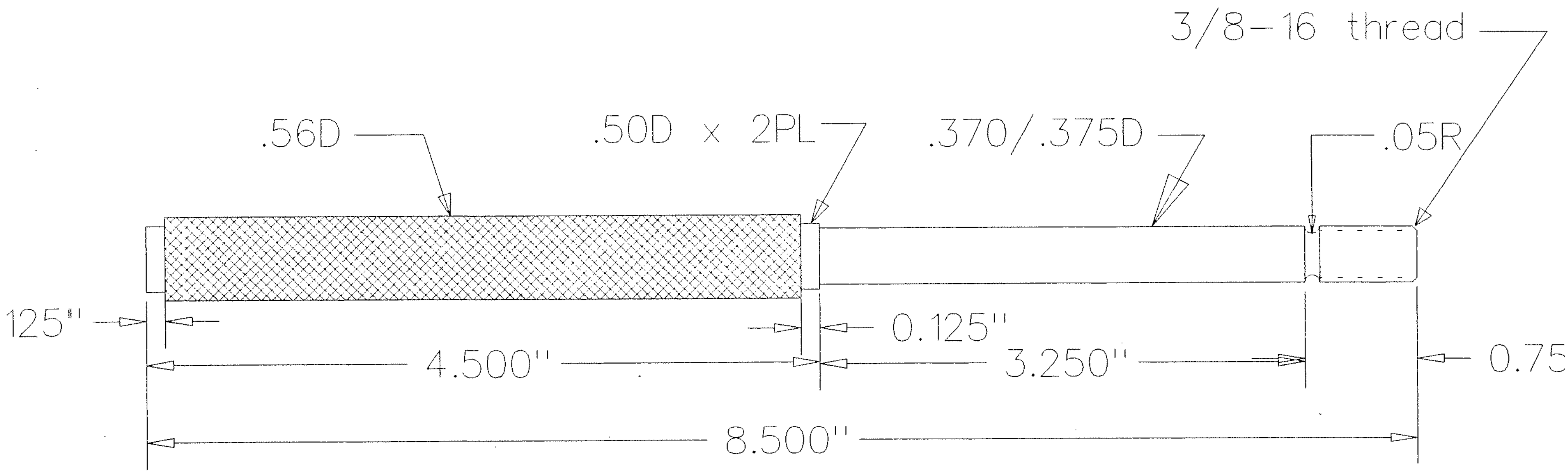
1. Each part manufactured has a detail drawing.
2. Each part made has a routing sheet.
3. Each consists of one or many elements.
4. Documentation helps to control costs, improve efficiency and improves shop floor management.
5. Traceability. A paperwork trail shows who, what, when, how the part was manufactured and processed. From its raw material source to buyer, the paperwork trail is necessary. If the product, such as mayonnaise, were defective, you'd want to make sure the company knows which lots are bad, notify everyone so that the product can be removed from the public. You'd want to make sure they changed that engine in the 737 you're a passenger in. A couple examples of everyday situations.

The routing sheet that follows is for the hammer and the head assembly. It contains operations in their correct sequence and gives a description of each operation. Note, if you were making these by the thousands or if you had different machinery, your processes would be different.

Finally, some advice from carpenters of old:

1. Think thrice,
2. Measure twice, and
3. Cut once.

Have fun and enjoy this activity.

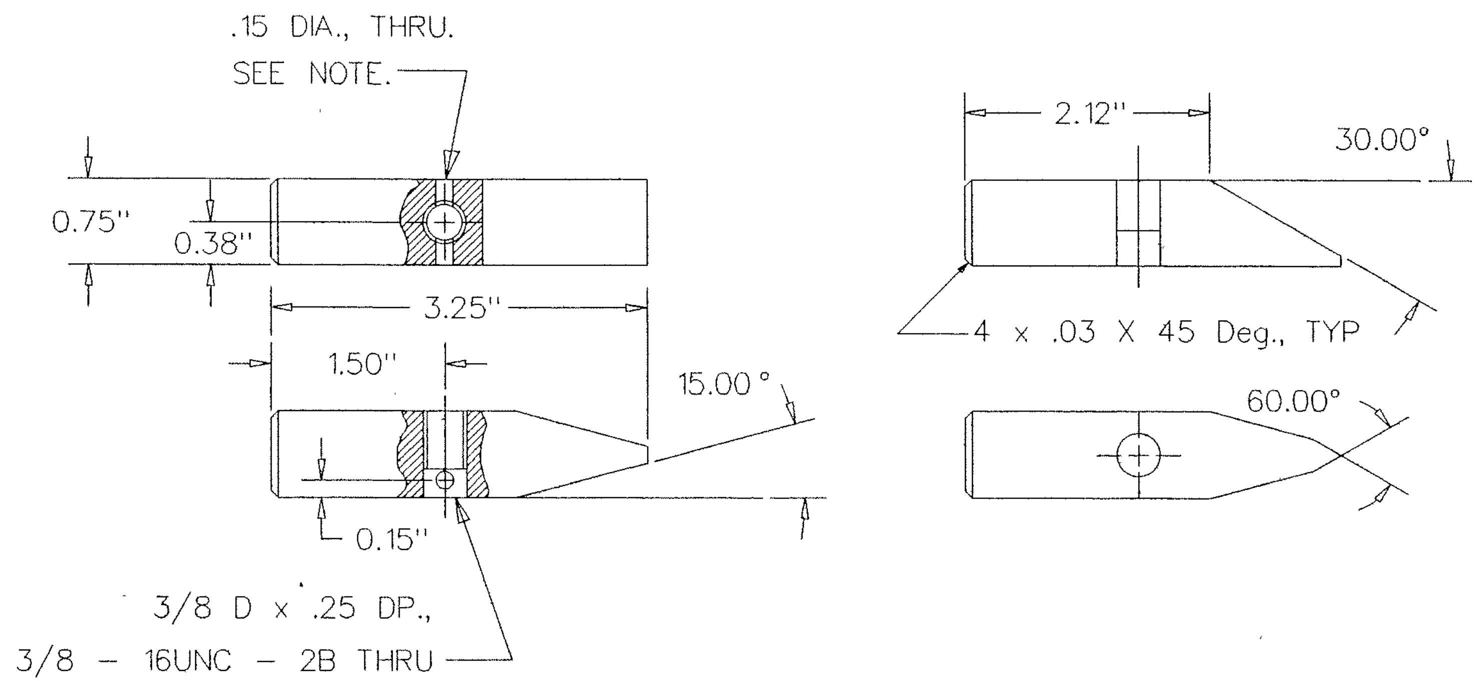


NOTES:

MEDIUM KNURL

3/4 DIA. 1018 CRS

BREAK ALL SHARP EDGES



NOTES

DIMENSIONS APPLY TO ALL HEAD STYLES.

HARDEN TO 50 - 55 Rc

GRIND SIDES TO .74/.73 DIMENSION

DRILL HEAD PIN HOLE PRIOR TO HARDENING.

ASSEMBLE HANDLE TO HEAD, DRILL THRU AND PIN.

MACHINE EXCESS HEAD LENGTH.

TOLERANCES	elfTECH
.XX +\- .010	MAT''L: 3/4 SQ. 4140
.XXX +/- .003	NAME: HAMMER HEAD
ANGLES +/- 1/2 deg	DATE07/03 NUMBER H-2

ROUTING SHEET – HAMMER

Hammer Handle

Oper #	Description	Machine
10	Cut ¾ Dia. 1018CRS stock to 8 5/8" long.	Cut Saw
20	Chuck. Face one end to clean and center drill. Reverse ends and repeat.	Lathe/3 or 4-jaw chuck
30	Chuck. Expose about 5 ¼". Position center. Turn .56D about 5" long. Knurl. Turn .500D x .12 Long. Break sharp edges.	Lathe/ 3 or 4 jaw
40	Mark 4.500 length to 4.38 for turning.	
50	Chuck. Position center. Turn .500 D to 4.38 mark. Turn Turn .375D holding .12 and 4.50 Dims. Undercut. Cut 3/8 – 16UNC thread.	Lathe/3 or 4 jaw

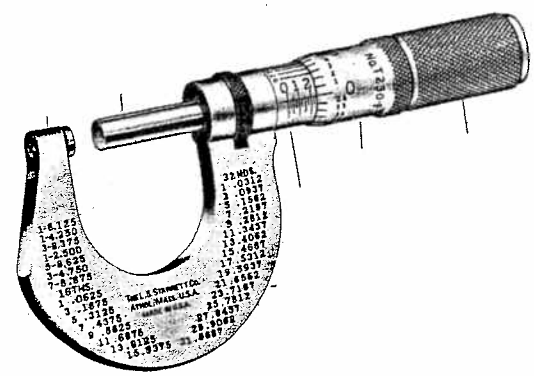
Hammer Head

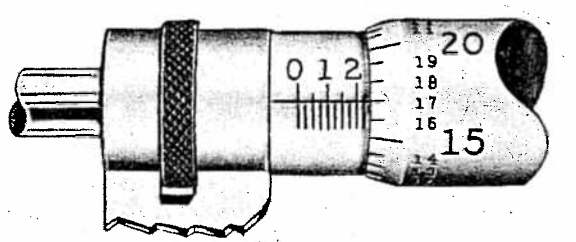
10	Cut 3/4Sq. 4140 stock to 3 5/16 long.	Cut Saw
20	Mill one end clean and square.	Bridgeport/vise
30	Reverse. Mill end clean, square and 3.25 long.	Bridgeport/vise
40	Mill .03 x 45 chamfer on end, four edges.	Bridgeport/fixture/vise
50	Mill head angle and style.	Bridgeport/vise
60	Locate. Center drill, drill 3/8D x .25 deep. Drill 5/16D hole thru. Tap 3/8-16 thru.	Bridgeport/vise
70	Layout and drill .15D pinhole.	Drill press/vise
80	Harden and temper.	Heat-treat furnace.
90	Grind sides clean holding .74 / .73 dimension.	Surface grinder

Assembly

10	Clean threads with an old tap.	Bench
20	Assemble handle to head as tight as possible.	Bench/vise
30	Drill .156D thru using head hole as a guide.	Drill press/vise
40	Pin the assembly.	Bench/arbor press
50	Chuck the handle into the lathe so the head is against the chuck jaw. Face off extra material until the handle is flush with the head.	Lathe/chuck
60	Inspect	Inspection

BASIC MEASURING





BASIC MEASURING

In manufacturing, the ability to make parts that fit precisely, function and endure as designed requires the ability, skill and proper use of measuring tools. Tools range from the simple, a scale (ruler), micrometers and verniers to digital micrometers and verniers connected to computers to help statistical process. It is important that novices understand that the more precise a dimension must be, that greater care must be exercised in its machining. That means that it becomes more difficult to machine the part <u>consistently</u> to tolerance and also, the <u>more</u> it costs to make that part. Good design gives the shop the greatest amount of tolerance in machining to help keep the cost down.

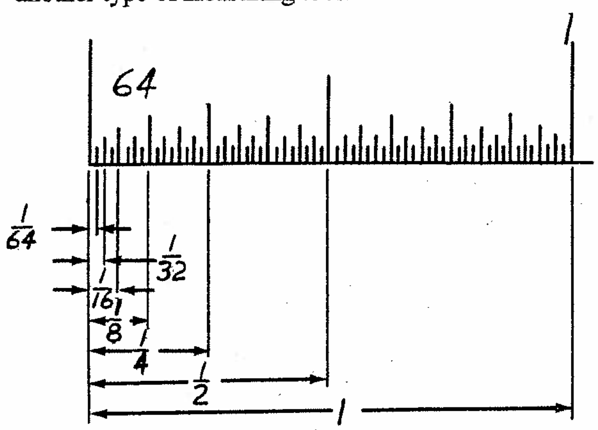
Discrimination

Discrimination is defined as the smallest amount the measuring tool can measure. Scales break down the inch to 1/64" increments and metric scales break down the millimeter to .5 mm's. Micrometers can discriminate to .0001" or .01mm's. You need to choose the right tool to take the precise measurement.

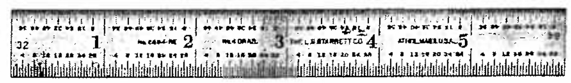
We will cover scales, micrometers, vernier calipers and dial indicators.

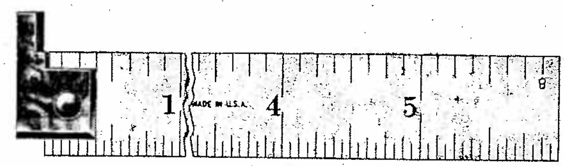
Scales

Scales are steel rulers used to make measurements that do not need great precision. Generally, the most common scale is the 6" scale. They come in many other lengths. Generally, each scale has four different graduations: 1/8, 1/16, 1/32 and 1/64. Practically speaking, any dimension 1/32" or 1/64" is made with another type of measuring tool.



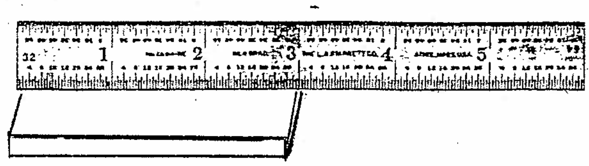
Fractional divisions of an inch


Courtesy The L. S. Starrett Company

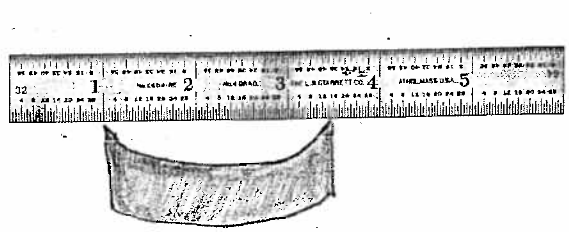

Courtesy The L. S. Starrett Company

There are a few things to remember when using a scale.

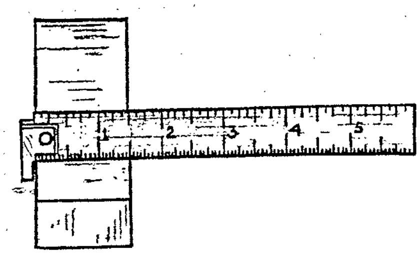
1. Always set the scale on edge to get a good reading. This eliminates what is called "parallax" error. Parallax error occurs when your eye sights down over the scale and your line of sight veers to the left or right of the line because of the thickness of the scale and because of your "predominant" or "shooting" eye body mechanics.



2. When possible, use the 1" mark as the starting reference point. The corners of the scale may be nicked, giving you a false reading. Make sure to subtract the 1" from your measurement!



3. Be certain that the scale is aligned parallel to edge of the work. If not, you'll "lengthen " the dimension.



Hold the rule straight across the piece.

4. If the scale has to be flush to the end of the work, use your fingernail. Flicking your fingernail past the work and scale, you'll feel if the edges are flush. If they aren't, you'll "catch " your fingernail.

The MICROMETER CALIPER

The micrometer is the staple of all machinists measuring tools. The thimble/sleeve assembly can be fitted into frames from 1" to 60" wide. Other micrometer variations are the depth micrometer, inside micrometer and special designed tools. English micrometers measure (discriminate) to one-tenthousandth of and inch (0.0001"). If you a take a strand of your hair, chop the diameter into thirty equal pieces, one of those pieces is what we're able to measure to. A metric micrometer measures to one-onehundreth of a millimeter (0.01mm).
Micrometers measure in decimal format. Let us consider an English or inch micrometer.

The inch is divided into one thousand equal parts. One of these parts is 1/1,000 or .001". Micrometers precisely this small amount. Cellophane paper, found on a cigarette package, is 0.001" thick. A micrometer is like an adding up all of those thousanthds of an inch.

Here's the terminology used in the shop. All shop people understand these terms, what they mean and how they're applied.

1/10	.100"	one hundred thousandths
1/100	.010"	ten-thousandths
1/1,000	.001"	one thousandths
1/10,000	.0001"	one tenth (of a thousandth)

The jargon is that everything is expressed in thousandths. The 1/10 value is not said "one tenth" of and inch but "one hundred thousandths" of an inch. The reason being is that manufacturing measuring tools measure in thousandths of an inch increment.

Shown below is a typical outside micrometer and its related parts.

Frame Holds all of the parts together.

Anvil The fixed measuring surface.

Spindle Is mated to the thimble and goes through a precise nut. As the thimble turns, the spindle moves in/out a precise amount.

Sleeve The sleeve is fixed into the frame. It has 40 equal parts inscribed on it. One turn of the thimble moves the spindle 1/40 of a turn, or .025".

Thimble Has 25 equal spaces around its circumference. It measures partial rotation of the spindle, in thousandths of an inch.

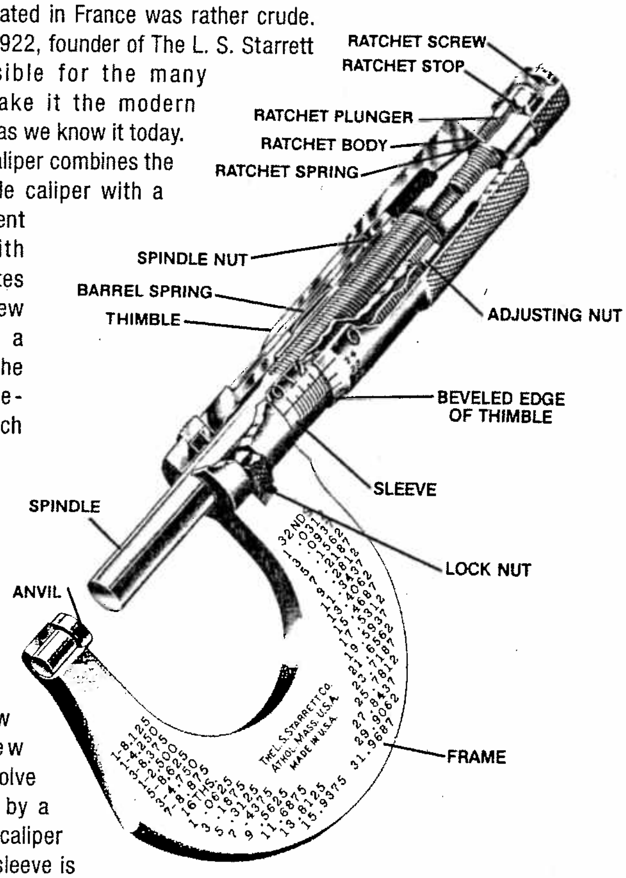
Ratchet Allows an even pressure to be applied between the measuring surfaces so the force being applied to the measuring surfaces be consistent. That is really important. The tool is not a C-clamp!

The following illustrations from the L.S. Starrett Company explain how to read a micrometer and give a little history of the tool.

Starrett®

MICROMETERS

The micrometer originated in France was rather crude. Laroy S. Starrett (1836–1922, founder of The L. S. Starrett Company) is responsible for the many improvements that make it the modern precision measuring tool as we know it today. In effect, a micrometer caliper combines the double contact of a slide caliper with a precision screw adjustment which may be read with great accuracy. It operates on the principle that a screw accurately made with a pitch of forty threads to the inch will advance one-fortieth (or .025) of an inch with each complete turn.

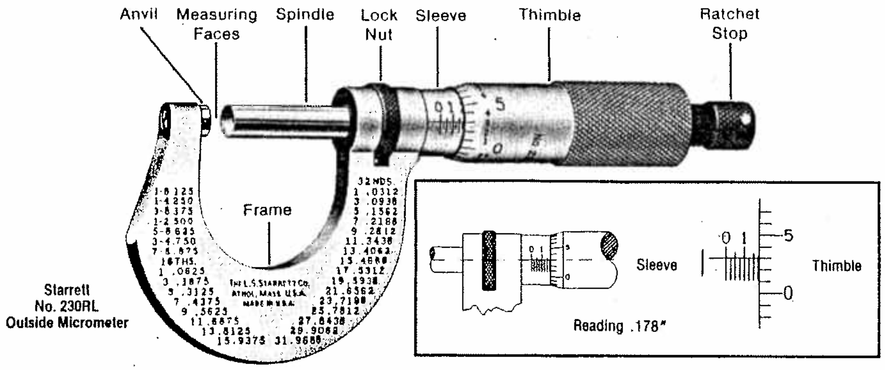


As the sectional view illustrates, the screw threads on the spindle revolve in a fixed nut concealed by a sleeve. On a micrometer caliper of one-inch capacity, the sleeve is marked longitudinally with 40 lines to the inch, corresponding with the number of threads on the spindle.

Note: See pages 17–21 on "How to Read."

Starrett®

HOW TO READ A STARRETT MICROMETER CALIPER GRADUATED IN THOUSANDTHS OF AN INCH (.001″)



Since the pitch of the screw thread on the spindle is 1/40″ or 40 threads per inch in micrometers graduated to measure in inches, one complete revolution of the thimble advances the spindle face toward or away from the anvil face precisely 1/40 or .025 of an inch.

The reading line on the sleeve is divided into 40 equal parts by vertical lines that correspond to the number of threads on the spindle. Therefore, each vertical line designates 1/40 or .025 of an inch and every fourth line which is longer than the others designates hundreds of thousandths. For example: the line marked "1" represents .100″, the line marked "2" represents .200″ and the line marked "3" represents .300″, etc.

The beveled edge of the thimble is divided into 25 equal parts with each line representing .001″ and every line numbered consecutively. Rotating the thimble from one of these lines to the next moves the spindle longitudinally 1/25 of .025″ or .001 of an inch; rotating two divisions represents .002″, etc. Twenty-five divisions indicate a complete revolution, .025 or 1/40 of an inch.

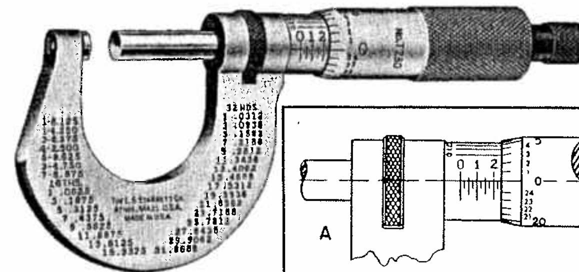
To read the micrometer in thousandths, multiply the number of vertical divisions visible on the sleeve by .025″, and to this add the number of thousandths indicated by the line on the thimble which coincides with the reading line on the sleeve.

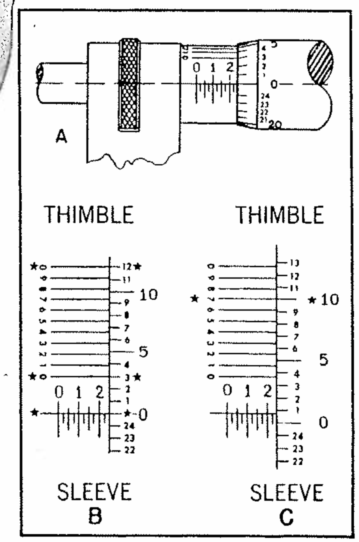
Example: Refer to the illustration above:
The "1" line on sleeve is visible, representing .... .100″
There are 3 additional lines visible, each representing .025″
3 × .025″ = .075″
Line "3" on the thimble coincides with the reading line on the sleeve, each line representing .001″
3 × .001″ = .003″
The micrometer reading is .... .178″

An easy way to remember, is to think of the various units as if you were making change from a ten dollar bill. Count the figures on the sleeve as dollars, the vertical lines on the sleeve as quarters and the divisions on the thimble as cents. Add up your change and put a decimal point instead of a dollar sign in front of the figures.

Starrett®

HOW TO READ A STARRETT MICROMETER CALIPER GRADUATED IN TEN-THOUSANDTHS OF AN INCH (.0001″)





If you have mastered the principle of the Vernier as explained on page 28, you will have no trouble reading a Vernier micrometer in ten-thousandths of an inch. The only difference is that on a Vernier micrometer, there are ten divisions marked on the sleeve occupying the same space as nine divisions on the beveled edge of the thimble. Therefore, the differnece between the width of one of the ten spaces on the sleeve and one of the nine spaces on the thimble is one-tenth of a division on the thimble. Since the thimble is graduated to read in thousandths, one-tenth of a division would be one ten-thousandths. To make the reading, first read to thousandths as with a regular micrometer, then see which of the horizontal lines on the sleeve coincides with a line on the thimble. Add to the previous reading the number of ten-thousandths indicated by the line on the sleeve which exactly coincides with a line on the thimble.

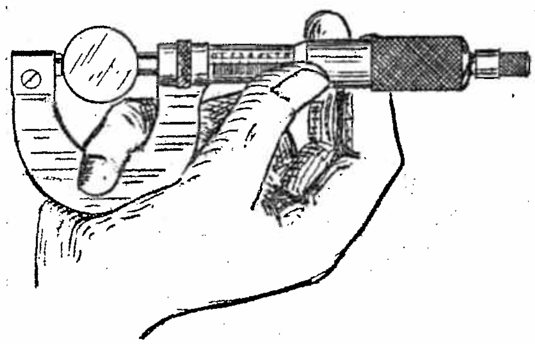
In the above illustration ("A" & "B"), the 0 on the thimble coincides exactly with the axial line on the sleeve and the Vernier 0 on the sleeve is the one which coincides with a line on the thimble. The reading is, therefore, an even .2500″. Illustration "C" above, the 0 line on the thimble has gone beyond the axial line on the sleeve, indicating a reading of more than .2500″. Checking the Vernier shows that the seventh Vernier line on the sleeve is the one which exactly coincides with a line on the thimble, therefore, the reading is .2507″.

Notes On Micrometer Usage

1. Use gentle pressure when measuring. This is not a C-clamp. It is important that you *consistently* use even pressure so your readings can be consistent, precise and accurate.

2. Clean the anvils before each use. This removes dirt and eliminates erroneous measurements.

3. Hold the tool properly. The little finger is wrapped around the frame. The thumb and index finger are wrapped around the thimble. This gives you complete control of the tool. Micrometers larger than one inch or 25mm's need two-handed use, the left hand holding the frame and the right hand turning the thimble.



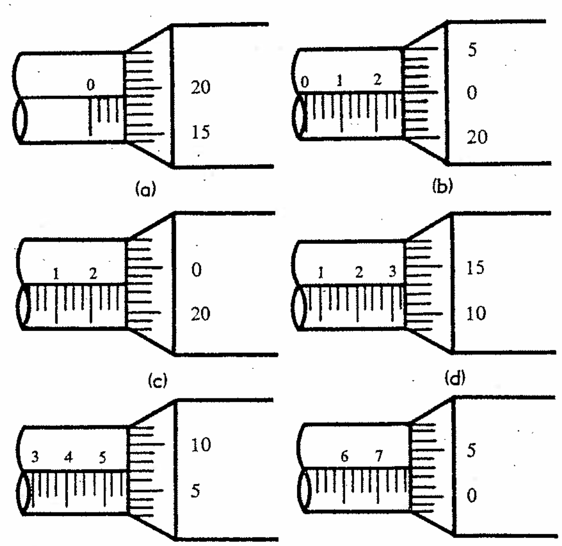
4. Never "twirl" or spin the micrometer open or closed. This could cause the spindle to snap shut tight and bend the frame.

5. DON'T DROP IT! If you do, you could bend the frame and ruin a very expensive tool. Get inspection department to check the tool's accuracy before re-using.

6. When putting the tool away, leave the faces open a little bit. This allows air to circulate and prevent the faces from rusting shut.

7. Verify the tool's accuracy before each day's use. Clean the anvil faces and close the spindle until it meets the anvil. The zeros should line up. If not, get inspection to verify the tool's accuracy. The use of a gage block or standard is also recommended.

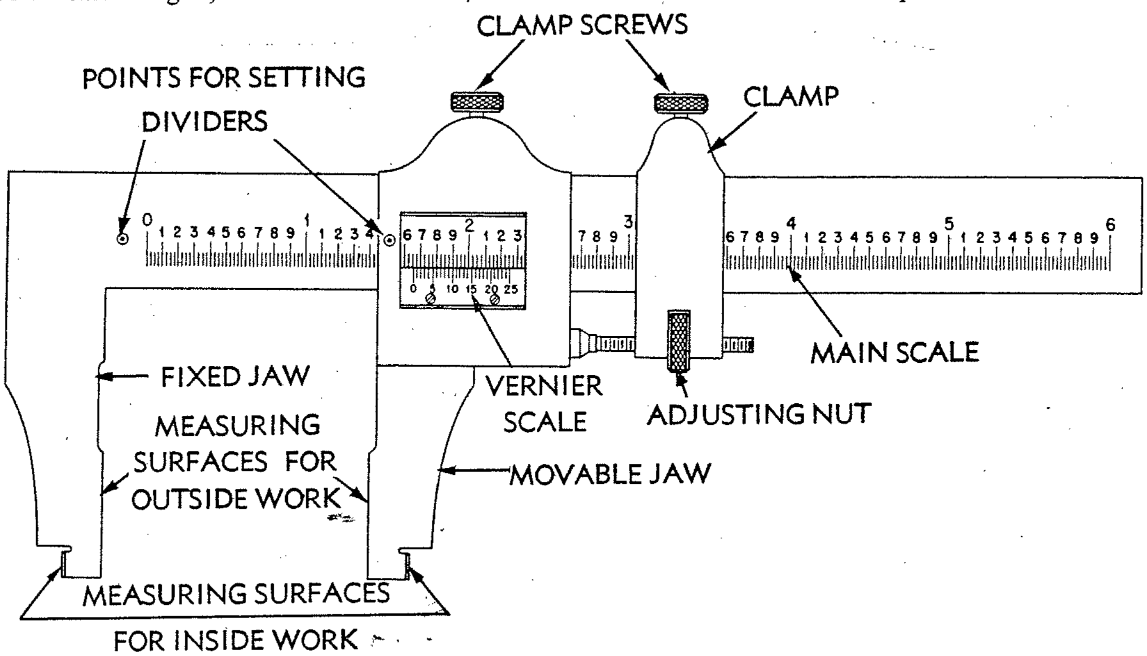
8. Clean the anvil and measuring surfaces before each use. This removes dirt and oil.



The VERNIER CALIPER

The vernier consists of a sliding scale and a fixed scale. The number of spaces in an equally given length from both scales creates the discrimination between the scales. This design is also found on rotary table, indexing devices, surveyor's instruments or any measuring instrument needing precise control for precise measuring. Basically, the vernier scale has a distance of 25 (or 50) equal parts. The main or fixed scale has 24 (or 49) equal spaces occupying the same distance. The difference between one of the 25 (or 50) spaces and one of the 24 (or 49) spaces is .001".

There are many styles of verniers. Some have points for inside measuring, some have "dial" scales instead of vernier scales (which makes them easier to read), some are electronic digital, some have depth scales, some measure angles, some are metric. You get the idea. Shown below are the main parts of a vernier.



Main Scale — This "L" shaped part is made of a solid hardened stainless steel one piece design. Note that the major divisions are 1" increments. Note that the inch is divided into 40 equal parts and that each length is .025" apart, just like on the micrometer.

Moveable Jaw — This jaw slides back and forth on the main scale. The workpiece is measured between the fixed and solid jaw. Of importance, is the vernier scale. The positive of the fixed and moveable scales determine the reading.

Fine Adjustment Nut — This adjustment knob is used to get the measuring faces to just make contact with the work piece. When the faces make contact without any looseness or "play", you have the correct "feel".

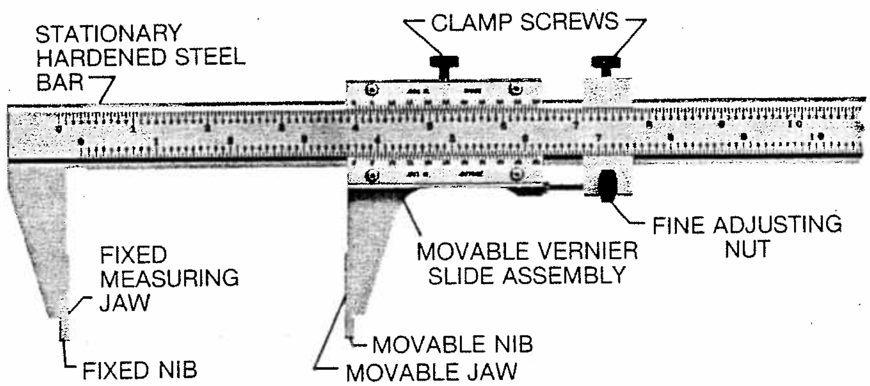
To read a vernier, note how many inches, .100 inches and how many .025" that the zero of the vernier or moveable scale has passed the main scale. Now, looking carefully, find which .001" line of the vernier scale lines up best with any line on the fixed scale. Add this thousandth vernier scale reading to the main scale reading to the above. This is your measurement.

The following pages give a little history of the tool and how to read a vernier scale.
This information is from the L. S. Starrett Company in Athol, Massachusetts.

VERNIER TOOLS

The Vernier was invented by a French mathematician, Pierre Vernier (1580–1637). The Vernier caliper consists basically of a stationary bar and a movable Vernier slide assembly. The stationary rule is a hardened graduated bar with a fixed measuring jaw. The movable Vernier slide assembly combines a movable jaw, Vernier plate, clamp screws and adjusting nut.

The Vernier slide assembly moves as a unit along the graduations of the bar to bring both jaws in contact with the work. Readings are taken in thousandths of an inch by reading the position of the Vernier plate in relation to the graduations on the stationary bar.



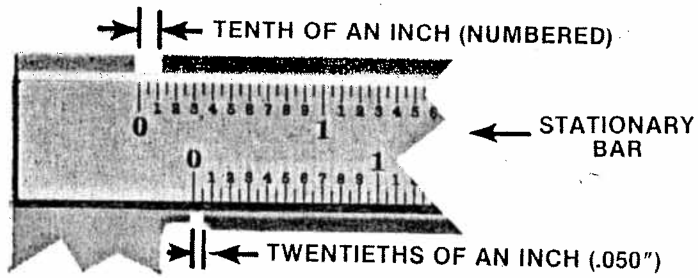
Starrett modern Vernier gages feature an improved long Vernier with 50 divisions instead of the older style with 25 divisions. The 50 division Vernier plate, with widely spaced, easy-to-read graduations and in combination with half as many bar graduations as previous old style instruments, make possible faster, more accurate and greatly simplified readings without a magnifying glass.

The Vernier principle is applied to many tools such as Vernier Height Gages, Vernier Depth Gages, Vernier Protractors, Gear Tooth Vernier Calipers, etc.

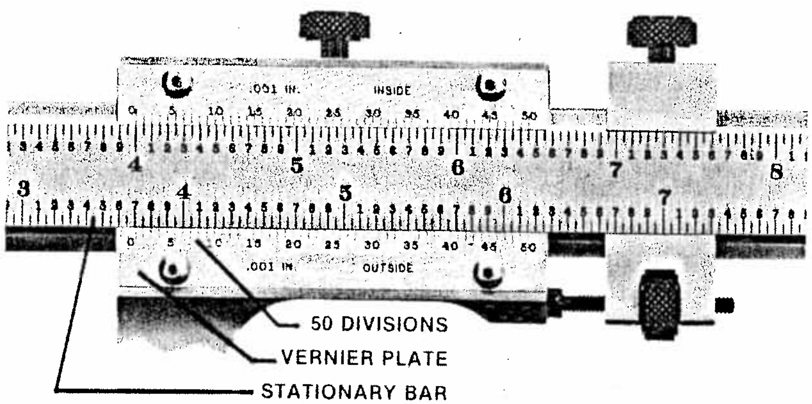
Starrett®

HOW TO READ VERNIER CALIPERS (ENGLISH)

The bar is graduated into twentieths of an inch (.050"). Every second division represents a tenth of an inch and is numbered.



The Vernier plate is divided into fifty parts and numbered 0, 5, 10, 15, 20, 25, . . .45, 50. The fifty divisions on the Vernier plate occupy the same space as forty-nine divisions on the bar.

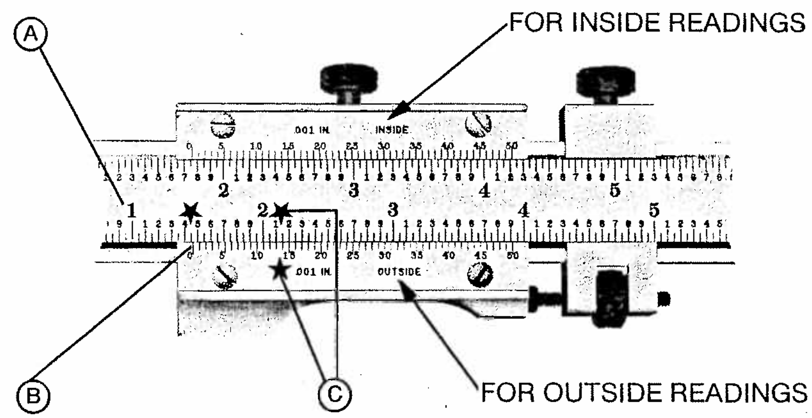


The difference between the width of one of the fifty spaces on the Vernier plate and one of the forty-nine spaces on the bar is therefore 1/1000 of an inch (1/50 of 1/20). If the tool is set so that the 0 line on the Vernier plate coincides with the 0 line on the bar, the line to the right of the 0 on the Vernier plate will differ from the line to the right of the 0 on the bar

Starrett®

HOW TO READ VERNIER CALIPERS (ENGLISH) (continued)

by 1/1000; the second line by 2/1000 and so on. The difference will continue to increase 1/1000 of an inch for each division until the 50 on the Vernier plate coincides with the line 49 on the steel rule.



To read the tool, note how many inches, tenths (or .100) and twentieths (or .050) the mark on the Vernier plate is from the 0 mark on the bar.

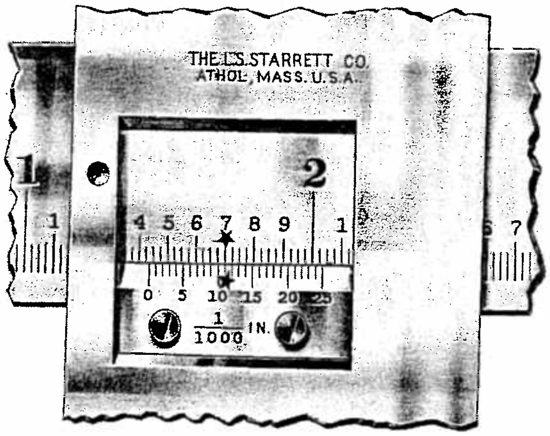
Then note the number of divisions on the Vernier plate from the 0 to a line which EXACTLY COINCIDES with a line on the bar.

EXAMPLE: In the above illustration for outside measurements the Vernier plate has been moved to the right one inch and four hundred and fifty thousandths (1.450) as shown on the bar, and the fourteenth line of the Vernier plate EXACTLY COINCIDES with a line, as indicated on the illustration above. Fourteen thousandths (.014") of an inch are, therefore, to be added to the reading on the bar and the total reading is one and four hundred and sixty-four thousandths inches (1.464).

YOU ADD TO GET YOUR MEASUREMENT

A. 1.000 on the bar
B. .450 also on the bar
C. .014 on the Vernier plate (outside)

1.464 is your measurement



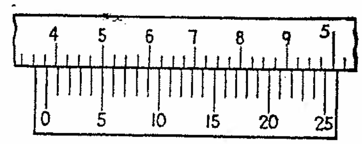
Courtesy The L. S. Starrett Company

Fig. 4-14. Vernier reading of 1.436

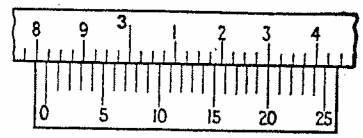
In Fig. 4-14—

the large #1 on the bar		= 1.000
the small #4 past the #1	(4 × .100) =	.400
1 line visible past the #4	(1 × .025) =	.025
the 11th line of the vernier scale coincides with a line on the bar	(11 × .001) =	.011
Total reading		= 1.436

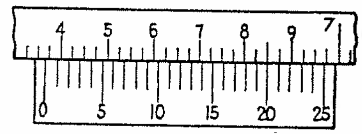
6. What is the reading in each of the following vernier diagrams?



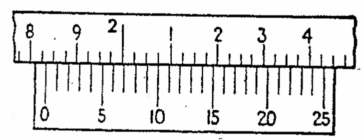
(a)



(b)



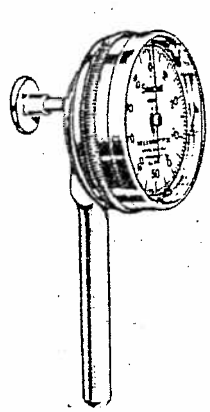
(c)



(d)

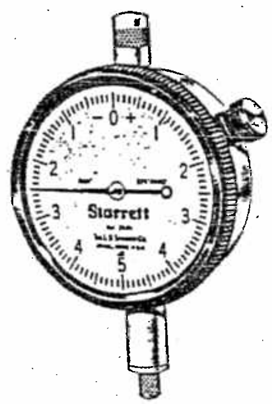
DIAL INDICATORS

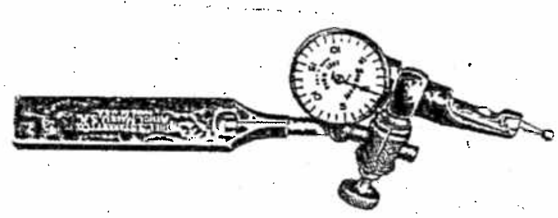
Dial indicators are used to precisely set up machines, set up tools and take measurements. Basically, they are one-handed watches. They measure in thousandths of an inch or tenths of a millimeter. As shown, they come in various styles and designs and are used in combination with magnetic bases, clamps or specialty tooling.



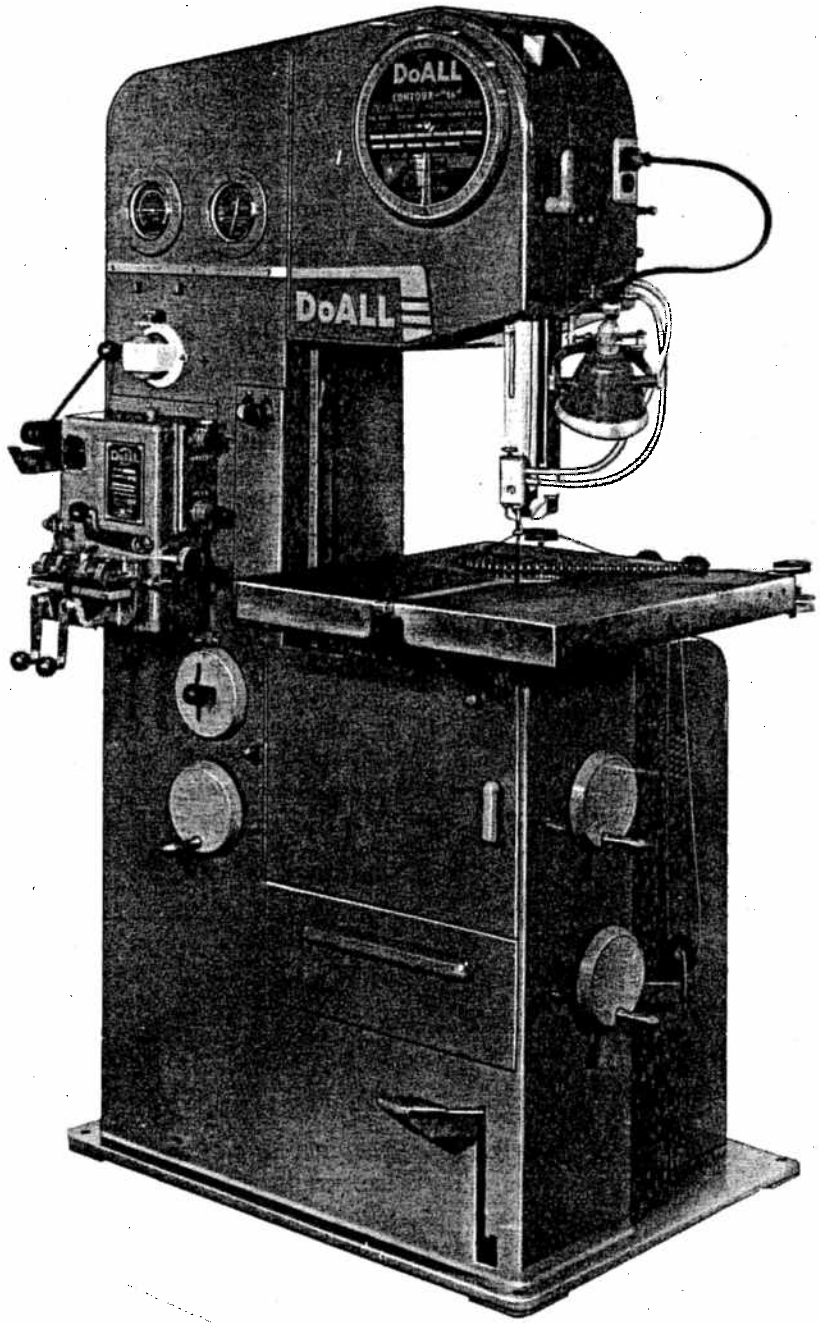
Indicators are made to a tolerance of +/- 1% *of its travel range.* Therefore, an indicator with one inch travel has an error range of +/- .010" within the one inch travel. If you use a gage block to set a zero dimension and the dimension has a +/- .005" tolerance, the error range is now +/-.00005". The indicator's travel in now limited to a travel of +/- .005".

Indicators are fragile instruments and must be well cared for.





METAL SAWING



METAL SAWING

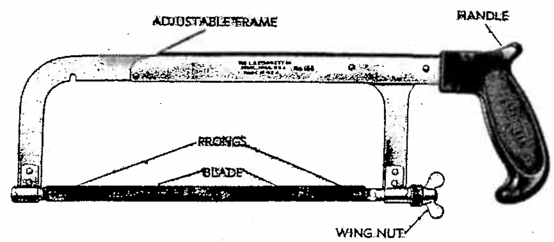
You need to make some thing, you get a piece of stock and you will find that it is too long. This happens whether you're cutting conduit, PVC plumbing, dimensioned lumber or metal bars prior to machining. The first thing you need to do is cut the stock a little longer than needed. This gives you better control of the stock so that you can better precisely make it to required size. In metal a cutting, we cut metal with either a reciprocating saw or a band saw. Let's explore these.

Reciprocating Saws

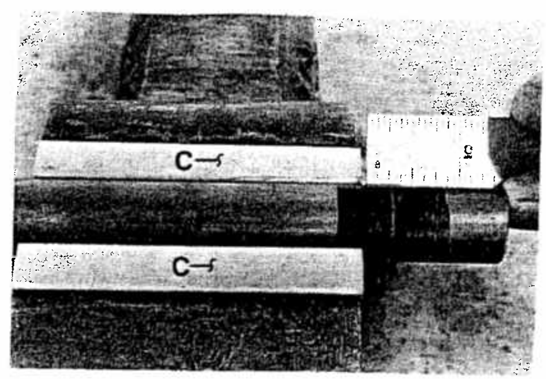
Reciprocating saws have the blade moves in a back and forth straight line motion. The saws can be machine powered or hand powered, called hacksawing. For the most part, these saws do not produce a straight, square or precise cut. They also require more time to cut off a piece because they only cut in one direction, the forward stroke. In today's manufacturing, they are replaced with the more efficient band saw.

The Hacksaw

For a quick job, you can cut stock that is ½" diameter or less with a hand hacksaw. This is an efficient, but slow cutting process. It saves time rather than setting up a power saw for a one-time cut. Anything over ½" should be power cut.



The illustration shows a frame style hacksaw. It Basically has a "U" shaped frame with a blade mounted on the bottom and a handle to hold it with. A wing nut tightens the blade in the frame. The saw teeth always point away from the handle.



To use the saw, begin by marking the required length. Extend the mark about ¼" beyond the vise jaw and tighten the vise. (This is important. If you extend the part more than that, the part will vibrate and make it difficult for you to cut.)

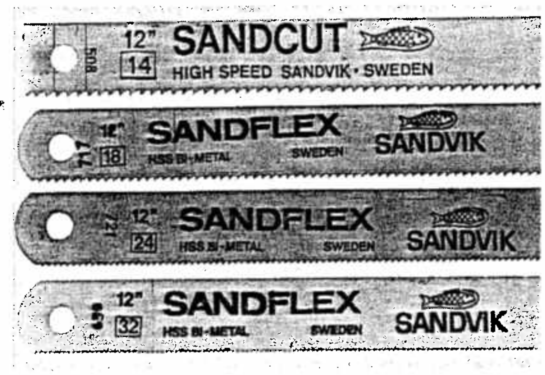
Stand with your feet at about a 45° angle to the work. Have a comfortable stance. Hold the saw by the handle and the end of the frame with your other hand. Use a smooth motions at about 120 strokes per minute. As the saw nears the end of the cut, be careful. The piece will quickly break and you'll jam your knuckles on the piece. It will hurt.

Cutting metal requires a "high speed steel" metal blade. There are four teeth spacing, or pitch, styles you can choose from. Select a blade that will always have three teeth in the cut. This prevents the blade from catching in the work and breaking. For further information, see the blade section.

Hacksaw Blades

Hacksaw blades are made in various sizes, or pitch. Pitch is the number of teeth per inch. With these saws, it is important that at least three teeth are always in contact with the work. With only two teeth, the tooth grabs into the work and breaks the blade. The accompanying chart shows typical selections.

SELECTION CHART FOR HACKSAW BLADES	
14 TEETH FOR MATERIAL OVER 1" DIAMETER	ALUMINUM AND BRASS CAST IRON COLD ROLLED STEEL
18 TEETH FOR MATERIAL OVER 1/4" TO 1" DIAMETER	ANGLES AND CHANNELS CAST IRON COLD ROLLED STEEL
24 TEETH FOR MATERIAL OVER 1/16" TO 1/4" THICK	ANGLES (LIGHT) BRASS PIPE AND TUBING IRON PIPE
32 TEETH FOR MATERIAL UP TO 1/16" THICK	CONDUIT THIN SHEET METAL THIN TUBING

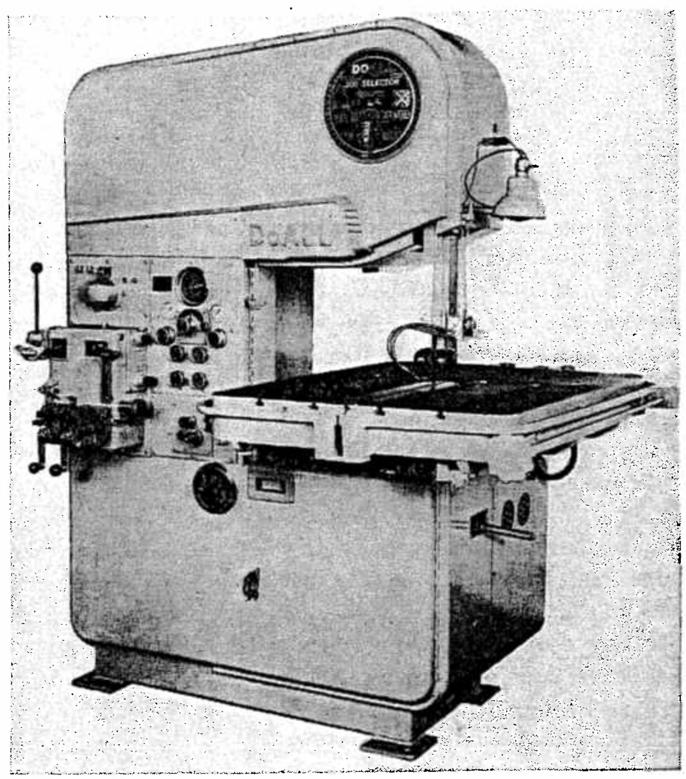


Band Sawing

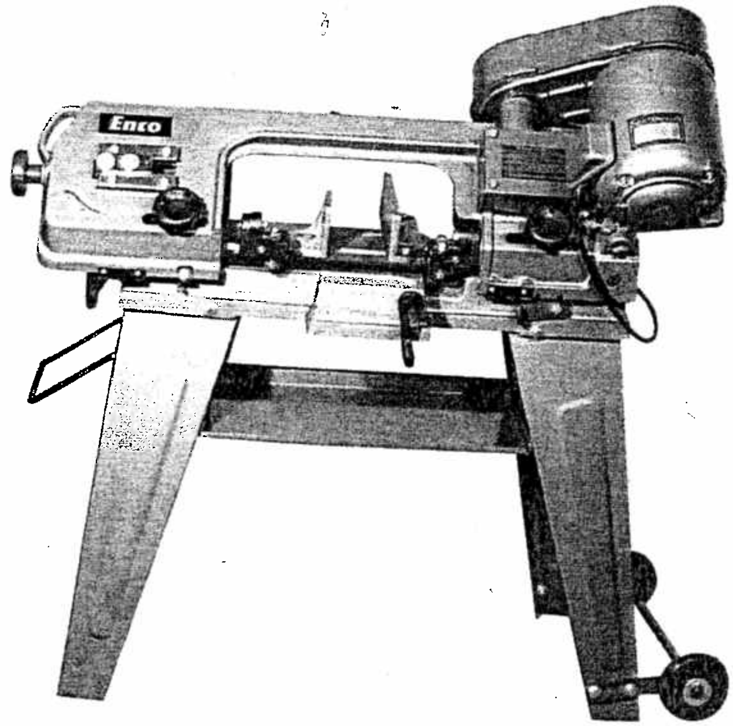
The machine shown is called a contour metal sawing machine. It is similar to a woodcutting band saw. It is used to quickly remove excess metal for final machining or cutting piece part profiles.

Band saw blades are made from high-speed steel band stock. The stock is cut to the desired length and the ends are welded together. Blades are of various pitches and tooth styles. For a better understanding of these blades, consult a manufacturers catalog. They provide much useful information.

Note that metal sawing band saws are more rigidly made that their wood counterparts and their blade speeds are a lot slower to meet surface feet speeds of various metals.



Horizontal/Vertical Band saws

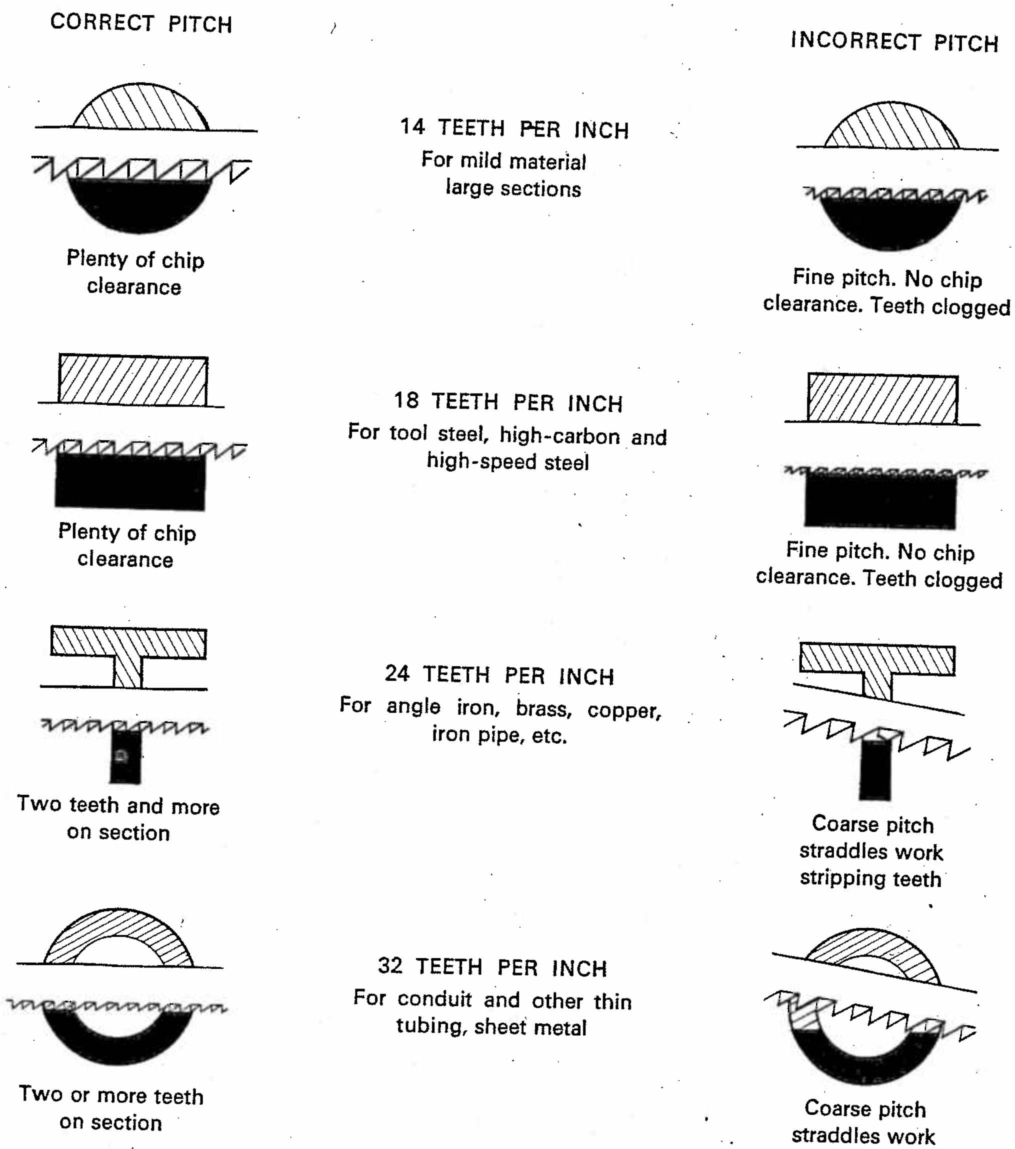


These combination machines can cut in either the horizontal position, to cut bar stock to length or in the vertical position as a vertical saw. These are popular in the experimental lab or the home shop. They use high-speed steel blades as their bigger counterparts. Production horizontal saws are used to rough cut to length production pieces. Today's machines are CNC controlled to cut pieces to precision lengths to Tolerances of +/- .003" consistently.

Band sawing is more efficient because the material is always being cut. That reduces the cutting time and improves production.

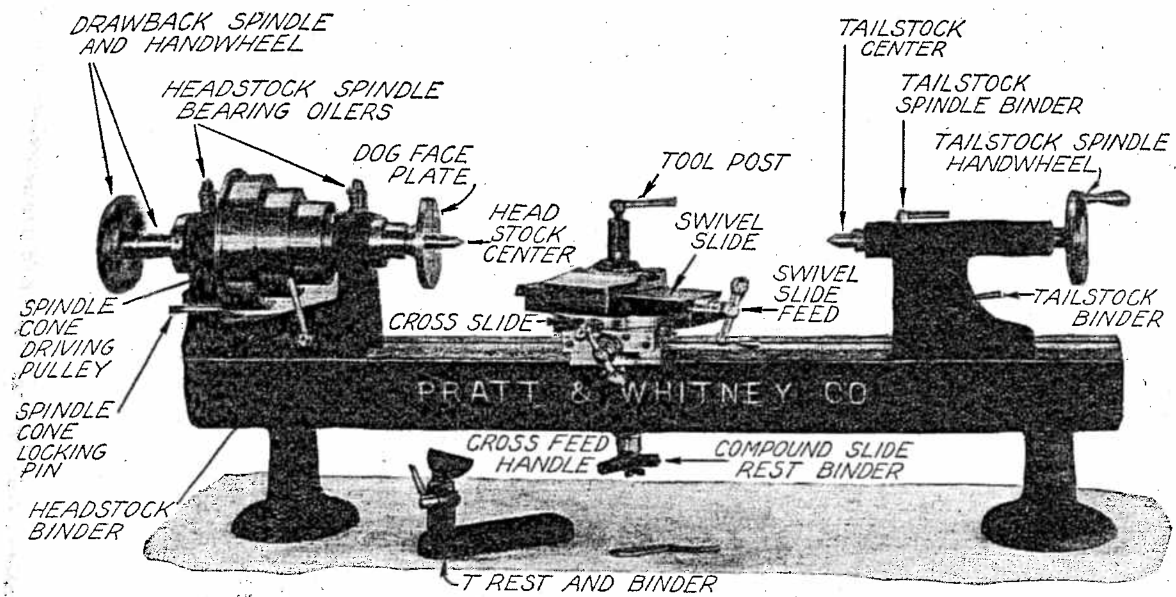
The chart on the following page shows correct pitch selection for various shapes. Also, the stock must be securely held in the vise. Always start the blade above the work and let it feed slowly in to the work. If the machine has coolant, use it. It cools the blade and helps to wash away the chips.

PITCH CHART and CUTTING DIRECTION



Hacksaw blades for various kinds of work

ENGINE LATHE



A Bench Lathe of Typical Design
Courtesy of Pratt & Whitney Company, Hartford, Conn.

THE ENGINE LATHE

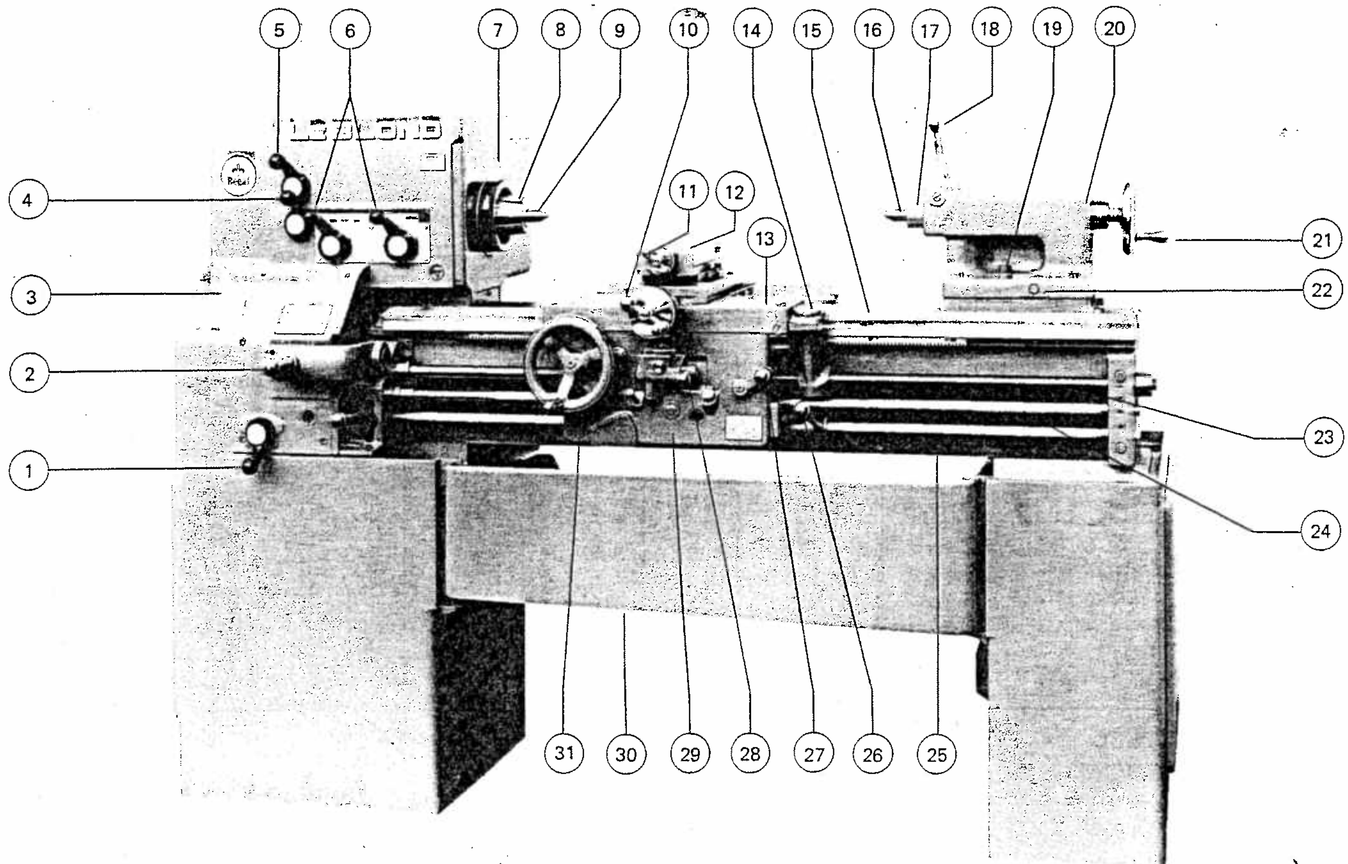
The lathe, often described as "the father of machine tools", had its beginnings as a log supported between two trees and rotated via a rope. The turner held a tool and made the tree round. The beginnings of the modern lathe were developed over four hundred years ago when John Wilkinson of England put all of the elements together to create the basis of the modern lathe. Wilkenson's contribution was to devise a means of producing accurate and consistent screw threads. With this development, wheels and axles, shafts and gears could be precisely manufactured. Also, more importantly, precise screw threads led to more accurate measuring and navigation instruments that led to further development of the New World. Shown is a typical modern engine lathe. Lathes come in all sizes from little bench top models that can turn a small part, like watch stems to big machines that will turn a submarine propeller shaft four feet in diameter by one hundred feet long.

Fig. 1. Early Tree Lathe

Other variations of the lathe are turret lathes, vertical turret lathes, model maker's lathe and such. Defined below are the major parts of an engine lathe. No matter what size, all engine lathes have the following;

Headstock	This unit is to the operators left. A spindle mounted through it, holds a work holding device. The spindle is connected to the motor via vee-belts and gears. A gear box or pulleys allows you to change the spindle RPM.
Tailstock	On the operator's right, this unit slides on the ways. It allows a tool, such as a drill chuck or a live center to be mounted in it. The tail stock can be positioned any place on the ways and locked down.
Ways	The ways are like precision "railroad" tracks that all major lathe parts rest on. The headstock, tailstock and carriage all depend on the ways for the lathe to accurately machine parts.

ENGINE LATHE



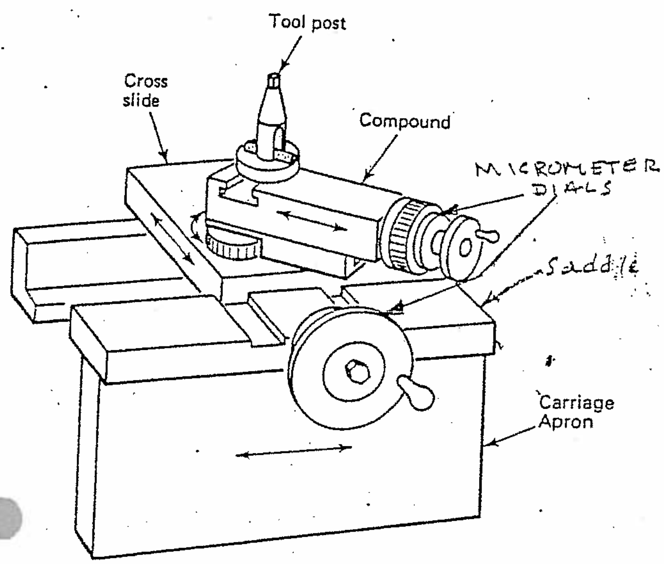
Courtesy R. K. LeBlond Machine Tool Co.

1 FEED COMPOUND KNOB
2 TUMBLER LEVER
3 QUICK CHANGE GEAR BOX
4 FEED DRIVE LEVER
5 FEED REVERSE LEVER
6 SPEED SELECTOR LEVER
7 HEADSTOCK
8 HEADSTOCK SPINDLE
9 LIVE CENTER
10 CROSSFEED HANDWHEEL
11 COMPOUND HANDWHEEL
12 COMPOUND REST
13 SADDLE
14 THREADING DIAL
15 BED WAYS
16 DEAD CENTER
17 TAILSTOCK SPINDLE
18 SPINDLE CLAMP
19 CLAMP NUT
20 TAILSTOCK
21 TAILSTOCK HANDWHEEL
22 ADJUSTING SCREW
23 LEAD SCREW
24 FEED SCREW
25 LATHE BED
26 SPINDLE START LEVER
27 HALF NUT LEVER
28 POWER FEED LEVER
29 APRON
30 CHIP PAN
31 CARRIAGE HANDWHEEL

Bed — The foundation of the machine. It is made from a big mass of cast iron (usually) the can maintain its level and parallelism. The bed must be massive enough to absorb cutting vibrations and cutting forces.

Carriage Assembly — The carriage assembly shown contains all the major components of a modern lathe. The carriage holds the tool, allows the tool to be precisely moved into the work a controlled amount and a power feed mechanism allows the tool to longitudinally travel to turn the diameter.

The compound rest and its relation to the carriage. The "standard" tool post is also shown.



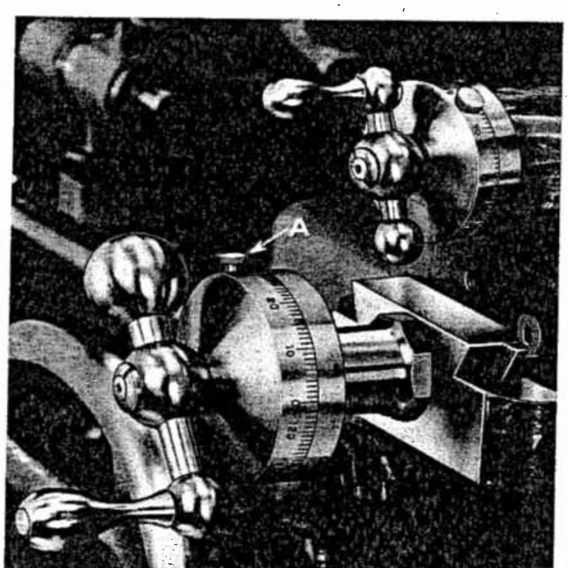
The saddle is an "H" shaped casting that travels on the ways. It controls logitudedinal turning.

The cross-slide, mounted on the saddle holds the cutting tool. It allows the tool to move in/out and controls the work diameter.

The compound rest, mounted on the cross-slide can be swiveled 360. It is used to machine tapers and set at 30 degrees to cut. The tool post holds the cutting tool.

The apron, on the front of the carriage, has a power rod going through it. It gives power drive to the carriage.

Micrometer Dials — These dials are connected to an accurate leadscrew. As the screw is turned, the tool is moved in or out. The dials are graduated in .001" increments and allows you to precisely control the tool's depth of cut.



Courtesy South Bend Lathe, Inc.

Fig. 10-24. Micrometer collars on compound slide and crossfeed. Note the thumbscrew (A)

Dials can be either direct or indirect. With direct dials, a tool fed in .100" on the dial will remove .100" on the diameter. (In actuality, the tool moves in only .050". See further on.) With an indirect dial, you need to divide the .100 by two and move the tool in .050" on the dial. This occurs because the tool actually cuts the RADIUS of the work. As the part rotates, the tool cuts .050" per side and reduces the diameter by .100". Just to confuse the situation, some lathes have a direct dial on the cross-slide and an indirect dial on the compound. Trial cuts, as explained later, help to figure how the dials are calibrated.

Quick Change Gearbox	A transmission, which by changing the position of the gear box levers, the feed rate or the number of threads per inch, can be changed without changing the gears connecting the leadscrew to the spindle.
Leadscrew	A precise thread connected to the spindle by gears. It goes through the apron and provides power to the carriage for automatic feeds and for thread cutting.
Chaser Dial	Mounted on the apron, when engaged, it shows the relationship of the tool's position to the spindle to allow precise thread cutting.

LATHE CUTTING TOOLS

For efficient cutting, a lathe tool needs several considerations. First, is the cutting tool material, followed by the material you are cutting and finally the tool shape or geometry. The following will discuss how to make a High-Speed Steel (HSS) tool for lathe work. There are many tool material types, with tungsten carbides being the most popular. Unless you have a powerful and rigid lathe, carbides are inefficient. Sharpening a HSS tool blank and using it to cut he handle will give you an appreciation of cutting tools.

LATHE TOOL BITS

A lathe cutting tool is called a tool bit. Properly sharpened, it provides a smooth cutting and long lasting tool. Cutting principles found here are also apply to end mills, drills and other tools.

High Speed Steel is a favorite because it is relatively inexpensive. It can be sharpened to any shape and works well on older or low horsepower machines. HSS is used to make drills, taps, end mills, saw blades for steel and wood, and cutting tools used in die work. Some are titanium-nitride coated, gold colored, for better tool life.

A tool bit starts out as a square tool blank. Cutting edges are ground to produce the shape. A tool with its cutting edge on its *left*, cuts from right to left (tailstock to headstock) and is called a *right* hand tool. A tool with its edge on the right cuts from the opposite direction and is called a left-hand tool! See the illustration on the following page for cutting tool illustrations and types.

As the illustrations show, the tool must be ground to certain geometry. They should be ground to the angles given, but they don't need to be exact. As long as the geometry is ground in, the tool will cut.

Side Cutting Edge Angle	This angle eases the tool into the cut. The cutting forces start at the furthest edge and moves in towards the radius. It prevents a sudden impact shock of the tool meeting the work. As the tool exits the cut, it provides a smoother exit.
Side Relief Angle	This is a clearance angle that prevents the side cutting edge from rubbing against the work. For a tool to cut efficiently, all cutting forces need to be directed to the tool nose radius. The side cutting edge and side relief angles are ground at the same time.
Front or End Edge Angle	This angle serves two functions. First, it prevents the front or end from touching the diameter. Secondly, it forms the cutting tool tip.
Front or End Clearance Angle	This is a clearance angle. It prevents the front from rubbing on the diameter as the work revolves. Both of the front angles are ground at the same time.
Back Rake	This angle starts at the cutting tip and drops down lengthwise.
Side Rake	This angle starts at the cutting tip and drops down width-wise. This makes the positive cutting edge. Back and side rake angles are ground together using a pedestal grinder.

The side rake and back rake are the critical angles of the tool. They guide the chip away from the cutting edge. They also form the "shear plane", or the angle that determines how much cutting force is needed to cut the material.

Positive rake means that the rake angles put the cutting tip as the highest point on the tool's top surface. The point diagonally opposite is the lowest point of the tool. Positive rake is used on steel and aluminum.

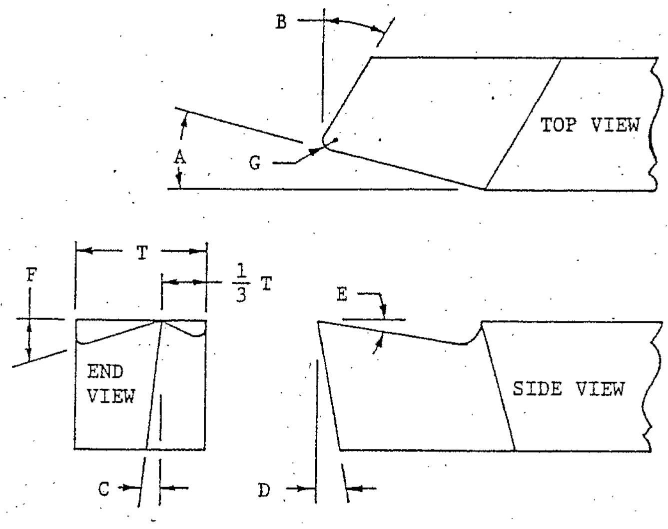
Negative rake tools put the tip as the lowest point on the top surface and its opposite diagonal point as the highest. This style is used on brass and interrupted cuts. Carbide tools are negative rake tools. It also requires more horsepower to make the cut though the stock. As a side note, rake angles are called top angles.

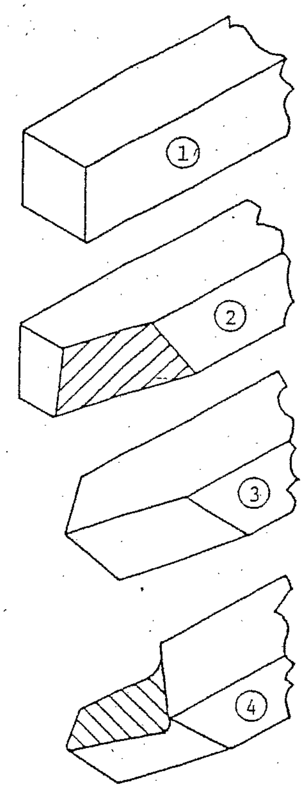
Tool Nose Radius — Grinding creates a sharp tip. This tip is very weak. Rounding it off, from top to bottom, creates great strength. Also, it leaves a better finish on the work. Generally, a tool nose radius of .015" is used.

Carbide tools, called inserts, are generally manufactured to finish size and shape. They are mounted in a special holder designed to hold their geometry and the correct cutting angle to the work. When the cutting edge dulls, the insert is rotated to the next position to expose a new edge.

GENERAL PURPOSE TOOL BIT
FOR COLD ROLLED STEEL

TOOL DESIGNED FOR
STRAIGHT TOOL HOLDERS





TOOL GEOMETRY

A = SIDE CUTTING EDGE ANGLE ---------- 15 DEGREES

B = END CUTTING EDGE ANGLE ----------- 30 DEGREES

C = SIDE RELIEF ANGLE --------------- 6 DEGREES

D = END RELIEF ANGLE ---------------- 10 DEGREES

E = BACK RAKE ANGLE ----------------- 10 DEGREES

F = SIDE RAKE ANGLE ----------------- 12 DEGREES

G = NOSE RADIUS --------------------- 1/32 INCH

T = TOOL BLANK WIDTH

NOTE:

STEPS 1 THROUGH 4 SHOWN ABOVE REPRESENT CORRECT GRINDING SEQUENCE.

LATHE TOOLS and CUTTING SPEEDS

Cutting tools are designed to certain geometric shapes for smooth and efficient cutting. Before discussing tool geometry, we should know how metal is cut.

A cutting tool is shaped like a "wedge". As this wedge enters the work, certain things happen. The tool or work rotates creating a force. Feed pushes the tool into the work. These two actions, rotation and feed, create friction which in turn creates heat. Heat is created as the tool is forced into the work. The heat loosens the molecules at the point where the cutting edge meets the material. The heating and pressure effect cause the molecules to distort and become separated from the base metal. This creates the "chip". This simplistic explanation is known as "Deformation Theory".

Cutting tool materials are made of a combination of materials. When metals are mixed together, we create an "alloy". These cutting tool alloys have certain traits.

1. High Red Hardness – A high temperature point prevents the tool from softening as the heat softens the steel and causes the steel to loose its hardness.
2. Abrasion Resistance – As metals rub over each other, the rubbing effect acts like sandpaper. This action abrades (wears) and dulls the cutting edge.
3. Shock Resistance – As the tool starts to cut, it suddenly encounters a resistance force. This sudden force is called "shock". Some cutting tool materials are better at handling shock than others.

Cutting tool materials are made of a combination of these requirements, depending on their use. Metallurgists can make alloys to meet the needs. The following is list of some of the more common types of these steels.

High Carbon Steel
This is the first of the modern steels (circa 1890's). It has a high carbon content. This high carbon content gives the tool steel the property of being hardenable. When the steel is hardened, it increases the high red hardness and shock resistance. Mostly woodworking tools, these steels are slowly being replaced by High Speed Steel and carbides.

High Speed Steel
Adding other elements to the high carbon steel, new properties can be created. High red hardness, abrasion resistance and shock resistance can be altered. Today, lathe tool bits, drills, end mills, taps and dies and other cutting tools are made of HSS. Some of these tools are Titanium Nitride coated. A small amount of TiN is deposited on the tool surface producing a gold color. This coating helps the tool cut easier and last longer.

Carbide
This is the industrial cutting metal of choice. This material works best on heavy duty, solid and high horsepower lathes. They don't work well on older small machines. There are essentially two types of carbides used metal cutting – one grade for cutting non-ferrous and cast-iron metals and the second for cutting steels. Grades C-1 to C-4 are the non-ferrous/cast iron types and the C-5 to C-8 are the steel cutting grades. Consult a manufacturer's technical manual for proper grade selection and use. Carbides are made from tungsten and cobalt. The proportions of each mixture determine high red hardness, abrasive resistance and shock resistance. That's why the four grades of each type.

Industrial Cutting Tools
Man-made diamonds, ceramics, cermets and other materials are used in modern manufacture. These are high tech materials used in CNC and volume or exotic material removal

Most school shops and home machinists use HSS and carbides because of their relatively cheap cost and versatile applications.

SPEEDS and FEEDS

In order for a tool to cut effectively, the tool or work must rotate at the correct "cutting speed". Cutting means that after one minute, a chip of a certain length must be maintained. This is controlled by the RPM. The correct speed is reached when the chip has a nice even surface and if steel is being cut, the chip starts to turn a tan color. If the speed is too slow, extreme pressure must be used to cut. This extreme pressure will actually tear the metal and could break the cutting tool. Conversely, if the speed is too high, the tool overheats, looses its hardness or even melts or worse yet, hardens the metal and then it can't be machined.

How do you determine speed?
Lots if time, money and research has been and is being spent on effective speed/feed and cutting tool combinations for various materials. For our purposes, we'll use the "100 Rule". This rule is perfect for conventional machines because these machines seldom have the calculated RPM. CNC machines and machines with variable speeds can better closely achieve the calculated RPM. The 100 Rule applies to HSS cutting tools. In order to determine the correct RPM, you need to know three things:

1. the material to be cut,
2. the cutting tool material, and
3. the diameter of the tool or work. Remember, on a lathe type machine, the work turns (which becomes the diameter), while a drill press type machine, the tool turns (and is the diameter).

"Surface Feet per Minute" determine speed. It means that after one minute of cutting, the chip, when uncoiled, should measure 100 feet long. Simplistically, that what it means. Each material has its own "ideal" SFM.

100 RULE – HIGH SPEED STEEL

Material type	Surface Feet per Minute
Low carbon steel, cast iron	100 sfm
Medium, high carbon, alloy	
Stainless steels	50 sfm
Copper, brass and bronze	150 sfm
Aluminum	300 sfm

For carbide cutting tools, use 3x the materials SFM

This information is needed to calculate RPM. The formula is;

$$RPM = \frac{4 \times SFM}{Diameter}$$

SFM: surface feet per minute
DIA: diameter of work or tool

Example 1: Turn 2" Dia mild steel bar stock with a HSS lathe tool then drill a ½" dia. Hole on its end.

RPM = (4 x 100sfm) / 2"dia = 400 / 2 = 200 lathe RPM

Drill a ½" diameter hole by 1" deep on the end of the piece.

RPM = (4 x 100sfm) / ½"dia = 400 / ½ = 800 lathe RPM

Rule #1: The bigger the diameter at the cutting edge, the slower the RPM. In this example, the lathe tool cuts at 2" diameter, while the drill cuts at ½" diameter. When turning, the diameter becomes smaller. Therefore, you need to increase the RPM as the diameter decreases.

Example #2: Drill a ¼" diameter hole in stainless steel, then one in aluminum.

RPM = (4 x 50sfm) / ¼"dia = 200 / ¼ = 800 RPM drill spindle speed

Drill a ¼" diameter hole in aluminum.

RPM = (4 x 300sfm) / ¼"dia = 1200 / ¼ = 4800 RPM drill spindle speed

Rule #2: The harder the material, the slower the spindle RPM needs to be.

The second factor is feed. Without feed, the tool doesn't cut. Take an electric drill, chuck a drill in it and turn it on. The drill won't cut unless you push, or feed, it into the work. Let's just consider lathe feeds.

Remember the tool nose radius? On a lathe, the tool nose radius is critical. TNR's are generally .015, .025 or .032. The smaller the lathe, the smaller the TNR. The rule is:

Feed rate is 1 TNR max for roughing cuts and ½ TNR for finish cuts.

The third factor is the equation is Depth of Cut (DOC). In metal cutting, the DOC is the least variable of the three. The rule is:

Rough-cuts use a DOC of 2 to 3 times the TNR and 1 TNR minimum for finish cuts.

Getting back to example #1, turn 2" dia mild steel, the STARTING combination with a HSS tool is:

Roughing cuts: 200RPM at .015 IPR at .045" DOC. (Use 050" DOC, easier to adjust the Micrometer's dial than by using .045. Math!)

Finishing cuts: 200RPM at .007 IPR at .015" DOC.

After calculating speeds and feeds, remember that all of this is *theory*. A lot of factors affect how well metal is cut. These include machine rigidity, set-up rigidity, tool sharpness and a host of other things. Use common sense. If it doesn't feel right, it probably isn't. These values are "starting values". Adjust accordingly. When using carbide tools, the general rule is to take the 100 Rule SFM values and multiply them by 3. It works. All else remains the same.

The following is how to sharpen a high-speed tool blank. It requires lot of practice. The last page of this section deals with speed and feed rules for drilling, turning and milling operations. If nothing else, it IS the most important page in the Notebook.

SPEEDS and FEEDS TABLE

100 RULE- Surface Feet per Minute (SFM)

Material	HSS	Carbide
Low carbon steel, cast iron	100	300
Medium carbon, tool and Stainless Steel	50	150
Copper, brass, bronze	150	450
Aluminum	300	900

Use to calculate RPM:

$$RPM = \frac{4 \times SFM}{Dia.}$$

Dia: use diameter of work or tool.

LATHE WORK – TURNING and BORING

FEED RATES: Roughing: 1TNR Max, by 2-3TNR DOC; Finishing: ½ TNR Max, by ½-1TNR DOC.
Threading RPM: use ¼ turning RPM / Facing RPM: over 4" dia, use ½ stock diameter
Recessing, grooving, parting: ½ turning speed, hand feed./Knurling: ¼ turning speed, .015/.020 IPR feed.
TNR: Tool Nose Radius/ DOC: Depth of Cut / IPR: feed in Inches Per Revolution

MILLING WORK – SPEEDS and FEEDS

SFM	IPR per Tooth
100/300	.007
50/150	.003
150/450	.010
300/900	.015

Slitting saws, form cutters: use ½ correct IPR value.

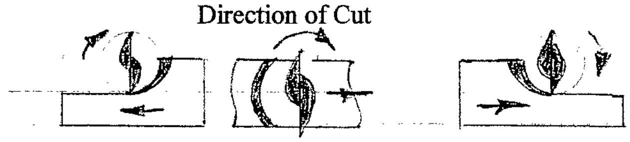
IPM= feed in Inches per Minute
IPM=RPM x Feed per Tooth x # of Teeth
IPR= feed in Inches per Revolution
Rough cuts: DOC = .050 - .100 Min.
Finish cuts: DOC = .030 Min @ ½ roughing

END MILL Feed Rates per Tooth

To ½ Dia	.0002 / .003
½ Dia and up	.003 / .007

The smaller the end mill diameter and the harder the material, use a lower feed rate.

Direction of Cut



CLIMB NUETRAL CONVENTIONAL

DRILLING and REAMING

Drilling Feeds in IPR

To 1/8 Dia	.0002 / .002
1/8 to ¼	.002 / .004
¼ to ½	.004 / .007
½ to 1"	.007 / .015
1.00 Dia up	.015 / .025

Drill IPM = RPM x Drill IPR

IPR for C'bore, S'face, C'sink

To 3/8 Dia	.004
3/8 – 5/8	.005
5/8 - 7/8	.006
7/8 – 1 ¼	.007
1 ¼ - 1 ½	.008

Note: Use 1/3 correct RPM for the above.

Reamer Rule: To ½ Dia, drill 1/64" (.015) undersize, 1/2 to 1.00 Dia, drill 1/32" (.031) undersize.
RPM is ½ drill size diameter. Feed is twice drill size diameter.

BAND SAWING

Always have a minimum of three (3) teeth in cut. Choose correct pitch blade.
High Carbon Steel blade: Feet per Minute use ½ HSS material feet per minute.
Bi-metallic HSS blade: Feet per Minute use HSS values.

POINTS TO REMEMBER

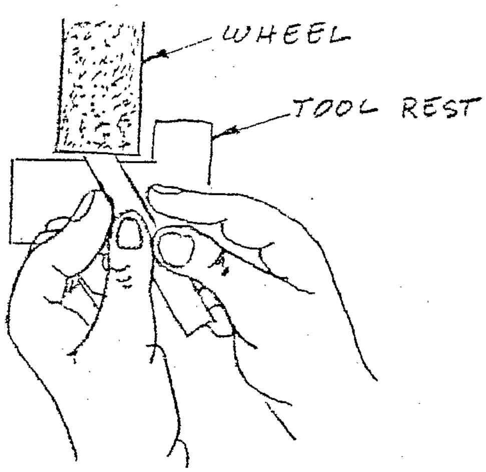
These are beginning guidelines. Adjust accordingly for:
Rigidity of set up, rigidity of work and actual cutting performance.
Machinist experience will help in establishing good cutting performance and cutting action.
These values are applicable to both conventional and CNC machines.

LATHE TOOL GRINDING

High Speed Steel lathe tools are ground to shape by hand using a pedestal grinder. Once you've mastered the technique, it's pretty easy, but oh to master it! Remember, the goal is to have a tool having proper geometry for our application.

Before grinding the tool, let's review some details.

1. ALWAYS wear safety glasses. No need to explain why.
2. Don't have more than 1/16" space between the wheel face and the tool rest.
3. Dress the wheel so it is sharp. If a rotating wheel shows any kind of gloss on it, the wheel is dull and loaded with metal particles and must be "dressed" (remove the embedded metal particles).
4. Quench the tool often in water. Don't overheat the tool, as it will lose its hardness. If the tool turns blue, you lost some hardness.
5. Hold the tool blank firmly. You need to be in control of it. Use your thumb and first finger of each hand. The first finger of your right hand rests on the tool rest's edge and acts as a guide to move the tool back and forth in a straight line.



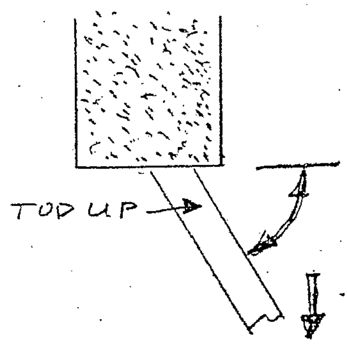
Proper holding technique of the tool blank, the wheel and tool rest. Remember to move the blank left and right so as not put a groove on the wheel's face.

The normal sequence of sharpening is:

1. Grind the end or front cutting edge and front clearance angles first.
2. Grind the side cutting edge and side clearance angles next.
3. Grind the back and side rake angles last.
4. Lastly, grind the tool nose radius. Don't make it too large or the tool will not cut very well.

See the following illustrations.

1. FRONT CUTTING EDGE AND RELIEF ANGLES



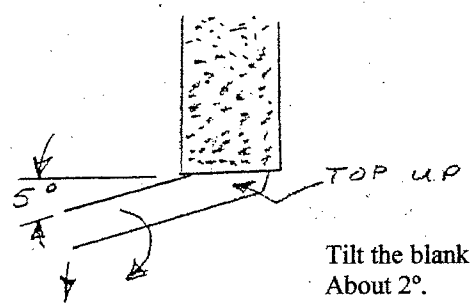
Front cutting edge:
Hold the blank from 75° to 80° to the wheel face.

Front relief:
Tilt this end down about 10°

Holding the blank at these approximate angles will grind in both angles at the same time.

2. SIDE CUTTING EDGE AND RELIEF ANGLES

Hold the blank about
5° to the wheel face.

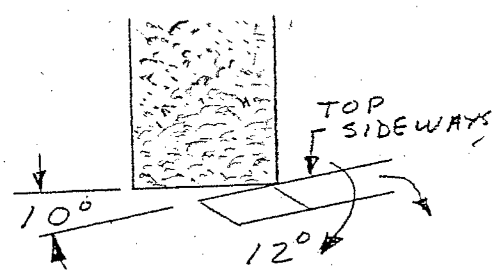


Tilt the blank towards you
About 2°.

Drop the far end of the blank
Down about 10°.

Holding the blank in these reference angles will properly grind in the side angles.

3. BACH AND SIDE RAKE ANGLES

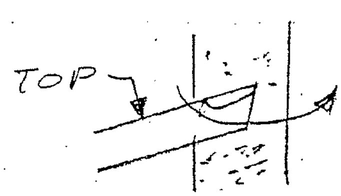


Back rake:
Position the blank about 10° to the wheel face.

Side rake:
Tilt the blank so the front cutting edge is perpendicular to he tool rest. Now, roll the blank about 12°.

This set up allows the rake angles to be round in. Note that corner diagonally oppositethe tip will touch the wheel corner first. When done, the tip will be the highest point and the opposite corner will be the lowest.

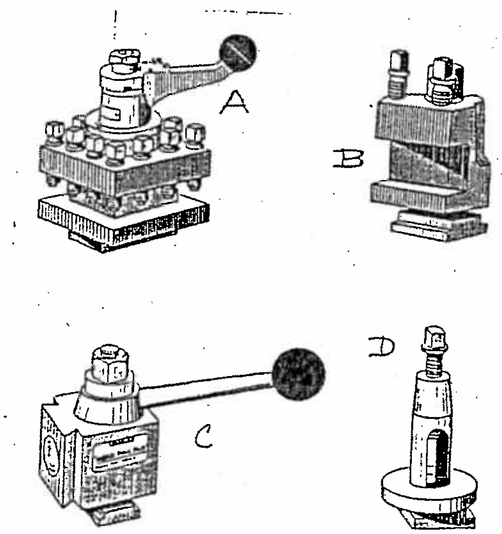
4. TOOL NOSE RADIUS



Hold the intersection of the front and side planes to the wheel face. Swivel the blank to grind in a radius of about .015". This works better if you use a little "body English".

SETTING UP THE CUTTING TOOL

The tool must be securely held in the lathe for it to cut efficiently. There are several tooling options depending upon the lathe compound rest design.



Four common types of tool posts.

A. Square turret post used on heavy-duty production machines. Tools are shimmed to get the tip on center.

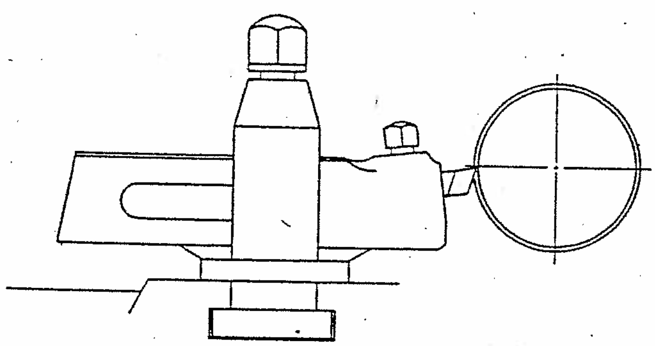
B. Solid holder used to hold carbide tool holders.

C. "Aloris' type holder tool post. The tool mounts in a holder that slides over dovetails. The handle forces a wedge to lock the adapter solidly. This is a production set up used to hold facing, turning, boring and other tool types.

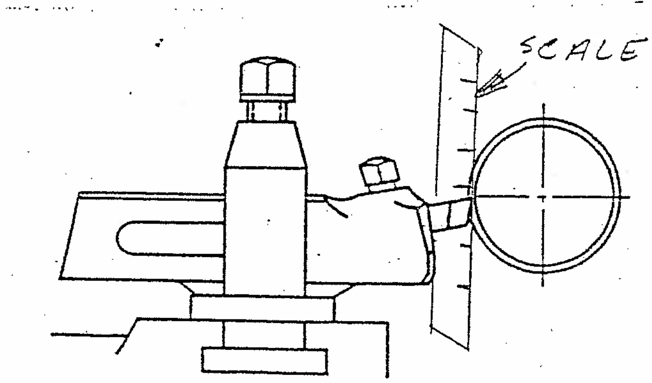
D. The rocker tool post is traditionally found on older engine lathes. The tool is mounted in a holder, which is swiveled to put the tool on center, then tightened down.

There are THREE RULES to remember when setting up any lathe tool.

Rule 1: The tool MUST be on center.

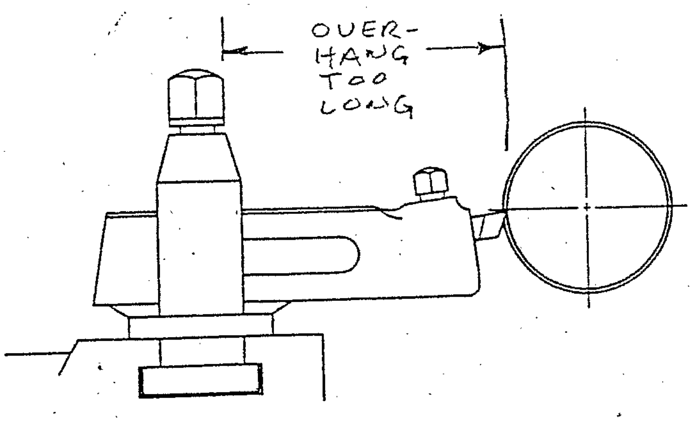


For the tool to cut accurately, it MUST BE on center or just a "hair" above it. If its below center, the tool will get sucked into the cut and ruin the part.



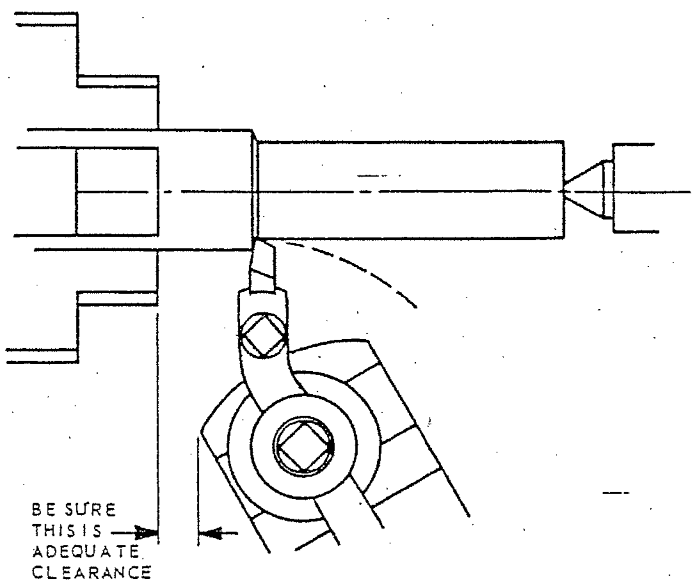
The easiest way of setting the tool on center is to use a scale. Place the scale on the diameter. Bring the tool tip so it just touches the scale. If the scale is perfectly on center, it will be vertical. If the scale is tilted in either direction, the tip is either too high or too low.

Rule #2: Choke up.



If the tool and or tool holder extends or overhangs too far, the tool will spring up and down as it cuts. Think of it as walking on a diving board. The further out you walk the more the board bends. When cutting steel, the tool essentially does the same thing. The springing action will cause to "chatter'. The finish will be poor and dimensions will not be held. The on-center illustration shows a properly choked up tool.

Rule #3: Tool clears the work and chuck.



Be certain that the tool is as far as practical to left on the compound rest. This helps prevent the chuck jaws from hitting the compound rest and damaging the machine. Put the spindle in neutral and spin the spindle by hand to check the clearance.

Check to be sure that the cut can be made without danger of the rotating chuck striking the tool holder or compound. Also, position the tool holder so it will swing clear of the work if the tool holder slips in the tool post.

These three rules apply to ALL lathe tools: facing, turning, boring, threading, knurling, necking, grooving and cutoff tools. A good machinist does all of this intuitively.

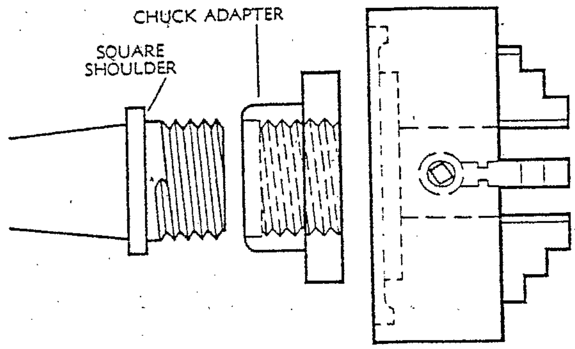
LATHE WORKHOLDING

Work must be held securely so it can be precisely machined. Work is held:

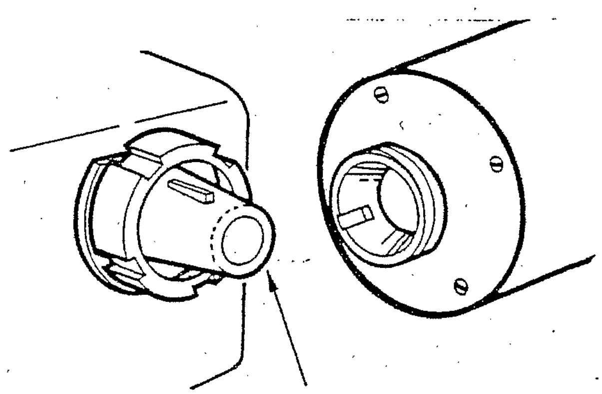
1. Between centers,
2. Using a 3 or 4 jaw chuck with and without a live center,
3. In collets
4. Special fixtures.

Work holding fixtures are mounted directly on the spindle. There are basically three spindle styles:

The threaded spindle is the oldest style. The chuck is threaded onto the spindle. It does not center itself precisely as other styles. It can unscrew itself off the spindle if run in reverse.



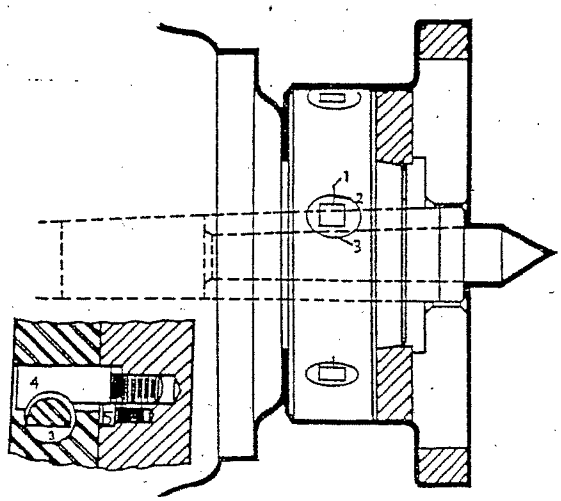
The tapered spindle has along taper that the chuck mounts on. The key drives the chuck so it doesn't slip under the cutting loads. A locking ring secures the chuck on the taper and prevents the chuck from falling off if the ring loosens.



TAPERED SPINDLE NOSE

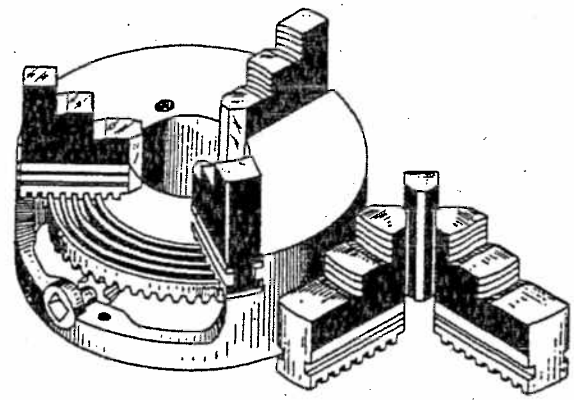
A cam lock chuck has fingers that come from the back of the chuck body. The fingers align themselves into the spindle face. As the cam is turned, it locks itself into the fingers notches, pulling the chuck tight on the short taper.

American standard lathe spindle nose



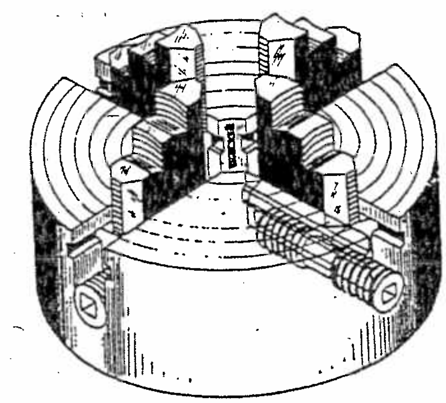
The 3-Jaw Universal Chuck

The 3 jaw universal chuck is used to hold round work either by the outside (O.D.) diameter or by its inside (I.D.) diameter. When turning the socket with a square wrench, all three jaws move in or out simultaneously. When new, a 3-jaw chuck is accurate within .002" runout. When worn, they runout as much .030". That's why you must know when to use another type chuck for precise machining.


3-jaw universal geared scroll chuck

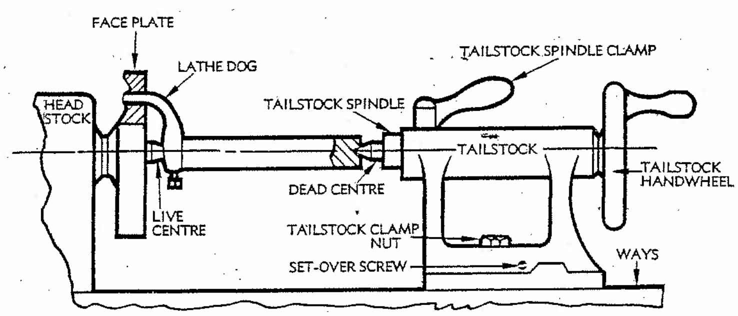
The 4-Jaw Independent Chuck

The 4-jaw chuck is called an independent chuck because each jaw moves individually and independent of the other jaws. The chuck can hold round work, square work, ellipses, ovals or most any other shape. The work can be moved to one side to machine an eccentric. Work in a 4 jaw chuck can be set to "zero" runout.


4-jaw independent chuck

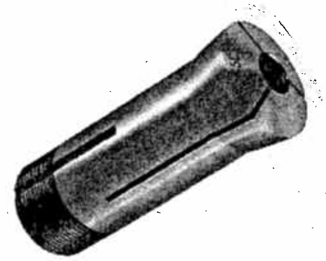
Turning Between Centers

Turning between centers provides an accurate means of machining a shaft straight, turn shoulder diameters concentric to each other and the parts centerline. The ends are faced and a cone is drilled into the end. The cone matches the centers' point. A ball bearing tailstock live center turns with the work.



Collets

Collets are steel "fingers" that mount into an adapter. As the collet is pulled into the adapter, the fingers collapse and tight against the works' O.D. This design makes the collet very accurate, as the collets' runout is what the spindle's run out is. Collets are of the steel finger type or of the rubber flex type.



A few rules about mounting/dismounting chucks.

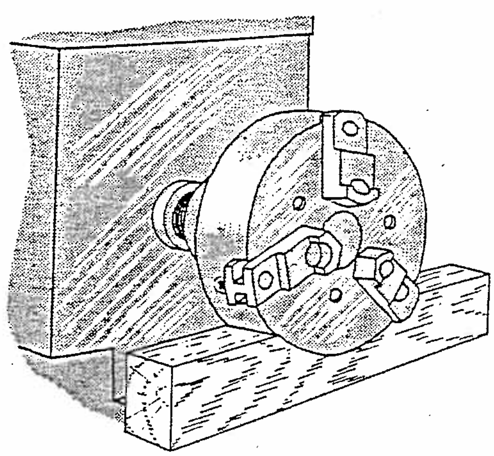
1. Chucks are heavy! Be sure to;
   A. place a board on the ways to protect them if you drop the chuck.
   B. Get help if you need it or use a small crane.

2. Always put the machine in its lowest gear to prevent the spindle from turning when loosening the chuck or tightening. it.

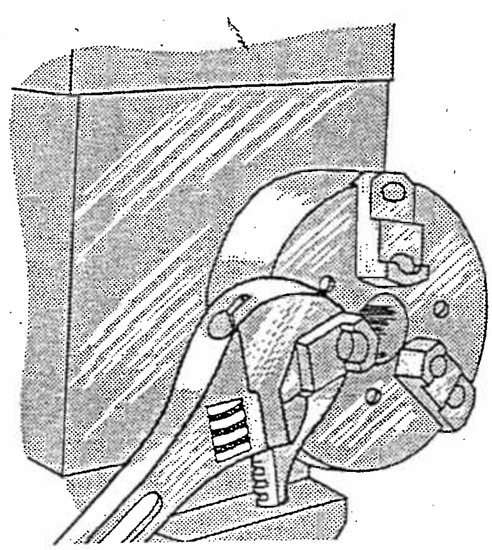
3. Always clean the spindle and mounting hole to be free of chips and dirt. This prevents runout and mis-alignment.

4. Use an adjustable wrench to loosen a threaded chuck.

5. Lastly, DO NOT risk injury to yourself. A chuck is heavy. If it falls, you could hurt your hand, feet or back. Get help if you need it.



Lathe chuck supported with wooden cradle.



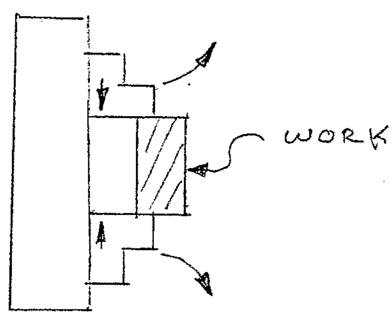
SETTING UP THE 3 JAW CHUCK

Once the 3-jaw chuck is mounted on the spindle, work holding is easy. Using the wrench, place it the square socket, turn and close the jaws on the round or hexagon stock. However, there are few precautions.

1. NEVER, NEVER leave the chuck wrench into any socket. It can become a deadly missile if the spindle is accidentally turned on. A good practice is to always HOLD it in your hands. This way, you'll never leave it in the socket by accident.

2. If the chuck has three sockets, tighten each socket with EQUAL pressure. This equalizes the holding force on all three jaws and prevents "springing" the jaws.

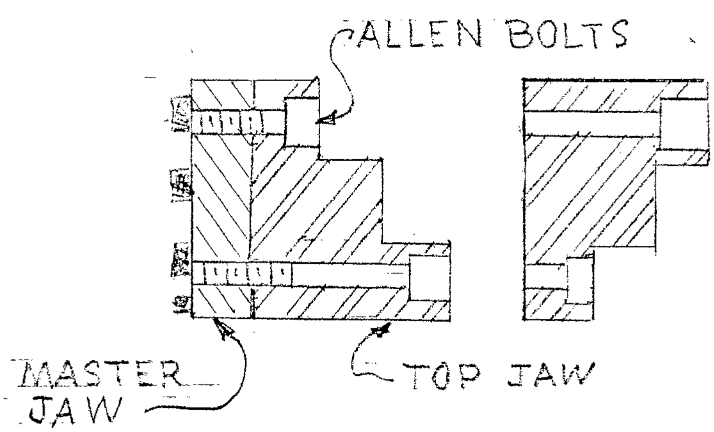
3. Don't over tighten the chuck jaws when machining thin pieces so as not to "spring" the jaws. See figure 2.



4. If the part extends more than three diameter lengths beyond the chuck jaw, it must be supported. Face the piece, extend it as far as necessary, center drill and place a live center to support it. Without support, the work will push away from the tool, called springback, and leave a tapered diameter. See figure 2.

5. When changing the jaws from OD work holding to ID work holding, look at which style of jaw set on the chuck. There are two styles, the "top jaw" set and the solid jaw set.

Top Jaw Set:



Loosen and remove the two Allen bolts.

Remove the top jaw from the master jaw.

Turn the jaw 180 °, install and tighten the bolts.

- REMOVE
- FLIP
- RESEAT

Solid Jaw Set

Solid jaws are two separate sets. One set is for OD holding and the other is for ID holding. On this style, you have to completely remove and replace each jaw. Notice that each slot is marked 1, 2 and 3. Note too, that each jaw is marked 1, 2 and 3. They slide into the corresponding slot.

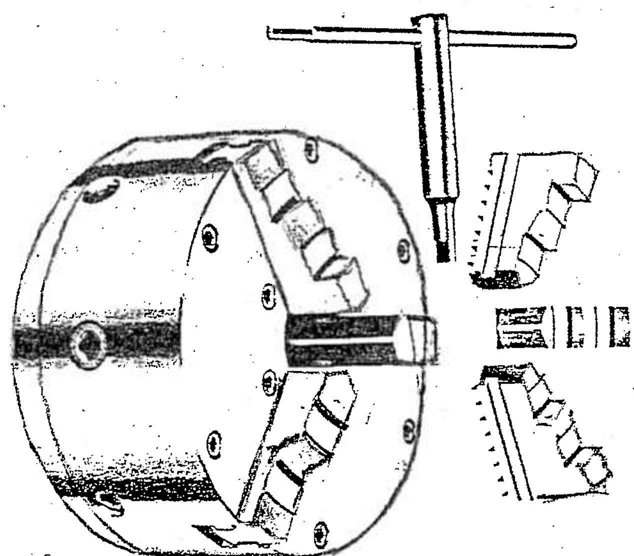
1. Remove the jaws by unscrewing them out. Turn a socket which will turn the scroll plate to drop off the jaw.

2. Turn the socket so the start of the scroll plate begins to show in slot number 1.

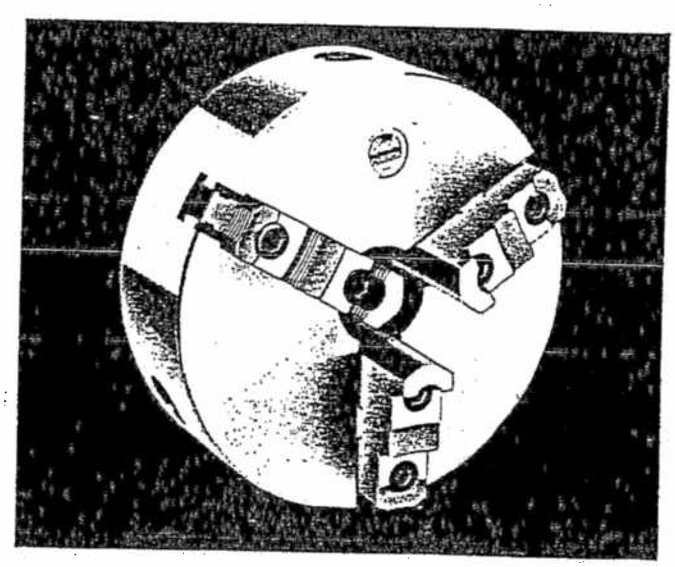
3. Insert jaw number 1 into slot number 1. Turn the socket so it catches the jaw and pulls it in.

4. Turn the scroll plate until the start of the scroll plate begins to show in slot number two. Repeat the above for jaws 2 and 3.

5. When done, bring all three jaws together to verify that you put the jaws in correctly and they meet in the center.



Universal three-jaw chuck (Adjust-tru®) with a set of outside jaws (Buck Tool Company).



Universal chuck with top jaws

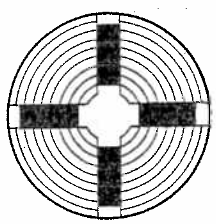
SETTING UP THE 4-JAW CHUCK

Of all the work holding devices, the four-jaw chuck is the most versatile. It holds round, square, rectangular, oval, odd shape work both on and off the spindle centerline. Rules 1, 2, 3 and 4 of the 3-jaw chuck also apply to the 4-jaw chuck. To reverse the jaws from OD to ID holding, unscrew the jaw from its socket, clean it out, reverse it and screw it back in. Simple.

Aligning the Work

Most people have a difficult removing eccentricity from the 4-jaw. It really is quite easy if you follow procedure. When the part is chucked and spun in the spindle, it wobbles. This is known as runout or eccentricity. Half of the runout is only on one side of the spindle's centerline. Think of it this way. An engine crankshaft moves a piston up and down because the piston connecting rod is connected to one side of the crankshaft. As the crankshaft rotates, it moves the piston up and down, let's say 3" total. That means that 1 ½" of the travel is on one side of the centerline. When the eccentric rotates about the spindle centerline, it revolves around a circle of 3" diameter. Move the 1 ½" to the center and you have eliminated eccentricity or runout. Aligning the work in a 4-jaw is to remove eccentricity.

Look at the front face of the chuck. Notice the rings or circles. These are called concentric rings. (Concentric means that the centers of all the circles lie on the exact same centerline.) We use these to "rough in " the workpiece. A dial indicator is needed to finish the task.



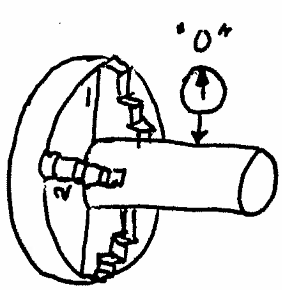
All jaws line up with the same ring on the face of the chuck

Notice also, that the jaws are numbered. Note that jaws 1 and 3, 2 and 4 are opposite each other. This is important.

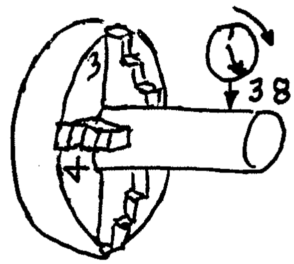
First of all, put the spindle in neutral. Begin the set up by chucking the work. Adjust all four jaws so the part looks centered by using your good "eyeball". You may have to loosen then tighten opposite jaws to move the work around so the work appears equally distant from a concentric line. You don't need to be perfect or exact here.

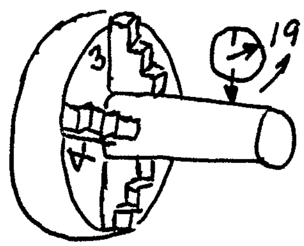
Set up the dial indicator and finish the process.

Set up the dial indicator so the stem is close to being on center. The illustrations show the indicator in a vertical position. This is for illustration only. In actual use, the indicator is in the horizontal position.



1. Rotate the chuck by hand to find the lowest reading. Set the dial's bezel to "0". Note the chuck jaw number. We'll assume jaw #1.
2. Put your finger in the socket for jaw #3. This is to make sure you use the right jaw. Rotate the chuck 180° so jaw #3 is at the indicator.





2. As you rotate the chuck, note the following:
   A; the direction of needle travel. This is IMPORTANT!
   B; The amount of needle travel.

3. We need to move the piece BACK towards half of .038". Loosen #1 jaw slightly. Put the wrench in the #3 socket and push the piece so the needle moves to the "19" reading. (.038/2=.019) Make sure the needle travels back towards the direction it came from to the "19" reading. Rotate the work. Jaw #1 should read "19". If its close, its OK for now.

4. Repeat the process for jaws 2 and 4. Rotate the work from jaw 1 to jaw 2. If the reading is not 19, adjust as before.

5. Reset the dial to "0" and fine-tune the centering process. After a while and once you understand the process, this only takes a few minutes.

Reviewing,

1. Set the dial indicator and find the low reading.
2. Set the indicator from a known jaw and rotate the work 180°.
3. Move the part so that the needle RETURNS TOWARDS the zero by one-half its travel.
4. Verify that the needle has traveled one-half the runout amount.
5. Repeat for the next pair of jaws.
6. Tighten all jaws. Verify the reading is zero runout all around.

LATHE OPERATIONS

Metal lathes are capable of many processes. The operations listed in the Notebook utilize most of these described in the machining of your hammer handle. The explanations and tips help you to understand and try these operations.

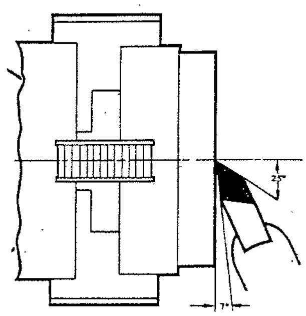
Facing

Facing is machining the end of the work so it is smooth, square to the centerline. Reversing the part end for end and facing it allows us to machine the work to the required length. Facing is usually the first operation in machining a work piece.

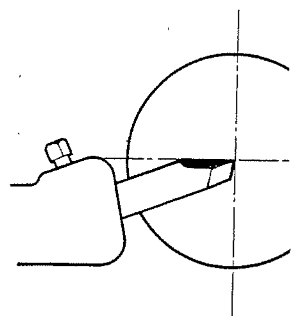
Operation Explanation

1. Set up the work n the chosen work holding fixture.

2. Set the tool so it is A; on center, and B; just the tip of the tool does the cutting. See illustrations.



TOP VIEW



END VIEW

3. Move the carriage so the tool will face about .030" from the end. Set the correct speed. Feed the tool from outside to inside. Hold the carriage's hand wheel with your left hand to prevent the carriage from moving to the right.

4. Use the work diameter as the diameter value in calculating the RPM. If the work exceeds 4" in diameter, use the works' radius as the diameter value in the formula. This is so centrifugal force does not cause the work to come flying out of the chuck. Always think and work safely.

5. The tool can be fed from out to in or in to out. The latter gives a better finish, but you need a center hole to allow the tool feed in from. Either method is acceptable and is a matter of preference.

6. Make sure the tool is on center. If it is above center, the tools' edge will rub at the center. If it's below center, you will leave a "teat" in the middle. Adjust the tool so it is on center. As an aside, you only need to feed to the center. When you reach the center, there is no more material for you to cut.

Turning

The operation of machining an outside diameter, OD, round, straight and to precision size with a single point tool is called turning. When you cut your hammer handle to the .56 diameter, you are turning.

Before turning the work, make sure that:

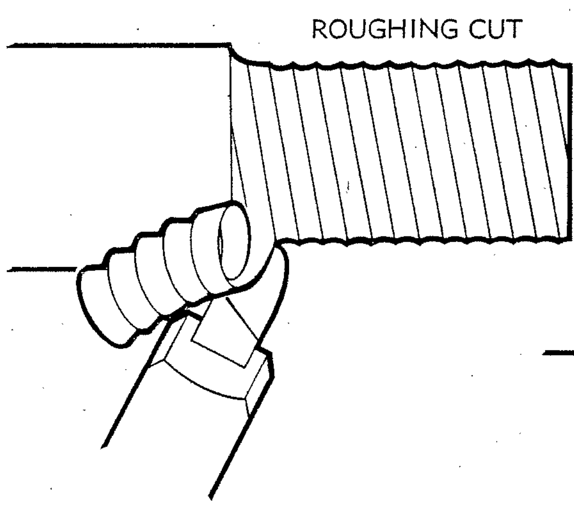
1. the work is securely held in the chuck,
2. faced and center drilled if need be,
3. the work is supported by a live center if need be,
4. the tool is properly ground to shape and sharp and lastly,
5. the lathe is set for the desired speed and feed combination.

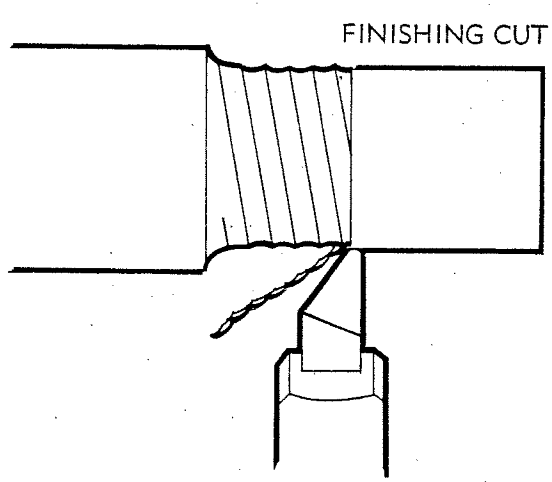
Our goal is to turn to finish diameter with a good surface finish. This is accomplished by tool geometry, the right combination of speeds, feeds depth of cut and type of cut.

Roughing Cuts

Rough turning is used to remove as much material as possible in the least amount of time. Surface finish is not important. From your speed and feed calculations:

RPM= (4 x 100)/3/4D = 533 RPM
The tool has about a .015" TNR. Use somewhere around that number for roughing feeds.
Subtract the .56D from .75D and the difference in diameters is .19. Divide by 2 and the required DOC (per side) is .09. The DOC should be between .030 and .045 deep or .060 to .090 on diameter. We need to leave about 1 TNR, .015" per side, for a finish cut. We will use two roughing cuts at .030" deep each. That will leave about.030" for a finish cut. The point is that a machinist has to plan ahead on the required cuts so there is adequate material for a smooth and precise finish cut.





NOTE:
Any work extending more than 3 lengths of its diameter needs to be faced, center drilled and have its end supported with a live center.

Example: a 1" dia. rod cannot extend more than 3" from the face of the chuck without a live center at its end.

Finish Cuts

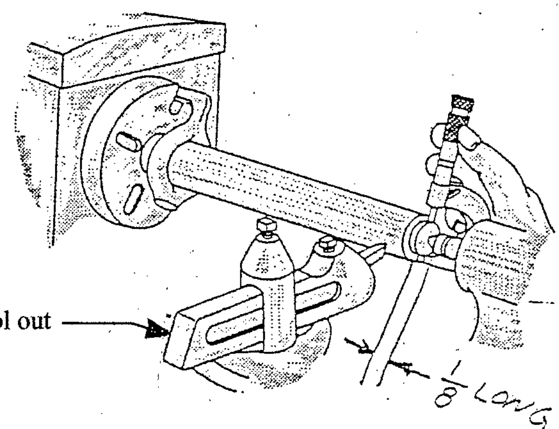
A finish cut produces the round, straight and precise diameter along with the finish as required from the drawing. Since this is a finish cut and the diameter is smaller, we need to make adjustments. The diameter should be near.600". Our RPM needs to be near 666 RPM. Our feed rate in now slowed to ½ the TNR, or to.007 IPR. We should have a depth of cut somewhere near .020". Reset the lathe to values close to these.

Trial Cuts

The trial cut verifies that the lathe tool is cutting to exactly the diameter we want. This cut is THE MOST IMPORTANT when finish turning. If you understand and master this concept, you will make precise parts consistently.

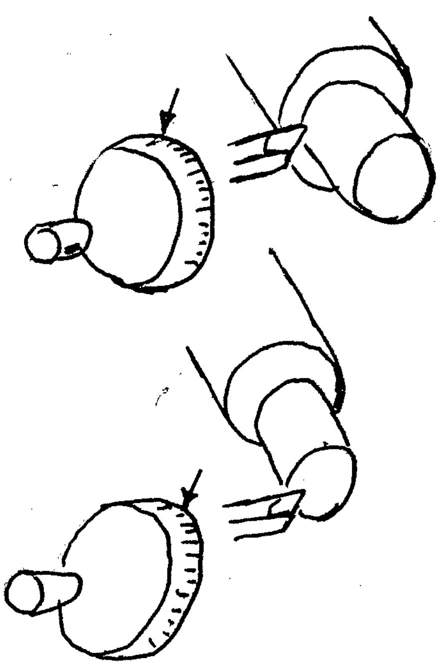
Procedure

1. On the finish pass, turn the lathe on. Move the tool in the needed DOC (remember, DOC is always a radius value).
2. Move the tool and take a 1/8" long cut. Shut off the spindle and feed.
3. DO NOT MOVE the tool out towards you. LEAVE IT at the setting. Using the carriage handwheel, bring the tool to your right, off the end.
4. Measure the diameter. If it is within tolerance, proceed as set up. If not, adjust accordingly. Beware of BACKLASH! Explained later.



DO NOT move the tool out as shown.

Turning Explanation



1. Turn the lathe on. Bring the tool so the tip just contacts the OD. Note the micrometer dial reading and bring the tool off the end to your right.
2. Feed the tool in about .025", but to the nearest even dial number, say .160. This is to make adding DOC to your present dial reading easier, math wise.
3. Turn the OD to length.
4. Stop the lathe. Return the tool to the right. DO NOT move the tool out towards you!
5. Take a micrometer reading. Feed the tool in. Take another cut. Repeat the procedure.
6. On finish cuts, change the RPM, feed rate. Feed the tool in. Take a trial cut. Measure. Adjust. Remember backlash. Continue.

BACKLASH

If your trial cut verifies that the tool is cutting to tolerance, continue on. If the diameter is too large (oversize), feed the tool in the required amount and take another trial cut. The trail cut must be on the full depth of cut. Cutting pressures vary on the amount taken. If your trial cut reveals that the diameter is undersize (too small), you need to readjust the tool and backlash.

When two parts mate together, one needs to be smaller than the other. When you screw a nut onto a bolt, the nut moves freely. This is called fit. On machine this is called "backlash".

If your cut is undersize by .002", you can't back off the .002" on the micrometer dial. What you'll do is just remove the mating surfaces of the nut to the bolt and the carriage will float around during the cut. If the cut is .002" undersize, what do you do?

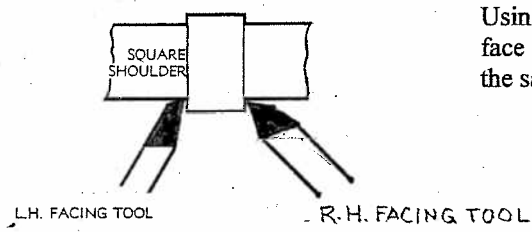
First of all, look at the micrometer dial setting. REMEMBER THIS NUMBER. Turn the dial so the tool BACKS OUT towards you by at least half a turn. This puts backlash into the nut. Now, turn the dial and stop the infeed .002" before you reach your old reading. You've removed backlash from the nut. Take another trial cut.

Rough cuts, finish cuts, trial cuts and backlash are major concepts in lathe work. Understanding feeds and speeds, depth of cut and cutting forces and their working relationship with each other are important. The coordinating of all these variables may seem huge, but after you've done it a few times and think how all these variables interact, it will come easy to you.

SHOULDER WORK

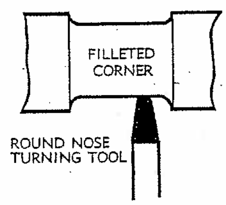
A shoulder is defined as the intersection of two diameters whose centers lie on the same centerline. Basically, there are four types.

Square



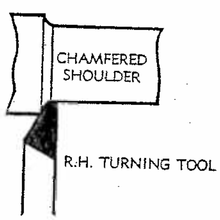
Using a facing tool, bring the tool to the desired length and face out. The shoulder is square to the centerline and a corner the same size as the tool's radius.

Filleted



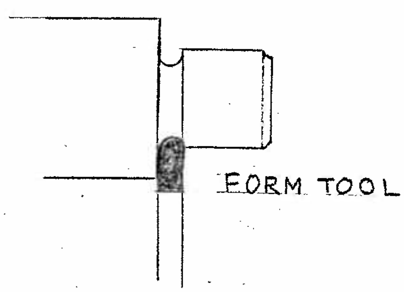
Grind a form tool having the required radius. Bring the tool to the corner and face out. Note that as you turn, you need to plan ahead and leave material for the fillet. Otherwise, your corner will be too far, the shaft too long and probably scrap.

Chamfered



Grind a tool to the correct angle and front clearance. Use a gage and set the tool to the correct angle. And feed to the required distance. Again, leave material for the chamfer.

Undercut

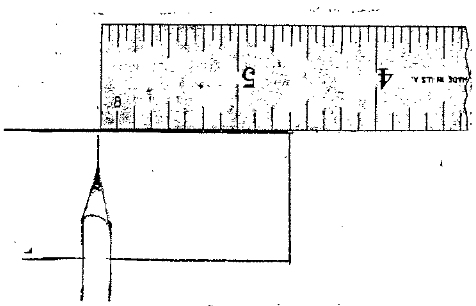


An undercut is needed for a piece to fit squarely to the shoulder or to have a groove for a threading tool to end its cut in without crashing.

Marking the Shoulder

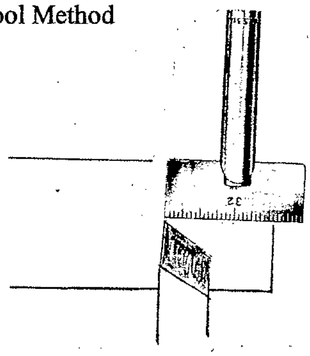
Generally, a shoulder needs to be turned to a length. If not accurately needed, several methods can be used. If the distance must be precise, use the compound rest.

Scale/pencil Method



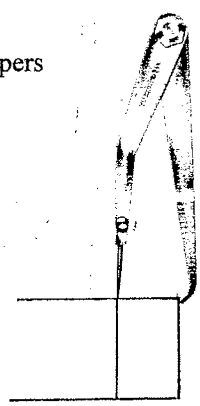
Use a scale and a pencil to mark the steel to the distance needed. Turn the lathe on and scribe the OD. Not very actuate, but may be adequate for the job.

Scale/tool Method



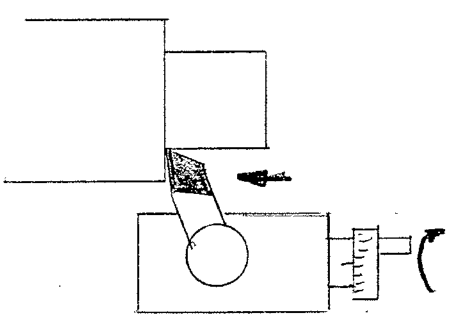
Using a scale so the distance is placed on the end, move the carriage so the tools tip lines up with the end of the scale. Turn the lathe on, scratch the surface. This is your stopping point

Hermaphrodite Calipers



Using a scale set the hermaphrodite calipers to the required length. Use layout fluid (Dykem) or a Magic Marker, ink up the circumference where the shoulder ends. Turn the lathe on, place the bent leg against the end and use the point to scribe the work.

Compound Rest Method



Set the compound rest to 90°. use the dial, turn the tool so it touches the end. Set zero. Feed the tool to distance. There are a few traps here.

1. The carriage can move back as you cut. Lock the carriage. Don't forget to unlock it!
2. You could cut a taper, as you don't have the compound rest set at EXACTLY 90°. Take a skin cut, measure at each end and adjust to remove the taper.
3. Check the micrometer dial. It could be either an indirect or direct dial. If an indirect dial and your shoulder needs to be ½", feed the tool travel ½". If it's a direct dial, feed

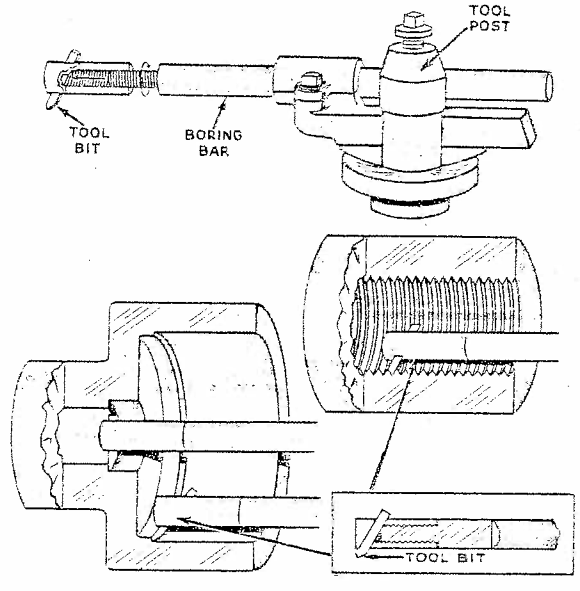
BORING

A drilled hole or a cast hole is never round or straight. To make a hole, called a bore, round, straight, and to precision size, we machine it with a single point tool. It is machining the internal diameter, or ID. The bore will also run true with its centerline.

The work is generally held in a chuck. The boring bar is setup to machine a straight or tapered hole, internal threads or other profiles. Boring is a rather difficult, exact and challenging process. It is also very difficult cut to exacting dimensions and difficult to take precise measurements.

Boring Rules

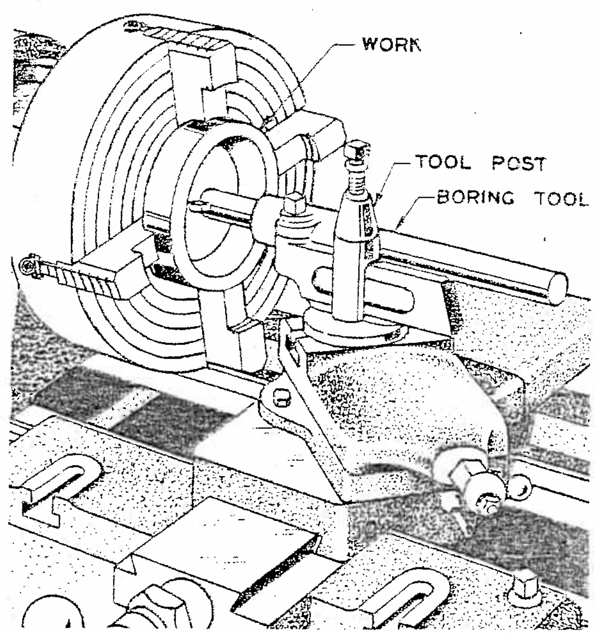
The boring bar must be as large as possible in diameter to bore the hole. It also must not extend out more than necessary beyond the holder. As the tool is hung way past its holding point, it is, in effect a cantilever beam. The tool has a habit to spring away and down from the work. As the tool is backed out of the cut, there is no longer a strong cutting pressure, so the tool returns to its normal position and back cuts on its way out. The rule of thumb is a boring bar must not extend than four of its diameter beyond the tool holder. That means a ½" diameter bar should not extend more than 2" beyond its point of being held. A one-inch bar should not extend more than 4 inches. Deflection, or bending, increases with the cube of the overhang. Clearly, to get a precise cut, rigidity is paramount.



Boring Tool

The tool must be ground as a "left hand" cutting tool. That means that the cutting edge is on the right corner of the tool bit. (Left hand tools cut from left to right.). The tool's cutting edge must be set on center. Below center, the tool will tend to chatter and not cut to precision size. The tool must also have to have more front clearance. Above center, the tool will have a tendency to rub on the newly machined ID. The tool must not rub against the ID. Therefore, the tool has a lot of front clearance.

The boring bar must also be parallel to the horizontal with the ways. It must also be parallel to the centerline of the lathe's axis. Make a mark on the bar or use a stop to signify the end of the internal cut. If you are boring a blind hole, this is important as the tool will crash into the bottom of the bore.



Boring

Boring Tool Details

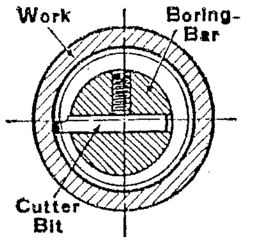


Fig. 83. Application of Boring Tool

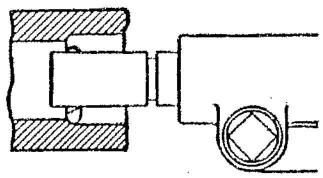


Fig. 84. Detail of Boring Tool

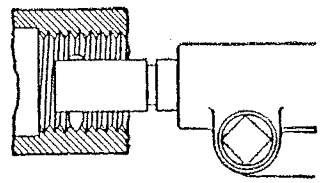


Fig. 85. Inside Threading Tool

Note that when the tool is machining the bottom of a blind hole, the tool's cutting edge must be ahead of the boring bar.

Speeds and Feeds

Speeds and feeds are calculated in the usual manner. However, if the tool "chatters" (vibrates) the rule is to increase the feed. Another thing to try is to have a deeper depth of cut. The next thing to try is getting a more rigid boring bar, or a combination of these solutions.

The given illustrations will help you set up the tool.

ID Measuring

Precision measurement of internal dimensions can be a challenging affair.

Telescope gages are spring-loaded tubes that expand into a hole. When the maximum diameter is "felt" by the machinist, the tubes are locked, removed and measured over with a micrometer. Subjective at best.
A vernier gage can also be used. It does not provide a precise measurement.
A bore gage is a dial indicator set up to measure a diameter. It only shows by how much under the bore it. It can only measure within a small range.
A bore micrometer has three legs that open to touch the ID. It provides precision measurement within its range.

In closing, refer to a good machine theory book for more information or ask a machinist.

THREADING

Threading is the operation of cutting a constant helical groove along a cylinder. The shape of the groove is dependent upon the thread style. Usually, the threads' included angle is 60°, with a slightly rounded tip.

Thread is intimidating. There are a lot of steps to remember. It requires good hand-eye co-ordination with the cutting action and a detailed setup. If you've never cut a thread before, it is a good idea to take a few practice passes by cutting air to get the feel for the co-ordination needed. Let's cut the 3/8"-16 thread. A side note, the process explained is for a "right-hand" thread. A "left-hand" thread set up is pretty similar.

Work piece

The work OD must be turned to thread size. This is generally .005" to .010" undersize. The 3/8 diameter needs to be 0.3737 to 0.3643 diameter. This is to make sure the thread doesn't interfere with the ID of the nut. Also, a groove, or undercut, must be cut. This is so the threading tool has a place to stop without crashing. The groove also provides a stress concentration point to actually make the part stronger.

Additionally, the work must have NO runout. MOST IMPORTANTLY, you CANNOT remove the part from the machine until the thread is done. If you, it becomes difficult to realign the tool so it picks up the thread.

Machine Set Up

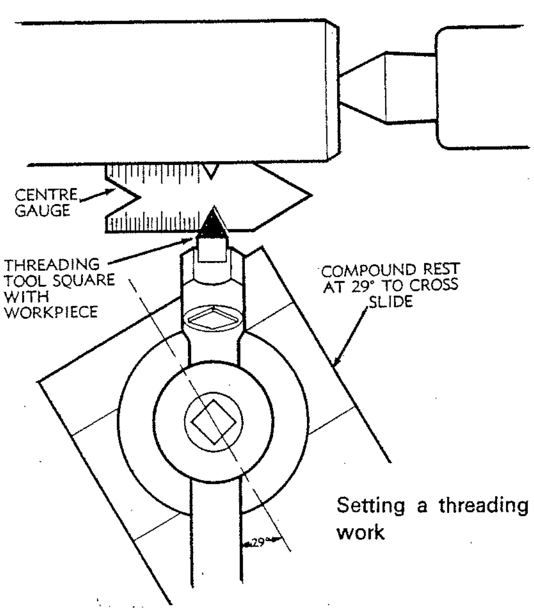
Set the RPM to about 100 RPM.
Set the Quick Change GearBox to cut 16 TPI.
Set the compound rest to 30° to the right. (29° to 30° is OK)

The Tool

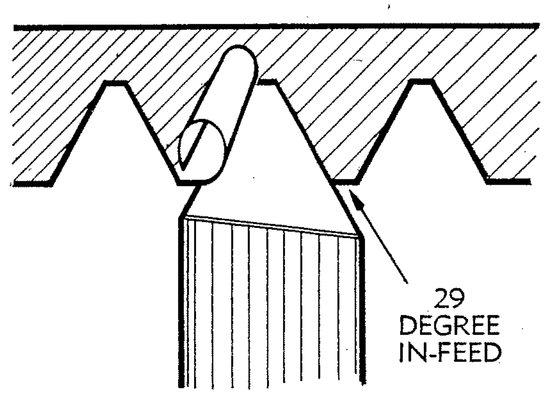
The tool must be ground at EXACTLY 60° angle. Also, the sides must have clearance. You don't want the sides to rub on the thread side. The tool has no back or side rake, though if you know what angle the tool sits in the tool holder, you can grind the top to match this angle. This will make the top parallel to the lathe's horizontal plane. Using a center gage, grind the tool so that no light shines between it and the included angle. When ground, stone a little tool nose radius.

Set the tool properly. The tool's 60° included centerline must be perpendicular to the diameter. Using the center gage, adjust accordingly. See the illustrations.

The tool is fed into the work with the compound rest. Only the leading edge of the tool cuts.



Setting a threading tool square with the work

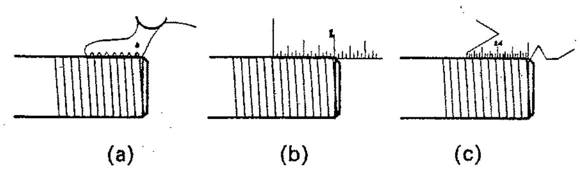


Tool fed at 29 degrees

Cutting the 3/8"-16 Thread

The last step in the set up requires that you orient the tool to the micrometer dials and the OD, and set the dials to zero-zero. (Similar to what you did when turning.)

1. Back out the compound rest, (CR) two or so turns. Turn it back IN about ¼ turn. Set the dial to "0".
2. Turn the machine on. Feed the cross-slide, (CS), until the tip just touches the OD. Set this dial to "0".
3. With a pencil, mark the CS and the saddle with a reference mark. This is to know which CS "0" orientation to stop at.
4. The tool is positioned to the right off the work. With your LEFT hand on the CS dial and your RIGHT hand on the split nut lever, engage the split nut when the chaser line and the reference line lines up. Keep your hand on the split nut lever. Note that the chaser dial has 4 or 8 lines on it. There are lots of reasons for this. Suffice to say that if you want to make this easy for yourself and not ruin the thread, pick a number and use only that number.
5. At the end of the cut,
   A. Back the tool out of the groove by at least a half turn, while,
   B. At the SAME TIME, lift the split nut lever to stop the carriage travel. Be quick with this action.
6. Bring the carriage to the right at the starting point. Reset the CS to the correct "0".
7. Stop the lathe. Use a scale and check for correct TPI. There should be 16 scratches per inch on the OD.



(a) Checking with thread pitch gauge
(b) Checking with steel rule
(c) Checking with centre gauge

Fig. 10-61. Checking the number of threads per inch

8. Using the compound rest, feed the tool in and take another cut following steps 4, 5 and 6. See the table at the end of this section for suggested depths of cut. The thread cannot be cut in one pass. It requires multiple cuts along with a finish pass. Use cutting oil. This is a must.
9. Repeat the procedure until the thread is fully cut and within tolerance. Check it to make sure and when done, you can remove it from the lathe.

Thread Indicator or Thread Chaser Dial

Mounted on the carriage, this device is used to let the operator know when to engage the split nuts. If the nuts are not engaged at the correct position, the tool will cut over previously cut threads and ruin the work. It makes the tool enters the groove at the same point each and every pass. Generally, but check the lathe manual,

For even threads, like 16 TPI, engage on any line.
For odd threads, like 11 TPI, engage on any odd OR even line.
For special threads, like 11 ½ TPI, pick a line and stick with it until the thread is done.

A safe practice is to pick a number and stick with it until the thread is done. If you've never cut a thread before, this is good advice.

Thread Chaser Dial

Before the chaser dial and the advent of the quick change gearbox, threading used to be an arduous process. Consulting a chart, the machinist would have to change the gears connecting the spindle gears to the leadscrew gears for the pitch he needed to cut. Once that was done, the tool set up, he was ready to cut the thread. He would engage the carriage to the leadscrew with the split nuts and leave them engaged until the thread was done. The cut was started. At the end of the cut, the tool was withdrawn, the machine stopped and put into reverse. The tool traveled to its start point where the machine was shut off. The machine was then put into forward, the tool fed in and the process repeated as many times as necessary until the thread was finished. Threads are still cut this way if your lathe does not have either of these accessories.

The Quick Change Gear Box

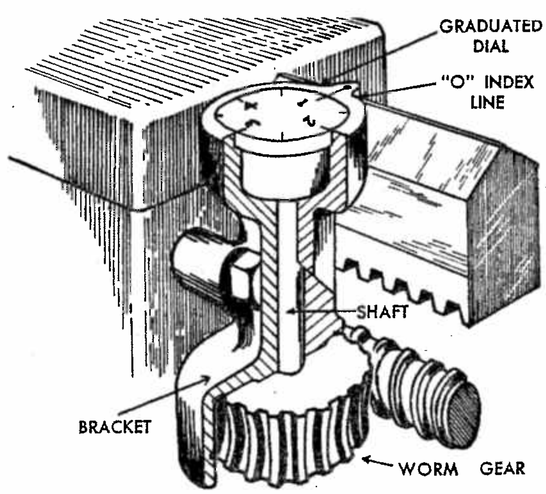
The QCGB is basically a transmission with all the proper gear ratios needed to cut threads. The machinist selects the pitch, moves the levers to the appropriate locations and engages them. This replaces the whole change gear process.

The Chaser Dial

Without the chaser dial, the machinist must engage the split nuts to the leadscrew and leave them engaged for the duration of the operation. That means reversing the spindle at the end of the cut, go back to the start point etc. as explained above. You need to be quick. If he shaft has a shoulder on it, and you crashed, you have to reset the tool and pick up the thread and follow a different procedure in order to ensure a good thread. A real pain if you're not paying attention.

The chaser dial prevents all of that. To start the cut, engage the split nuts when a line on the dial aligns with the reference line. At the end of the cut, disengage the split nuts, return to your start point, feed the tool in, wait for your selected number and reengage the split nuts to repeat the process. No stopping or reversing the spindle and all. Simple, effective and safe. Prevents a lot of scrapped work. Again, refer to the lathe's instruction manual or a good shop book to get a better understanding of the procedure.

Lastly, as previously stated, to make sure that the tool will pick up the same start point, pick a number and always engage the split nuts at this number.



Thread dial indicator

Depth of Cut

Note that the tool is fed in at a 29 ½° angle. This makes the front leading edge of the tool do the entire cutting. The question remaining is the depth of cut. Looking on the center gage, you'll find "Double Depth of Cut of American National Threads". Find 16, for 16 TPI. The DOC is .081". On a direct reading dial, the .081" is the total infeed amount. On an indirect reading, the total depth is half, or .040". This is why you need to know if the compound dial is direct or indirect.

Suggested in-feed depths per pass at 29° for a 16 TPI thread:

Direct In-feed			Indirect In-feed		
# of cuts	Depth of cut	End Dial reading	# of cuts	Depth of cut	End Dial reading
6 cuts @	.010 each	60	6 cuts @	.005 each	30
3 cuts @	.005 each	75	3 cuts @	.003 each	39
2 cuts @	.002 each	79			
FREE PASS. Check thread.			FREE PASS. Check thread.		

A free pass is to reset the tool to its last setting. Recut the thread. This "cleans up" the thread and removes any missed material. Feed the tool in .001" from the CROSS-SLIDE and take a final cut. The tool moving in at the 29° angle actually leaves the backside "stepped". This final cut smoothes the backside of the thread form. If necessary, use .001" cross-slide infeed to finish size if the thread is still oversize.

When cutting other thread pitches, the most practical DOC is .010" (.005") per pass. After 75% or so of the depth is reached, use about half of the DOC (.005) to get to final depth. The threading is a forming tool and doesn't cut efficiently or cleanly. Also, the material is not being cut at its best surface speed. This is the reason why DOC's are taken in steps.

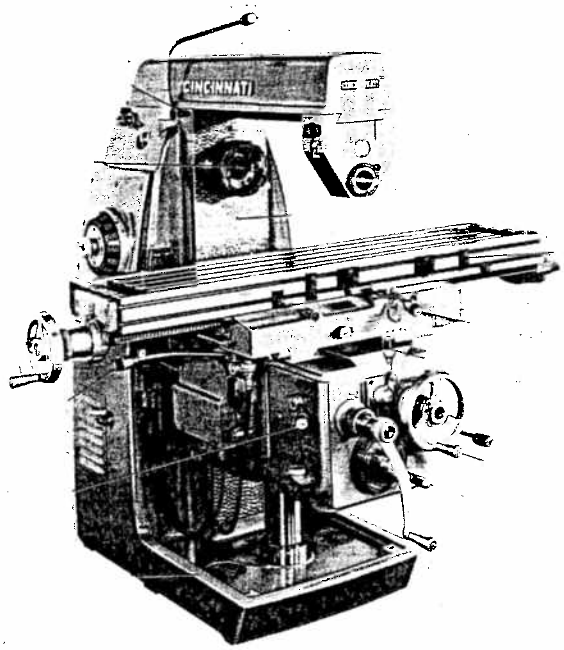
Thread Checking

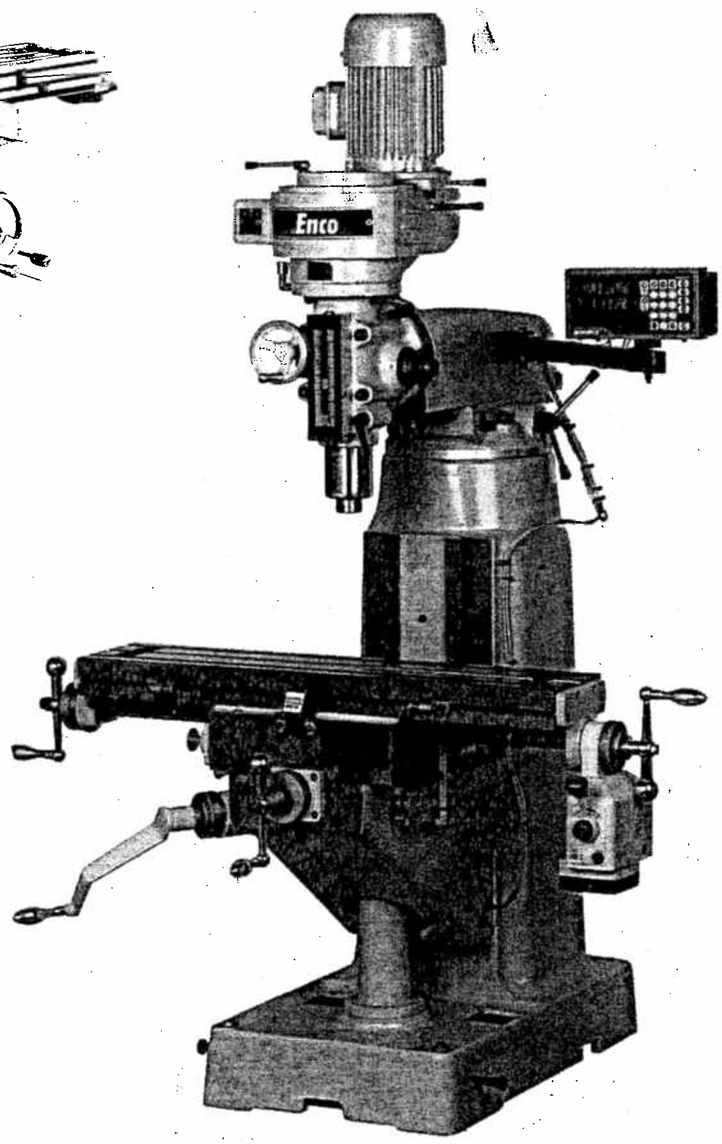
There several methods for checking thread fit. These are:

1. Mating part; screw the mating part onto the thread. If it fits smoothly and freely, it's OK.
2. Nut; if the nut screws on freely, it's OK.
3. GO/NO-GO Gage: the Go part assembles and the NO-GO end does not, the thread is OK.
4. Pitch Micrometer: a special micrometer that measures the pitch diameter. If the pitch diameter is within tolerance, the thread is acceptable.
5. Three-wire Method; using three wires, a regular micrometer and some math, this is a most effective and cheap way to measure pitch diameter.

The above are major thread inspection methods used in manufacturing. The section on threads goes into greater detail on this subject.

MILLING MACHINES





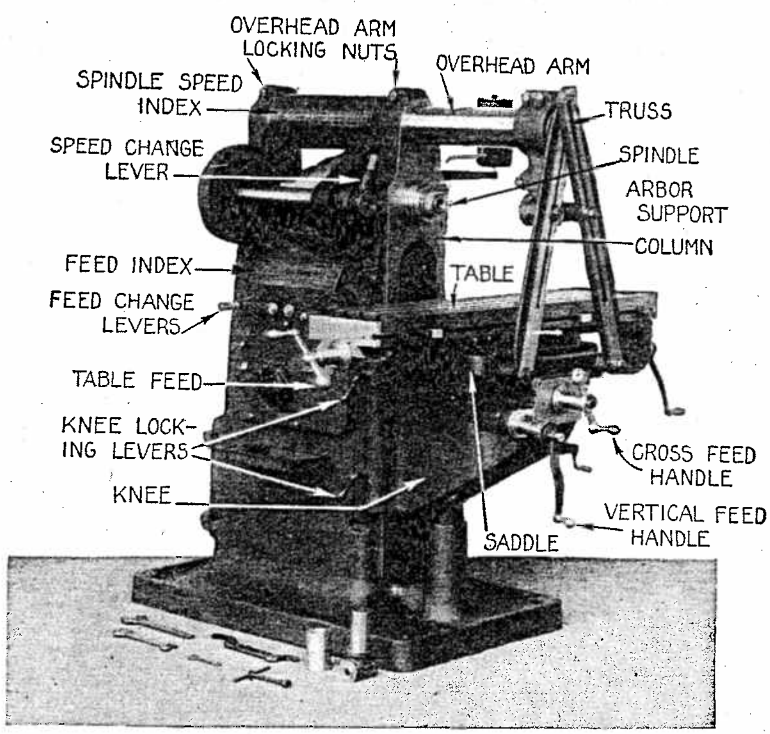
HORIZONTAL MILLING MACHINE

The illustrations on the following pages refer to a horizontal milling machine. The basic machine design for both the vertical and horizontal are similar. The major difference is that the cutter is mounted on an arbor going left and right instead of directly in a holder. The cutter is keyed, or locked, so it won't slip in the cut.

When mounting an arbor cutter, you need to add spacers on the arbor to take up space between the cutter and the arbor ends. You also need a key to drive the cutter. After the arbor is mounted in the spindle and the nut just snugged, the overarm and yoke, or outboard bearing and support must be locked in place. Once this is done you can finish tighten the arbor. This is to prevent bending the arbor.

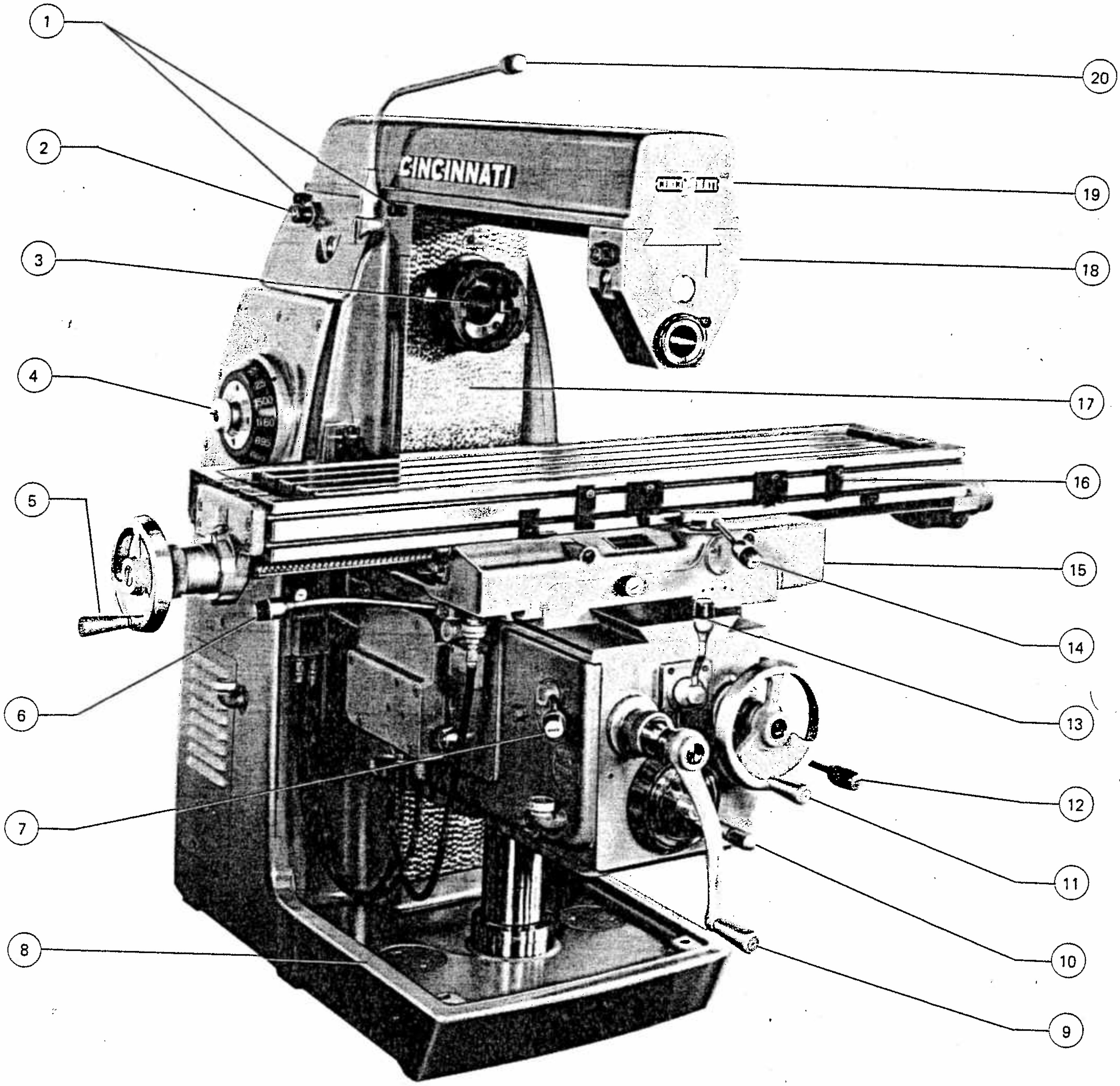
As you cut, you still need to use the speeds and feed rules as given. Remember the climb/conventional milling directions.

These machines are as small as bench top machines to behemoths that can hold work pieces weighing several hundred pounds.



Cincinnati Miller Universal Cone Drive
Courtesy of Cincinnati Milling Machine Company, Cincinnati, Ohio

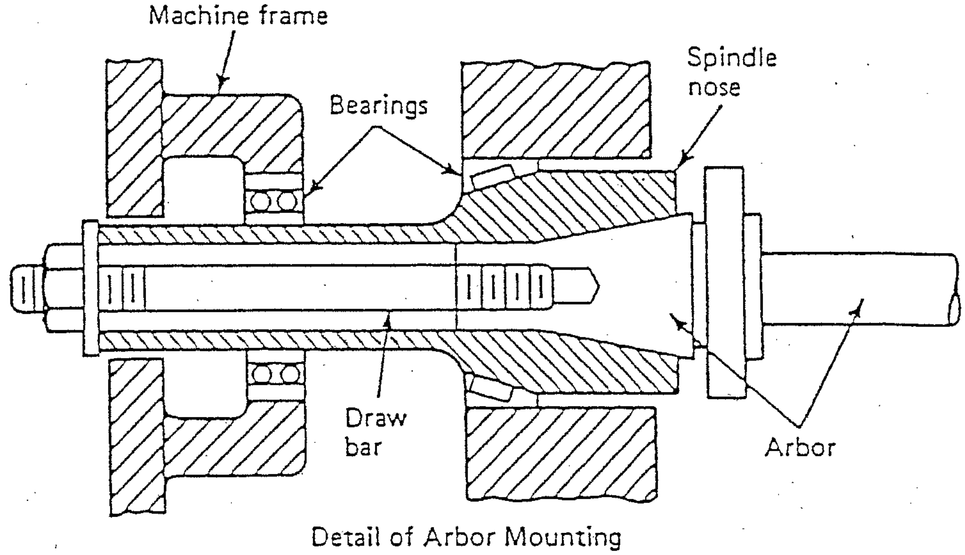
HORIZONTAL MILLING MACHINE

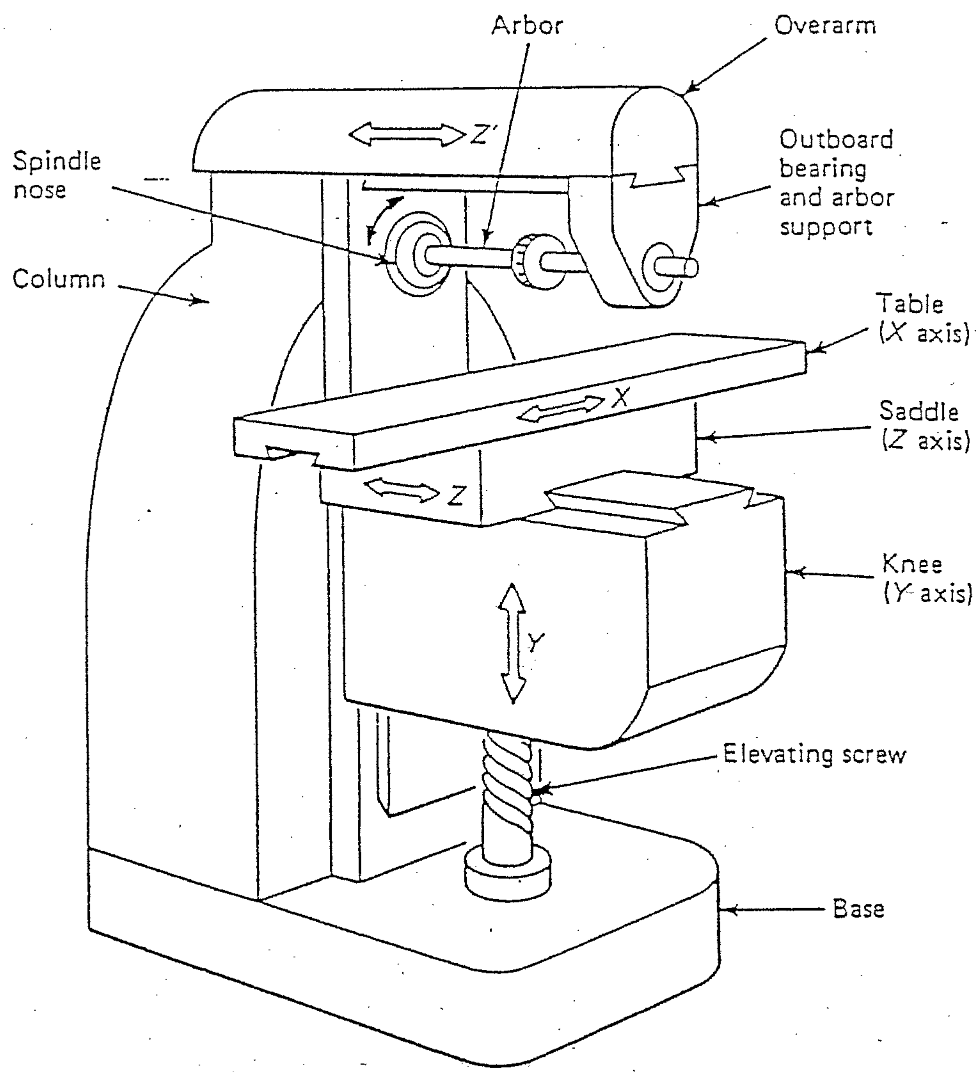


1 OVERARM CLAMP NUTS
2 OVERARM POSITIONING SHAFT
3 SPINDLE
4 SPINDLE SPEED SELECTOR DIAL
5 TABLE TRAVERSE HANDWHEEL
6 POWER TABLE FEED LEVER
7 POWER VERTICAL FEED LEVER
8 BASE
9 VERTICAL FEED HANDCRANK
10 TABLE FEED SELECTOR CRANK
11 CROSSFEED HANDWHEEL
12 RAPID TRAVERSE LEVER
13 POWER CROSSFEED LEVER
14 LONGITUDINAL FEED LEVER
15 SADDLE
16 TABLE STOP
17 COLUMN
18 ARBOR SUPPORT
19 OVERARM
20 SPINDLE START LEVER

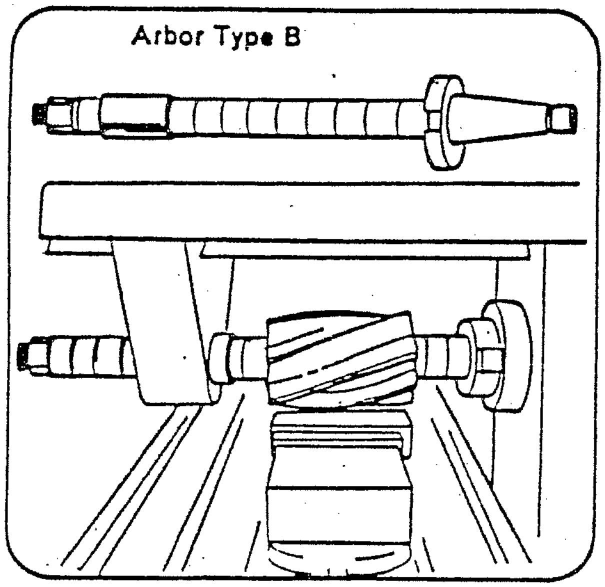
Courtesy of Cincinnati Milacron

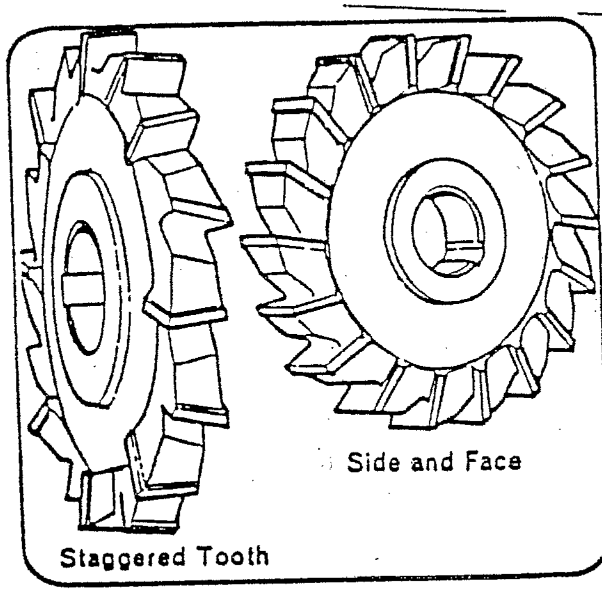
Fig. 12-2 Schematic drawing of a horizontal-spindle knee-and-column-type milling machine.



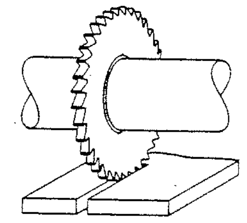
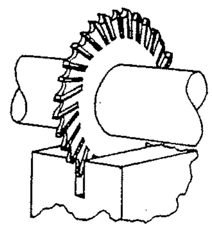
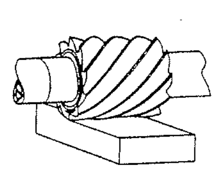
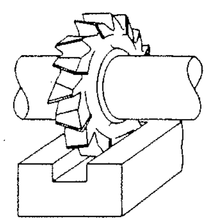
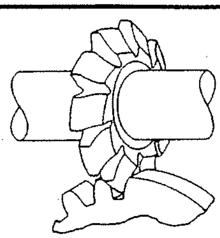
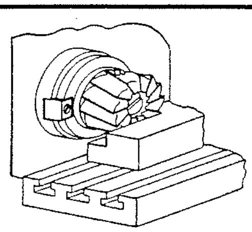
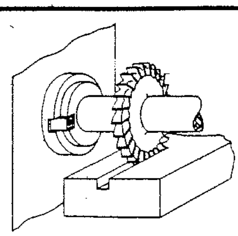


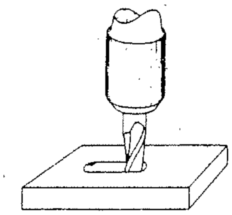
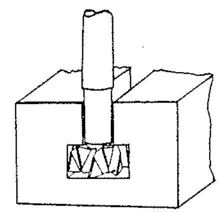
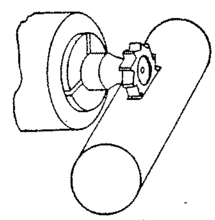
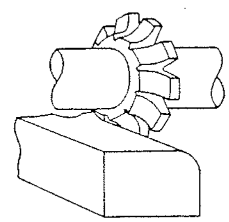
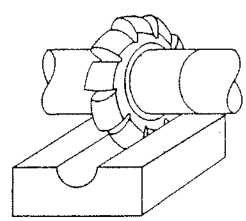
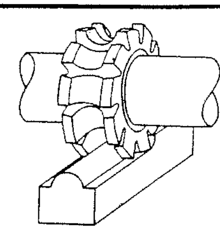
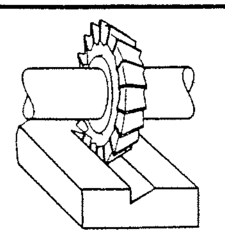
Arbor-mounted cutters: Horizontal mill





MILLING CUTTER APPLICATION CHART

H.S.S. CUTTER TYPES		REMARKS
PLAIN SLITTING SAW		USED FOR CUTTING OFF THIN STOCK AND SMALL TUBING AND MACHINING SHALLOW SLOTS. USE SAWS WITH STAGGERED TEETH AND SIDE CHIP CLEARANCE FOR DEEPER CUTS.
SLITTING SAW WITH STAGGERED TEETH AND SIDE CHIP CLEARANCE		THIS SLITTING SAW IS USED TO MACHINE DEEP SLOTS AND FOR SAWING OFF OPERATIONS. THEY ARE RECOMMENDED FOR MACHINING SLOTS 3/16 INCH OR WIDER.
PLAIN MILLING CUTTER		THE HEAVY DUTY PLAIN MILLING-CUTTER HAS A 45 DEGREE LEFT HAND HELIX AND IS USED TO MAKE HEAVY SLAB AND FACE MILLING CUTS ON MILD STEELS.
STAGGERED TOOTH SIDE MILLING CUTTER		THIS MILLING CUTTER IS DESIGNED FOR DEEP SLOTTING OPERATIONS IN STEEL. ALTERNATE RIGHT AND LEFT HAND SPIRAL TEETH PROVIDE AMPLE CHIP ROOM FOR FAST STOCK REMOVAL.
GEAR MILLING CUTTER		THE ARBOR TYPE GEAR CUTTER SHOWN IS USED ON A HORIZONTAL TYPE MILLING MACHINE FOR MACHINING SPUR GEARS AND HELICAL GEARS.
SHELL MILLING CUTTER		A SHELL TYPE MILLING CUTTER IS MOUNTED ON AN ARBOR AND IS USED FOR END MILLING PART EDGES, FACE MILLING LARGE FLAT SURFACES, AND MILLING A SQUARE SHOULDER AS SHOWN.
SIDE MILLING CUTTER		SIDE MILLING CUTTERS ARE USED FOR SHALLOW SLOTTING, SIDE MILL OPERATIONS, AND ALSO FOR SHALLOW STRADDLE MILLING OPERATIONS.

H.S.S. CUTTER TYPES		REMARKS
END MILLING CUTTER		THE END MILL IS USED FOR MILLING EDGES OF PARTS, SLOTS, HOLES, POCKETS, AND FACING. THE TWO FLUTE CUTTER IS GENERALLY USED TO MACHINE ALUMINUM AND THE MULTIFLUTE TYPE USED FOR STEEL.
T-SLOT MILLING CUTTER		THE T-SLOT CUTTER IS DESIGNED TO MACHINE T-SLOTS IN TABLES OF MACHINE TOOLS. A SLOT MUST BE FIRST MACHINED TO PROVIDE PROPER CLEARANCE FOR THE CUTTER SHANK.
WOODRUFF KEY SEAT CUTTER		THE KEY SEAT CUTTER MACHINES A CIRCULAR GROOVE IN A SHAFT TO HOLD A WOODRUFF KEY IN POSITION FOR THE PURPOSE OF LOCKING A PULLEY OR DETAIL TO THE SHAFT.
CORNER ROUNDING MILLING CUTTER		CORNER ROUNDING MILL CUTTERS ARE USED TO FORM A PART EDGE TO A SPECIFIED RADIUS. THESE CUTTERS CAN BE PURCHASED IN EITHER SHANK OR ARBOR TYPE CUTTERS.
CONVEX MILLING CUTTER		A CONVEX MILL CUTTER IS USED TO PRODUCE A TRUE CONCAVE RADIUS. THIS CUTTER IS USED TO MILL HALF CIRCLES AND MAY BE SHARPENED WITHOUT CHANGING THE FORM BY GRINDING THE FACE OF THE TOOTH.
CONCAVE MILLING CUTTER		CONCAVE MILL CUTTERS ARE USED TO PRODUCE TRUE CONVEX RADII. THIS CUTTER IS USED TO MILL HALF CIRCLES AND MAY BE SHARPENED WITHOUT CHANGING THE FORM BY GRINDING THE FACE OF THE TOOTH.
SINGLE ANGLE MILLING CUTTER		SINGLE ANGLE MILLING CUTTERS ARE USED FOR MACHINING DOVETAILS AND RATCHET TEETH. THIS CUTTER IS MADE AS AN ARBOR OR SHANK TYPE TOOL HAVING AN INCLUDED ANGLE OF 45 OR 60 DEGREES.

RAM TYPE VERTICAL MILLING MACHINE

The ram type vertical mill was developed by the Bridgeport Milling Machine company and is generally referred as "Bridgeport". The machine is very versatile, easy to use, and adaptable to many milling operations and configurations. It is a favorite of students, machinists, tool and die makers and mold makers worlds wide.

Defined are parts common to both vertical and horizontal mills.

Base/column: This is a rugged one piece casting that is the backbone of the machine. From it all the other major parts are attached.

Ways: There are two sets of ways. One is on the column that the knee moves vertically on. The second is on the knee where the saddle moves in/out in a horizontal plane.

Knee: Common to both styles, this piece moves vertically on the column ways. It also holds the saddle which rides in an in/out motion to the knee.

Saddle: Rides on the knee ways in an in/out motion. It too has internal ways to hold the table.

Table: Rides in the saddle ways in a left/right motion. The table holds the work, vise or other tooling to allow the work to be machined.

Parts common to Ram-type mills.

Ram: Holds the tool head and can be positioned in/out to a desired location. It can also be swiveled 180°. Attachments can be mounted on its' back tongue.

Toolhead: Consists of several parts. It can be tilted front or back 45° and tilted left or right 90°. This produces simple or compound angled cuts.

Motor: mounted on the top of the toolhead, it provides the power to turn a cutting tool. It can be reversed and its speed can be changed via V-belts or a variable speed drive.

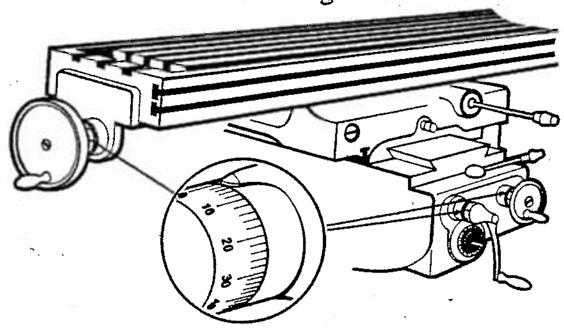
Quill: Mounts inside the toolhead. Connected to the motor, it allows the spindle to spin and lets the spindle move up and down. A clamping cam allows it to be locked in a down position.

Spindle: Mounts inside the quill. This is the device that holds the cutting tool and spins inside the quill. It holds tools firmly inside it with a drawbar.

Drawbar: Mounts inside the quill from the top. When screwed into a collet or tool, it pulls the tool tight into it to provide the necessary holding force to drive the cutter into the work.

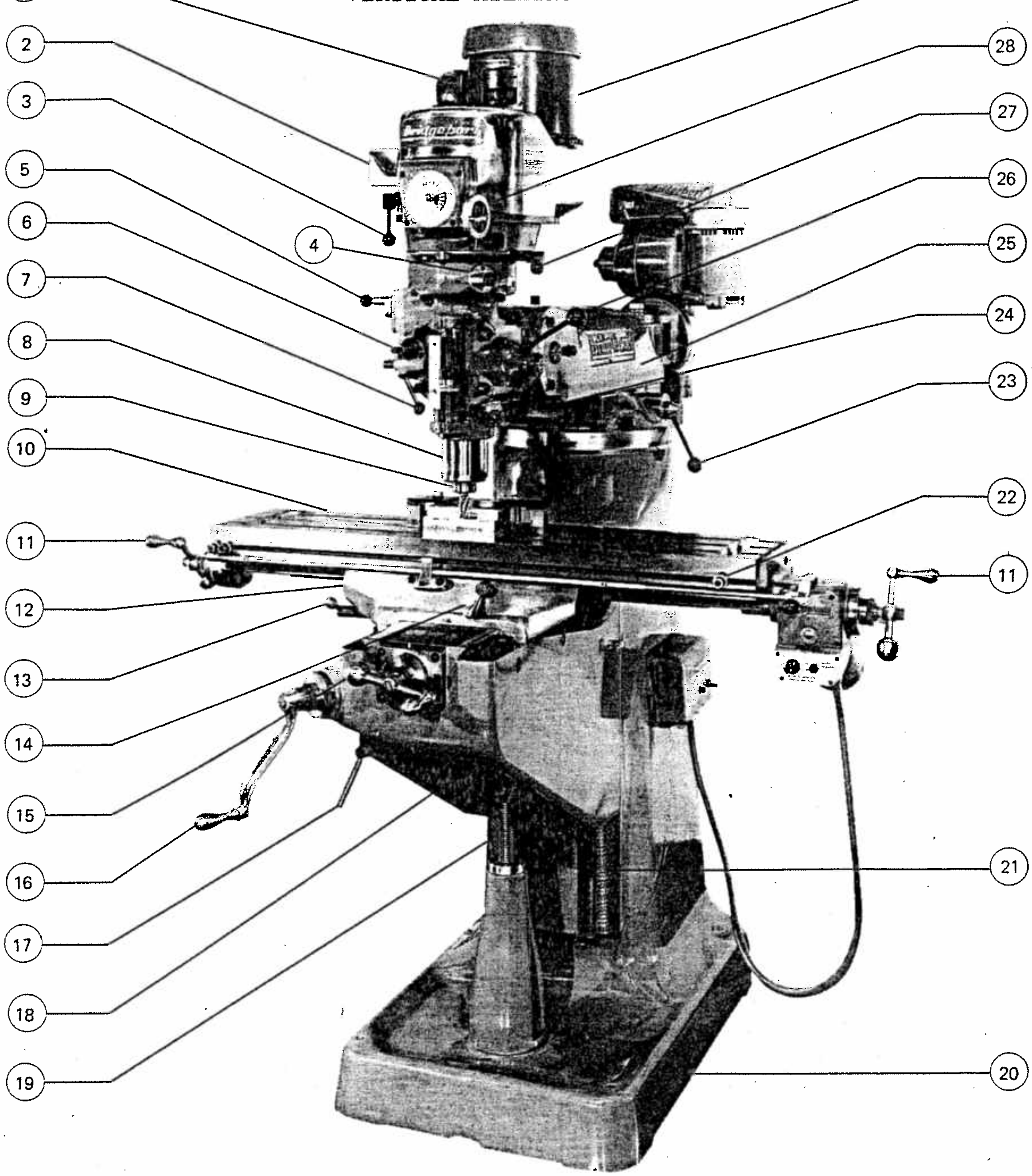
Micrometer Dials

These are mounted on leadscrews to precisely position the knee up or down, the saddle in or out and the saddle left or right. The dials are graduated in .001" increments. A position movement of 1.100" on the dial moves the knee, saddle or table exactly 1.100". these are direct reading dials.



Graduated dials

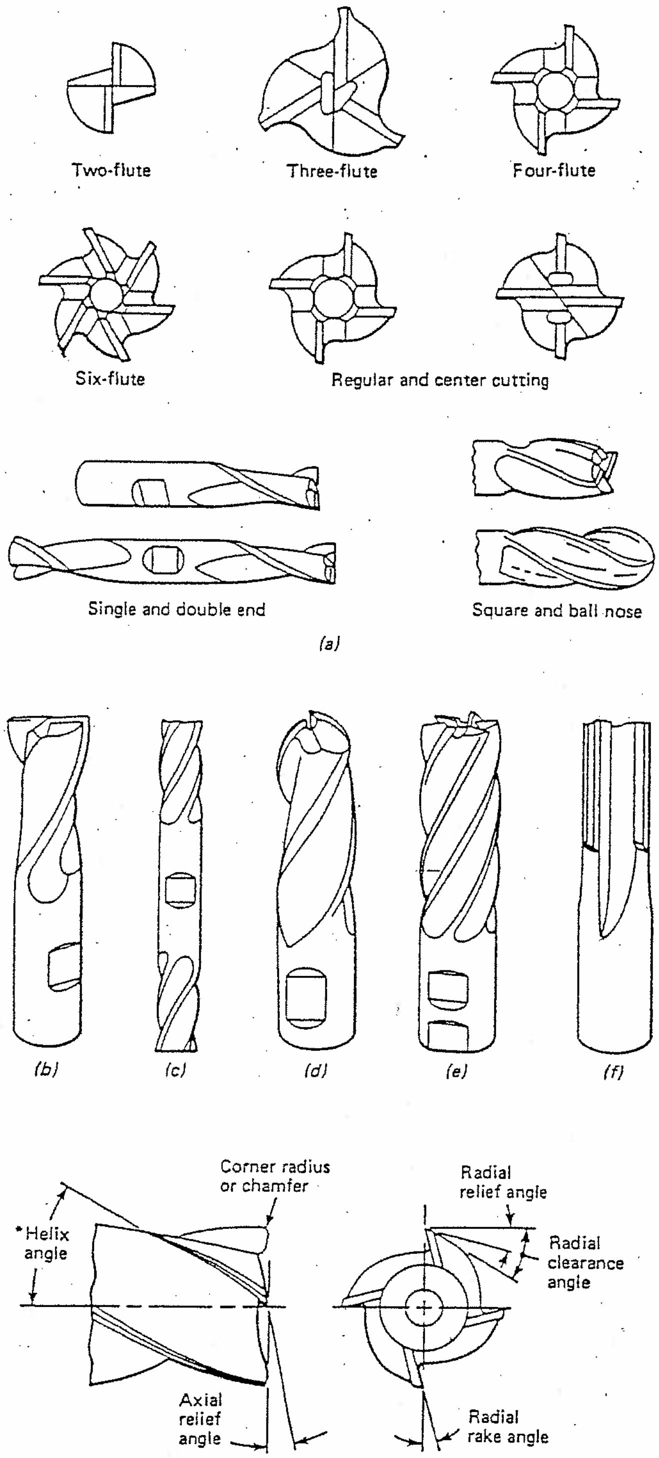
VERTICAL MILLING MACHINE



1 DRAW BAR
2 MOTOR SWITCH
3 SPINDLE BRAKE
4 POWER FEED ENGAGEMENT KNOB
5 QUILL FEED SELECTOR
6 FEED REVERSE KNOB
7 FEED CONTROL LEVER
8 QUILL
9 SPINDLE
10 TABLE
11 LONGITUDINAL FEED HANDLE
12 SADDLE
13 SADDLE CLAMP
14 TABLE CLAMP
15 CROSSFEED HANDLE
16 TABLE ELEVATING HANDLE
17 KNEE CLAMP
18 KNEE
19 TABLE ELEVATING SCREW
20 BASE
21 COLUMN
22 TABLE STOP
23 RAM POSITIONING LEVER
24 QUILL LOCK HANDLE
25 DOVETAIL RAM
26 QUILL FEED HANDLE
27 BACKGEAR CONTROL
28 VARIABLE SPEED CONTROL
29 DRIVE MOTOR

Courtesy of Bridgeport Textron

End Mills



*Same as axial rake angle in plain and face mills. R.H. helix shown. This four-flute end mill is not center-cutting and cannot be used for plunge cuts

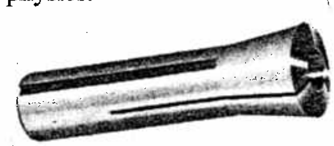
End mills are the primary cutting tool used on ram type machines. They can be had in different diameters, number of flutes, flat or balled ends, High Speed Steel or carbide. The examples are representative of the available styles.

The most common type is the two-flute end mill. It has cutting lips on one or two ends. The lips meet in the middle allowing the cutter to plunge into the work much like a drill does. These tools can be resharpened on the flutes and the ends, although this reduces their diameter by some.

The cutter RPM is determined by the end mill's diameter and by material type. Use the 100 Rule table. The data was compiled from end mill manufacturer information. Feed rate or chip load, is established by feed per tooth. Therefore you have to multiply the chip load or feed per tooth by the number of teeth on the cutter and the cutter RPM. See the table.

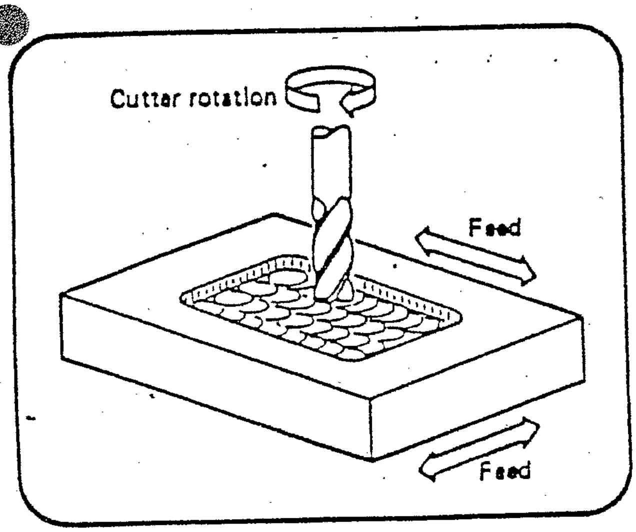
Depth of Cut for most ram type machines is about .050" deep. The reason being is that these machines don't have as much rigidity as vertical mills. Leave about .025" for a finish pass. Slow the feed rate by half.

End mills, in ram type machines, are held in collets having an R-8 taper. A drawbar pulls the collet tight into the spindle and solidly holds the cutter. Don't ever use an end mill in a drill chuck and mill with it. As the cutter cuts, the flute angle causes the cutting forces to resist in the opposite direction and pulls the drill chuck out of its adapter. It's called physics.

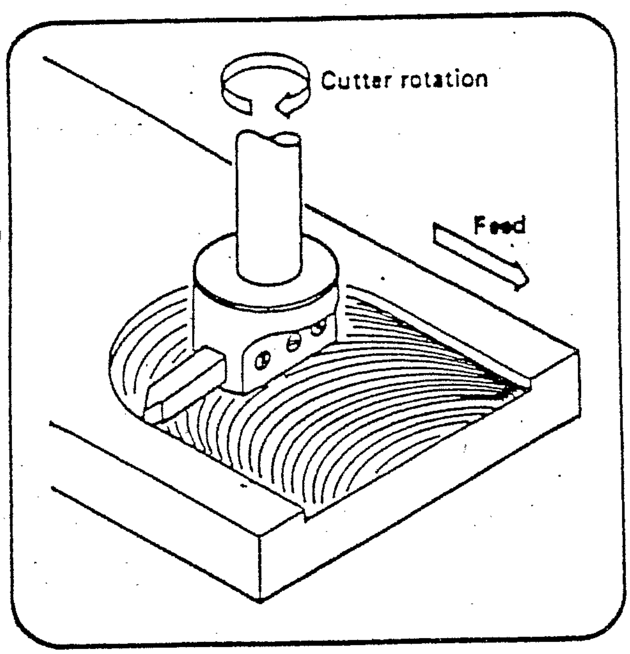


R-8 COLLET

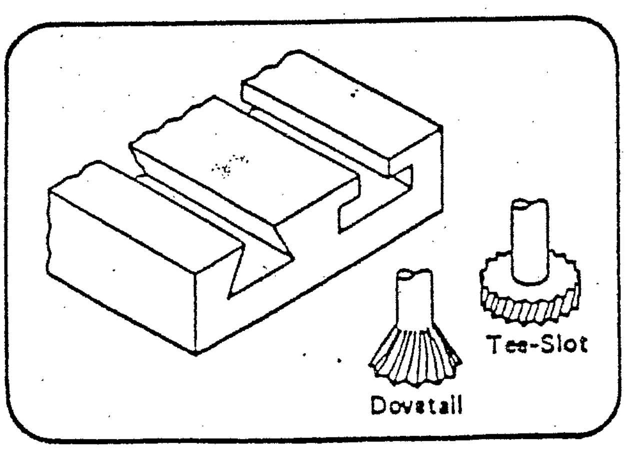
Arbor-mounted cutters: Vertical mill



Enclosed pockets are milled using an end mill.



The flycutter uses a single-point tool and is useful for surfacing large areas by taking light cuts, often at high speeds for good surface finishes.



Dovetail and T slots may be cut with the appropriately shaped cutter.

Mounting/dismounting Cutters

Cutters are mounted in the spindle and held by a collet. The collet has an R-8 taper that mates with the inside taper of the spindle. A drawbar pulls everything in tight.

Refer to the following illustration for the following information.

Mounting the Cutter

1. Find a collet with the correct diameter for the cutters shank diameter.
2. Slip the cutter into the collet.
3. Slide the collet into the spindle and that the key lines up with the collet keyway. It should slide in easily.
4. HAND tighten the drawbar.
5. Hold the brake against the pulley and tighten the drawbar with a wrench, generally a ¾" wrench.
6. Pull the wrench so the drawbar is tight, but don't over do it. Beginners usually over do.
7. Release the brake. Note. If your machine doesn't have a brake, put the spindle in its lowest RPM BEFORE you try to remove the tool. It helps to loosen or tighten a drawbar.

Removing the Cutter

1. Push the brake lever to prevent the spindle from turning. Using a wrench, loosen the drawbar ONE OR TWO TURNS ONLY. This is important so not to damage the drawbar end at the next step.
2. Using a hammer hit the top of the drawbar. This forces the assembly down and loosens the collet. Hold the cutter as you hit the drawbar. This prevents the cutter from falling damaging it.
3. Once the utter is removed, loosen the drawbar by hand to remove the collet.

RPM Selection

RPM's are selected by changing the V-belt from one pulley step to another. The pulley on the motor is large at the bottom and large at the top. Observe that the reverse is true on the spindle.

If the belt is on the top step, the RPM will be either 80 or 660 RPM. This is dependent if the spindle is in "backears" or direct drive. If the belt is in the bottom step, the RPM will either 325 or 2720 at the spindle.

Note that there is a "high" range (direct drive) and a low range ("back gears"). READ and FOLLOW the INSTRUCTIONS to change from direct to backgears and vice versa. This is IMPORTANT. If you don't, you'll grind the backgears when the mill is turned on. This is really a terrifying sound besides being an embarrassment to you!

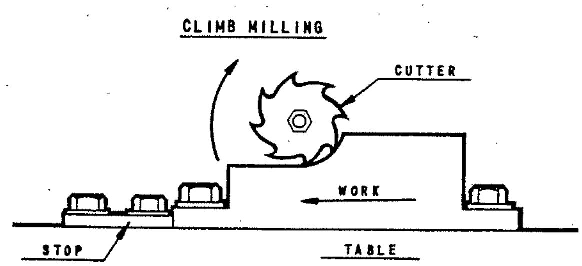
Variable Speed

Some machines have a variable speed drive. Turn on the motor, then turn the handle to the desired speed. DO NOT ever turn the speed dial lever when the motor is not turning. You'll bind up the belt and damage the drive. The drive can be put into low gear or back drive as the V-belt drive.

Cutter Direction

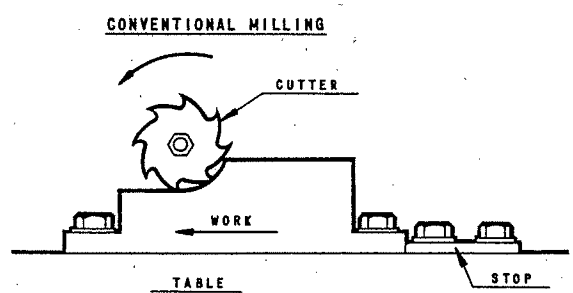
Milling cutters travel and rotate about its centerline. If the center of a cutter were traveling along the work centerline, one half of the cutter would be climb milling while the other half would be conventional milling. This is important because of the cutting forces involved and how they affect machine accuracy.

Climb Milling



When the direction of feed and the direction of cutter rotation are both in the same line of travel, the resulting forces are traveling in the same direction. In climb milling, the cutter starts with a thick cut and ends with a thin edge. The resultant force tends to pull the work ahead as it cuts. The leadscrew is pushing the table forward. When the cutter starts to cut the chip, the cutter grabs the work and pulls the table against the front side of the leadscrew. This results with damage to the nut and increasing the amount of backlash. Climb milling produces a good finish because the cutter starts with a big "bite". DO NOT climb mill on manual machines unless they have anti-backlash devices so as not to damage the leadscrew and nut.

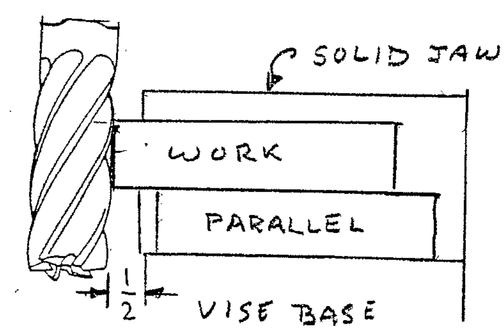
Conventional Milling



In conventional milling, the cutter rotation and direction of feed are opposed in the line of travel. In conventional milling, the cutter starts with a thin cut and ends with a thick end. This action makes the cutter rub against the surface before cutting and dulls the cutting edge and leaves a poor finish. Because the cutting forces are always opposed, the leadscrew is always against the nut and prevents damage.

Be aware of cutter direction when you machine the hammers' head angle. On manual machines, always conventional mill. If you need to return to your start point, go slow. When you cut, the cutter actually bends away from the work because of cutting forces. On your return cut, the cutter is stiffer because it isn't cutting as much material, climb cuts and leaves a better finish. In CNC practice, roughing cuts are done conventionally and finish cuts are climb milled.

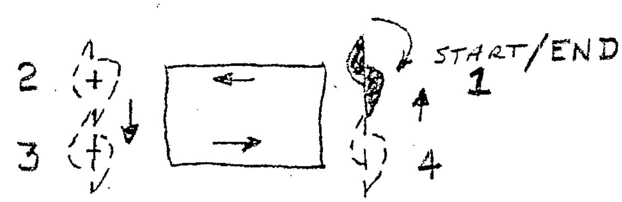
Milling the Hammer Head



You've cut the stock and now have to mill the end to make it smooth and square. Put the work in a vise and on parallels. Extend the part, on the left side of the vise, so it is beyond the end about ½" or so. The cutter will start away from you and end closest to you. Turn the machine on and bring the cutter so it touches the end. Next, move the cutter to start position. Move the table to the left about .015" to .020". SLOWLY feed the cutter to the front or end position. This is conventional milling. Slowly feed it back to the start point. Remember, it's climb milling on its way back. Repeat again if you need to. When done, flip the part around and repeat the process until the part in machined to length.

Face Milling

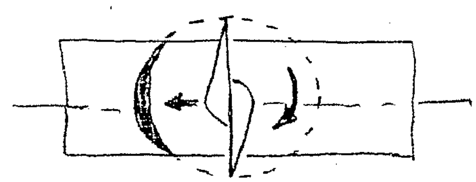
Once the part is to length, you can mill the angle head. Using a protractor, set the work in the vise and tighten the vise to hold the work to the 30° angle. As you mill the part, remember to ALWAYS conventional mill. Note the start point. You are going to mill a "box" pattern. In this operation, only the corners of the cutter will be cutting. This is called face milling.



Bring the cutter over the high point of the work. Bring the cutter so it touches the top of the piece. Now, move off the piece. Raise the table for a DOC about .030". SLOWLY feed the cutter to position #2. Move the table and bring the cutter to position #3. Slowly feed to position #4 and finally back to the #1 position. Raise the table for another cut and repeat the steps until you've reached the correct depth. As you cut, notice the cutter is always conventional milling. This is proper procedure on conventional machines.

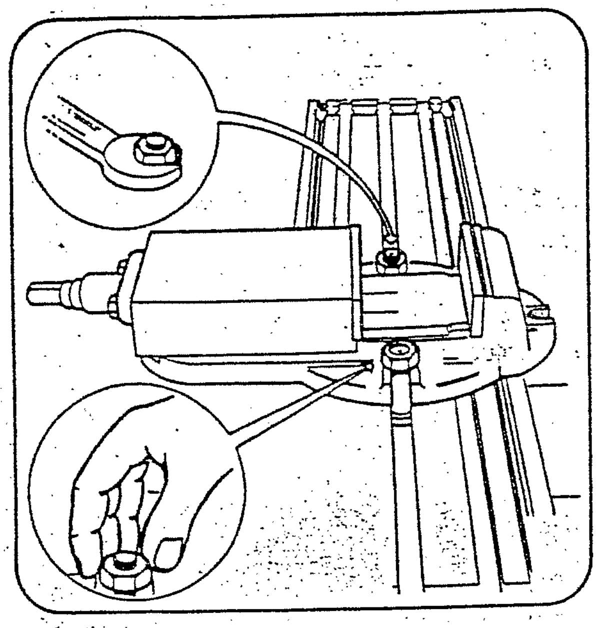
Neutral Milling

If the cutter is wider in diameter than the Width of the work, aligning the cutters' centerline to the works centerline causes the cutter to "neutral" mill. This means that half of the cutter is climb milling while the other half is conventional milling. The two forces cancel each other out. With this practice, the cutter can be fed in either direction. It is also the most efficient means of milling.



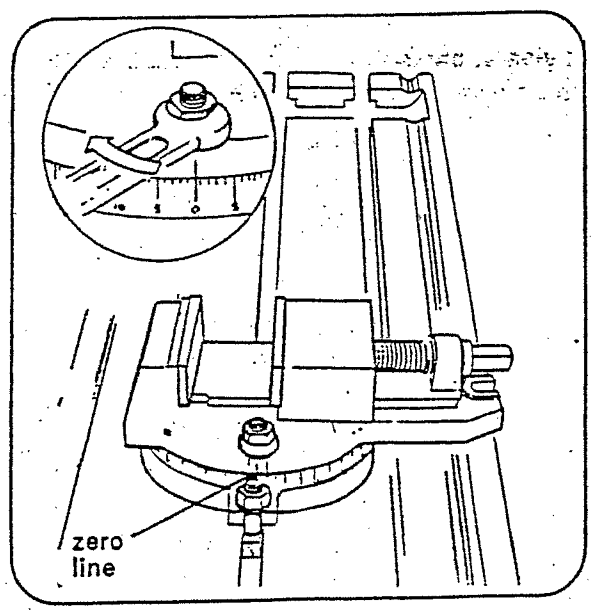
Work Holding – Vises

The most common work holding device used in the milling machine is the vise. These vises are pretty precise and need to be treated with care. By the way, they're also pretty heavy. There are two types, a plain vise and the swivel vise.



The Plain Vise

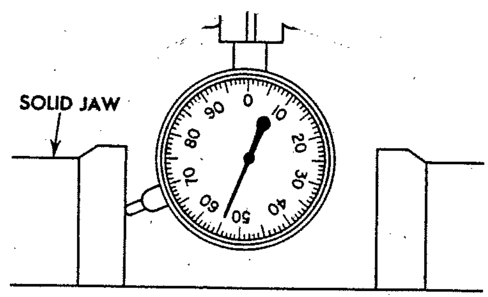
Mounted directly on the table, it is a commonly used. Before mounting, clean the surfaces and table. Place a t-nut, stud, washers and a nut in each slot of the vise. Aligning requires a little knowledge. First, alignment is done with a dial indicator. The trick is to snug one of the studs and use it as a pivot point. Do that. That is point #1. Mount a dial indicator in the spindle and secure the spindle so it doesn't turn or move. Bring the indicator so it makes contact with the solid jaw at point 1. Set it to zero. Move the table so the indicator travels along the solid jaw to the other end. Note which way the dial moves. At the end of travel, stop. Tap the vise so it moves until the dial reads zero. Snug the second stud. Go back to position 1, reset zero. Travel to position 2 and check the reading. It should read zero. Tighten everything down and recheck. Remember, the key is using one stud as a pivot point. The jaw should indicate zero along its entire length.



The Swivel vise

Basically a plain vise is mounted onto a base plate. The base allows you to turn the vise to a specific angle and mill to. In setting up the vise, align the base zero to the vise zero and tighten these bolts. Now, set up and align this vise exactly like you did for the plain vise. This is important as it allows you to swivel accurately to the angles required.

You've now swiveled the vise and need to re-align it. Set up a dial indicator and have it touch one end of the solid vise jaw and set zero. Move the indicator to the other end. Note the indicator reading. Loosen both nuts. Swivel the vise so the reading returns ½ the reading back towards the zero. Tighten the nuts. Recheck and readjust if necessary. The jaw should indicate zero along its entire length.



Aligning the vise with a dial indicator

Edge Finding

In milling, an important factor is to locate a feature an exact distance from an edge. There are several means of locating the cutter to the edge. In essence, we are locating the spindle's centerline to the edge or both edges so we can position it precisely to the location required.

Paper/cutter Method

Using this method, a strip of paper is held between the thumb and first finger. The paper is places between a rotating cutter and edge. The work is brought towards the cutter. As the work comes close, the cutter starts to cut he paper. You can feel the cutter cut the paper. When this happens, STOP. The cutter's edge is .about .002" away from the edge. Set the dial to zero. Move the cutter one-half its diameter plus the .002". This should put the spindles' centerline exactly on the works' edge. Set the dial to zero. Remember which way your backlash is going. This method works well for both horizontal and vertical machines.

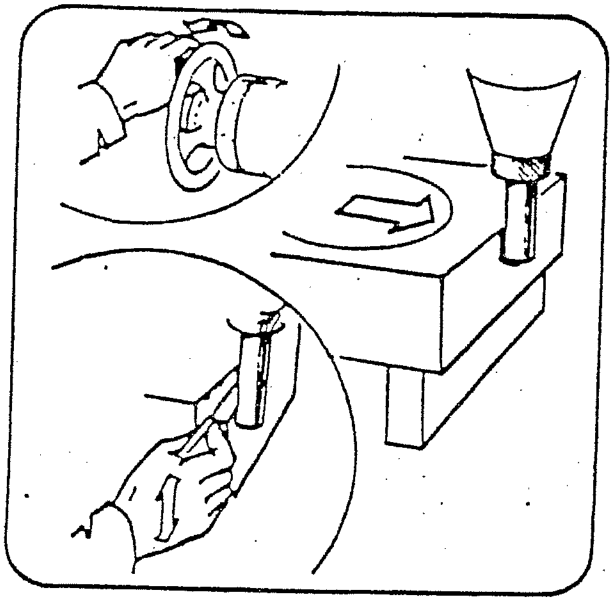
A couple of hints. Run the cutter, if practical, around 1,000 RPM. Make sure the cutter "pulls" the paper through it. You'll be able to feel the cutter cut the paper. There will also be paper dust that you'll be able to see. Really.

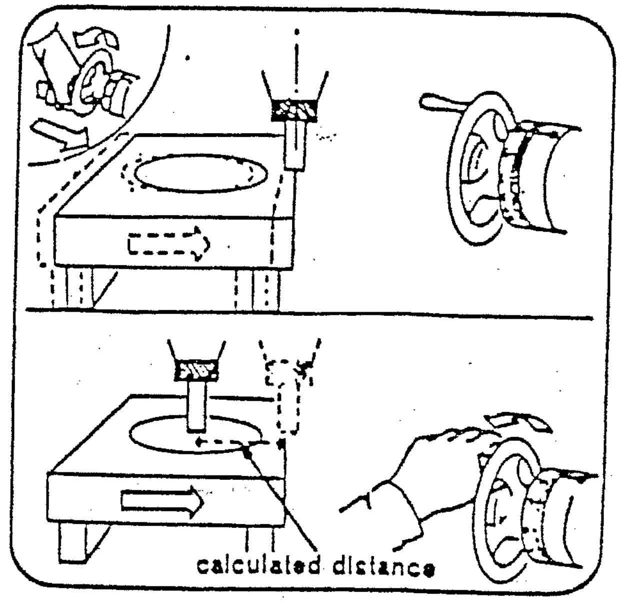
Edge Finder Method

A spring loaded edge finder is composed of a solid edge locator, a spring and a tube. As the edge finder turns, the solid locator wobbles out of center. As the work is brought to touch the edge finder, the locator straightens out. When the locator just barely goes past the spindle centerline, the locator "kicks out" to one side. This shows that the spindle centerline is one-half the locator's diameter away from the edge. Set the dial to zero. Move the table one-half the edge finder diameter. Reset the zero. The edge should be perfectly line to the edge. Remember your backlash direction.

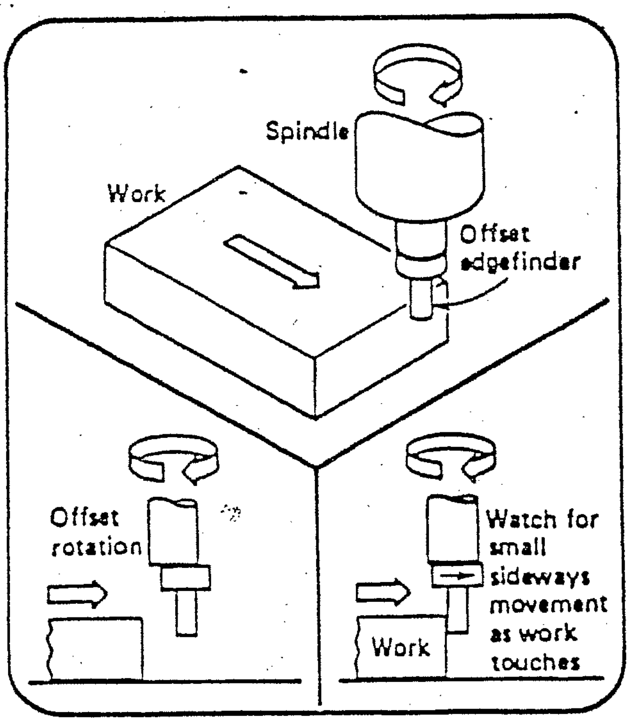
These two methods are the more common of many methods used. For learning practice and most machining applications, they are adequate. See the next section on using the edge finder to locate the tapped hole on your hammer head.

Edgefinding

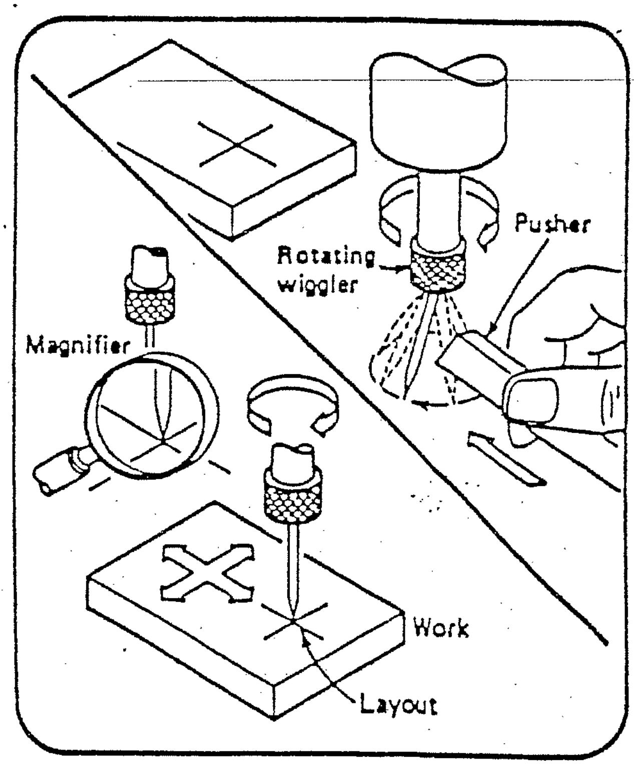




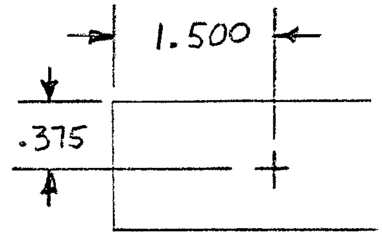
Using the offset edgefinder



Using a wiggler center finder



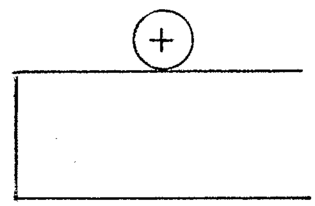
LOCATING THE 3/8" HOLE



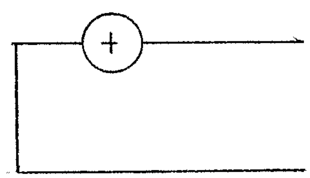
The drawing shows that the 3/8" tapped hole is located as shown. We will use the edge finder, E/F, to precisely locate the spindle to the dimensions.

Procedure

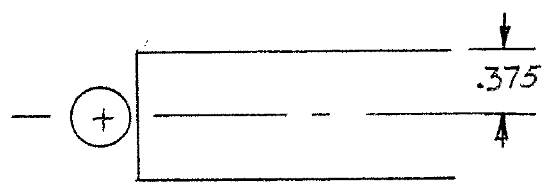
- Set the part on 1/8" thick parallels, and tighten the part in the vise.
- Mount an E/F in the drill chuck. We're using a .500" diameter E/F. Use around 600 RPM.



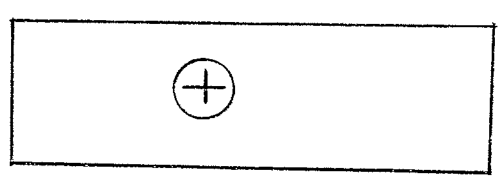
- Working from the solid jaw, bring the E/F, So it just touches the edge. Bring it in slowly until the E/F "kicks out".
- Set the cross-slide dial to "0".
- Recheck the procedure to verify that the E/F kicks out at the "0" setting. Reset the dial and recheck if necessary.
- Move the table half the E/F diameter, 0.250" in this case. Remember to lift the E/F above to work surface so you won't crash it into the work. The spindle centerline should be exactly aligned With the work's edge.



- Reset the dial to "0".
- Move the table in .375", so the spindle is now located on the work's centerline, .375" from the edge. Lock the table.
- Bring the E/F to the front of the work. Repeat the above procedure to locate The 1.500" dimension.



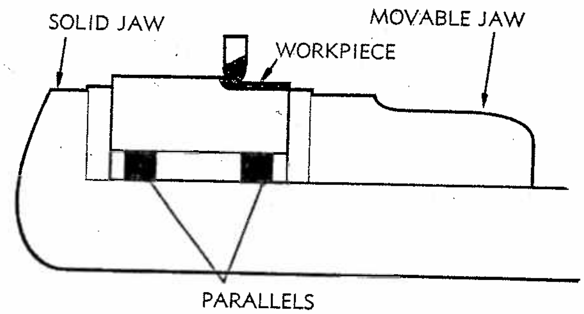
- Lock the table slides. Center drill, 3/8" drill by ¼" deep. Drill 5/16" diameter through and 3/8"-16 tap.



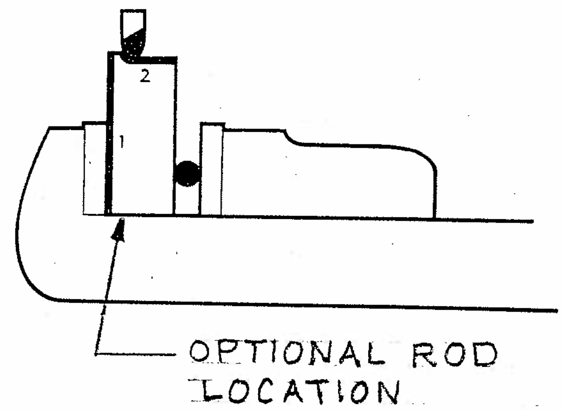
SQUARING A BLOCK

In order to achieve exactness when milling, the most important requirement is start with a straight edge or a square block. Milling surfaces square to each other requires attention to procedure. The given method can produce surfaces square to each other within, realistically, .004". Anything more precise than this usually requires a surface grinder and angle plate.

1. Set side 1 up as shown. Use a 1/8" thick parallel or round steel rod as a parallel. Always work against the solid or fixed jaw. Using as big a diameter end mill as practical or a fly cutter, mills side 1 to clean.

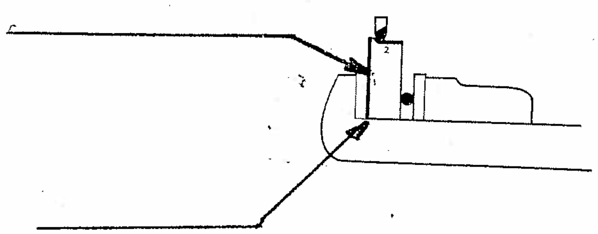


2. With side 1 against the solid jaw, set up the block as shown using two rods as parallels. The rods provide only one line of contact and minimize the effect of an uneven surface. Seat the part with a light hammer blow at the back edge.
   Mill side 2 to clean.



Remove from the vise. File the burrs and check for square using a solid square.

If the part is out of square as in figure A, place paper shims at the top of the jaw. Remill side 2. Recheck.

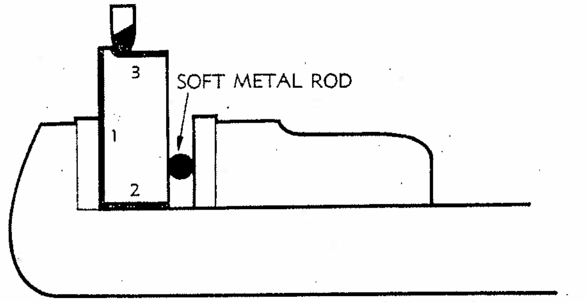


If the part is out of square as in figure B, shim the bottom. Remill side 2. Recheck.

NOTE: Side 1 is ALWAYS against the solid jaw. Use a precision square to check squareness.

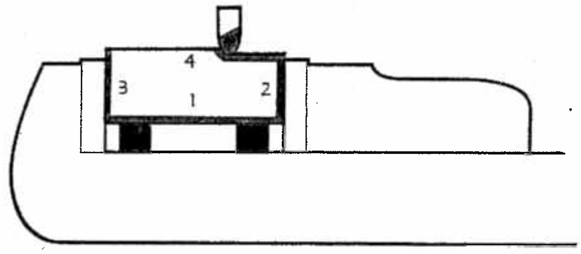
3. With side 1 against the solid jaw and side 2 on the rod or parallel, the piece shimmed accordingly, mill side 3 square to side 1 and to dimension.

   Recheck squareness when done.



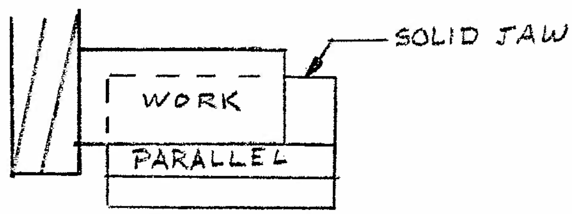
4. Place side 2 against the solid jaw. Use thin parallels or rods to seat the work. Tap with a hammer. Do not put more pressure on the vise handle to re-tighten some because you'll lift the piece up. Physics. Re-seat the work with a hammer. Mill side 4 square to side 1 and to dimension.

   Recheck squareness when done.

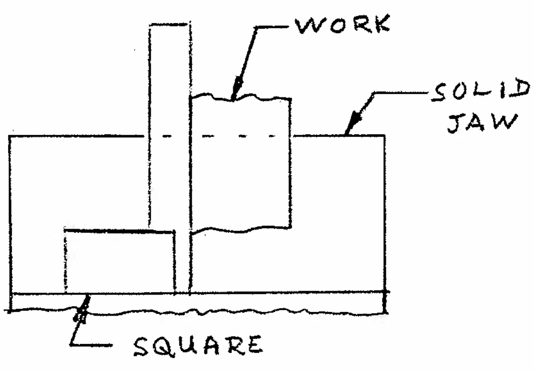


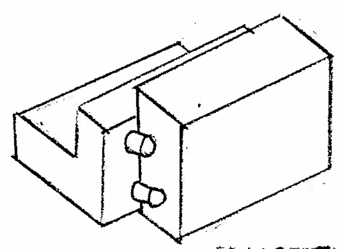
This just leaves the ends to mill square. Here are some options. Use shims if necessary.

Method A: The work is held horizontally and the end extends past the vise jaw.
An end mill machines the end smooth, square.
Reverse ends and mill to length.
This method was explained in machining the hammer head.

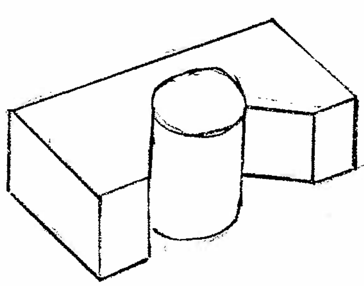


Method B: This method sets the part vertically against the solid jaw.
An end mill face mills the top surface smooth, clean and square. This method squares the surface to both surfaces.
Figures 1, 2 and 3 show the process.

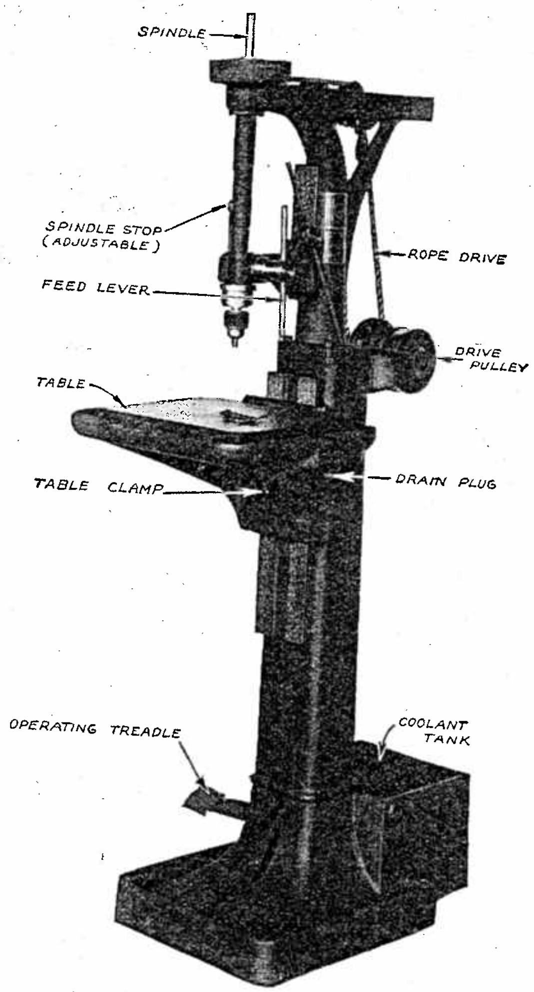




ANGLE PLATE &
WORK AGAINST SOLID JAW



WORK HELD IN A V-BLOCK
AND SOLID JAW



1-12″ Single Spindle, Belted Spindle Floor Type Ball Bearing Drilling Machine

Courtesy of Leland-Gifford Company, Worcester, Mass.

DRILLING AND DRILLING OPERATIONS

Almost every part manufactured has a hole of sorts machined into it. The traditional twist drill is used most often, but new styles of carbide inserts are used in modern production manufacture. As simple as drilling is, it can be a technical nightmare especially in hard or exotic materials such as titanium.

Drill Press

The drill press is the common machine used to drill holes, though drilling is also done on lathes, milling machines and other special type drilling machines. Hole producing is the same no matter what machine type is used.

Drill presses are generality of the "sensitive" type. This means that the operator feels the drill cutting because the tool is hand fed into the work. Drill presses can be bench mounted or floor styles. Production machines have power feeds, are heavier in design and weight and can have reverse for power tapping threads.

Major Parts

Base: Cast iron base or bottom that holds the column and is either bench or floor mounted.

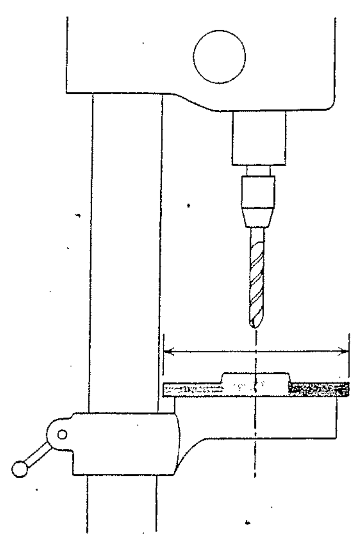
Column: A vertical steel tube mounted from the base. Attached to the column is the table and toolhead.

Table: Holds the work, vise or drill jig. It moves up/down on the column and can be locked in any position. Some tables tilt to the left and right. Please, DON'T EVER DRILL into the table.

Toolhead: Consists of the motor, drive pulleys, quill, spindle, drill chuck and feed lever. A drill chuck is mounted in the spindle. Some chucks have a Morse taper adapter that mate with the spindle taper. Some have a "screw on" chuck. Look on the chuck or on the spindle to find out.

Size

A drill press is sized by "the largest diameter circle it can drill to the center". This is a little old time marketing strategy. A 15" drill press drills to the center of a 15" diameter. The true size means that the distance from the drill center line to the column edge is 7 ½".
When you buy a drill press, keep this in mind.



Drill presses are measured by the largest diameter of a circular piece that can be drilled in the center.

DRILLING OPERATIONS

There are many things that can be done on a drill press or similar machine. There are many requirements that can be done to a hole. This is an explanation of some of them. See the accompanying illustrations.

Drilling: The process of producing a hole in a work piece using a tool, generally a twist drill.

Reaming: The process of remachining a hole to make the hole smooth and to accurate size with a tool called a reamer. Reaming does not make a hole straight.

Tapping: The procedure of making an internal thread in a hole using a tap.

Counter-Boring: The process of enlarging a previously drilled hole to make the top a larger diameter in order to let a bolt head sit below the top surface.

Spot-Facing: Machining the top surface of a casting to produce a smooth and flat surface to allow a washer and nut to sit evenly on the casting. This allows holding forces to be equally distributed on the surface. Use a counterbore tool to accomplish this feature.

Counter-Sinking: Enlarging the top of a hole to a conical shape that allows a flathead fastener to sit flush with the top surface. Generally 82° but can also be 60°, 90° or 110°.

Boring: The process of enlarging a previously drilled hole with a single point tool. This makes the hole round, straight and to size. Boring is also done on lathes and any other machine capable of drilling.

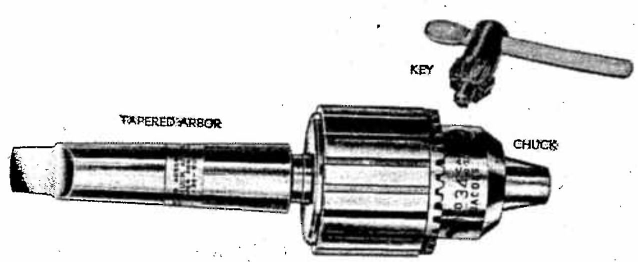
Tool Holding

Generally, drilling tools have either a straight shank or Morse taper shank. Straight shank tools are held in a drill chuck while tapered shank tools are held in a tapered holder or directly into the spindle. Whenever you use a drill chuck,

A. Push the tools' shank into the chuck as far as possible,
B. Tighten the drill with the correct chuck key in ALL THREE holes.

Both of these details will prevent damage to the chuck and produce better results.

Spindles may also have an internal taper, generally, a Morse taper. The taper will directly accept drills, reamers and drill chucks with a taper shank into the spindle. This provides better control of a tools' runout (wobble). When installing a taper into a spindle, make sure that both tapers are clean. This prevents runout and makes the tapers hold onto each other more effective.



Courtesy Jacobs Manufacturing Co.
Key-type drill chuck

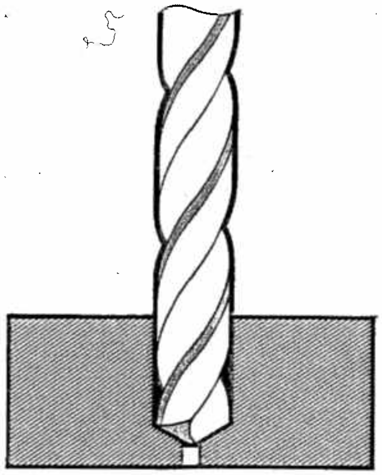


Fig. 8-6A Drilling

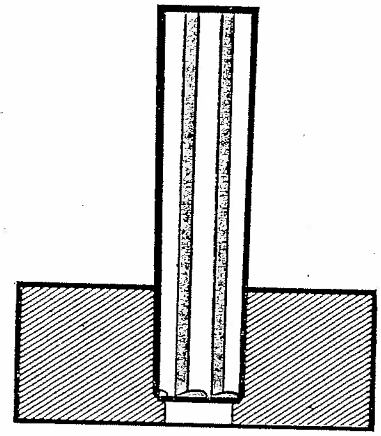


Fig. 8-6E Reaming

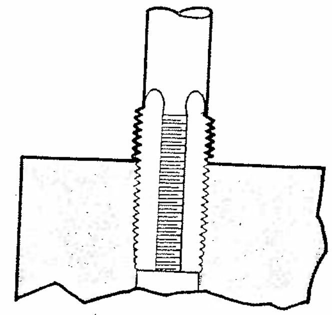


Fig. 8-6D Tapping

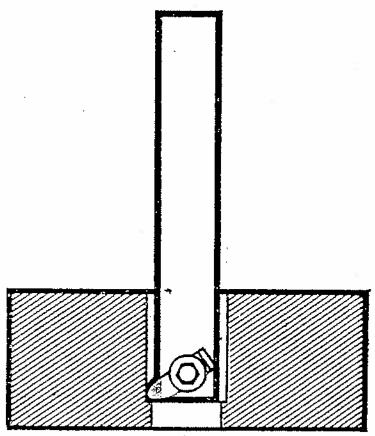


Fig. 8-6F Boring

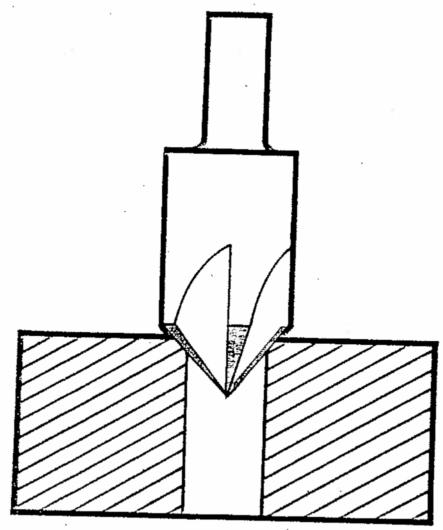


Fig. 8-6C Countersinking

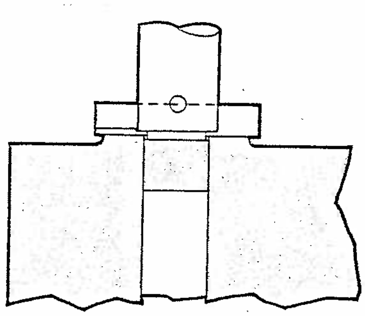


Fig. 8-6B Spot Facing

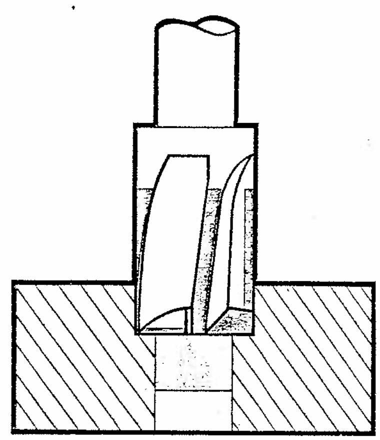


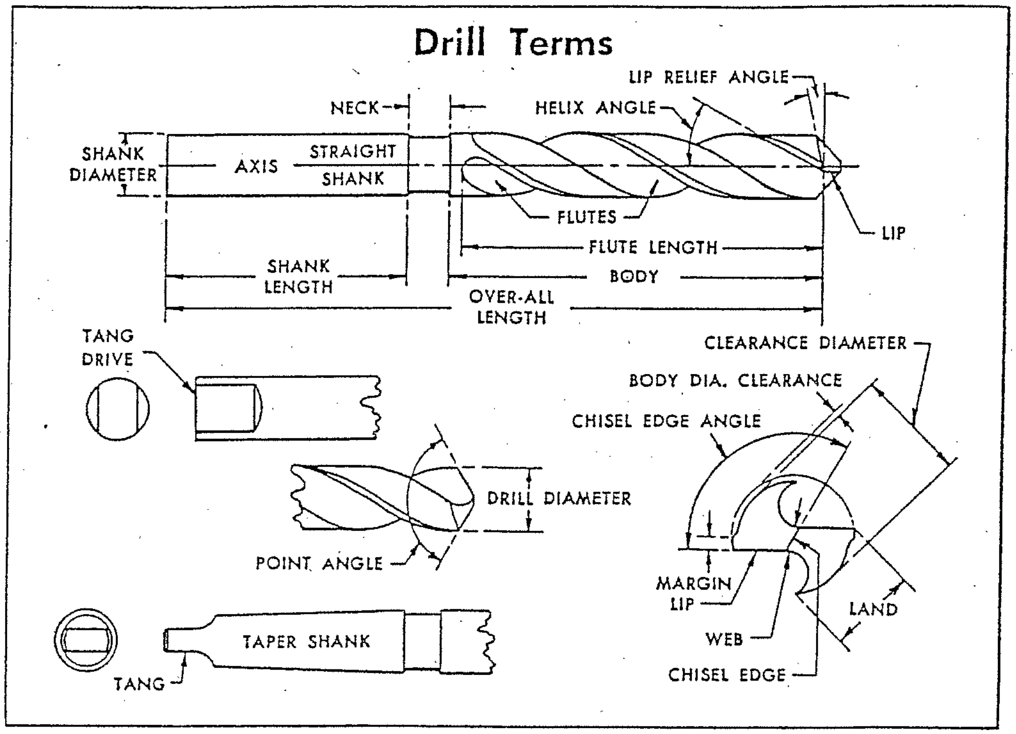
Fig. 8-6G Counterboring

When removing tapered tools from the socket, you need a tool called a "drift". It is a flat piece of metal with a flat bottom and a rounded edge on the taper. Insert the taper into the spindle, remembering the "flat to flat, round to round" rule. Use a hammer to tap the end. The drift will push the tapers art. REMEMBER to hold the tool because as soon as the holding grip is released, the tool will fall out.

When mating tapers and the taper is too small to fit in the socket, build it up with a taper to taper sleeve.

Drill Terms

The following illustration shows the major parts of a twist drill.



The shank is what holds the drill into the machine. As stated, shanks are either straight or tapered. The tang of a tapered shank is what does the driving while the taper only holds the tool securely.

Flutes are the grooves that allow the chip to flow out of the drill. The flutes also form the cutting lip, which does the actual cutting.

Drill diameter is the size of the drill. It is found stamped on the shank. IMPORTANT. When measuring the diameter of a drill, you must measure over the margins.

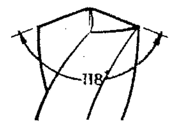
The web is the spine of the drill. It is tapered from the lip to the shank and the diameter gets progressively larger. That is why sometimes the lips are thinned when the drill is sharpened too many times.

Margins or land are the tiny area that is the full size of the drill. Margins are machined into the flute to prevent the sides of the flute from rubbing and to allow coolant to reach the drill.

The chisel or dead center is where the two lips meet. The dead center does not cutting and requires a great amount of force to feed the drill through the work.

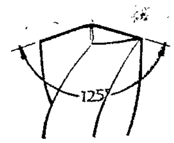
The lips are where the cutting actually is done. They are formed by the intersection of the flutes and drill point angle.

The drill point is the angle the lips make with each other. It is generally 118° but it can be changed for other materials.



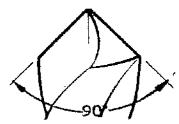
(a)

The 125° or 135° drill point angle is used on stainless steel and other hard to drill materials. The larger angle provides more metal behind the lips thus giving the drill more strength and durability.



(b)

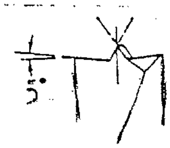
The 90° angle is generally used on wood, but the 118° point works well.



(c)

The spur point is used on sheet metal.

The tips of the "wings" act like a can opener and make the hole round. If you use a 118° point on thin metal, the lips break through the metal before the drill diameter and actually leave a triangular shape hole in the metal.



Drill Sizes

There are four systems used to define a drill's diameter. Two of these, letters and number were developed to meet tap drill and wire size requirements because fractional size drills were too small or too large to meet pitch diameter calculations. Also, today's drills can be coated with Titanium Oxide (TiN), a "gold" coating that helps preserve the durability of the drill in the cut.

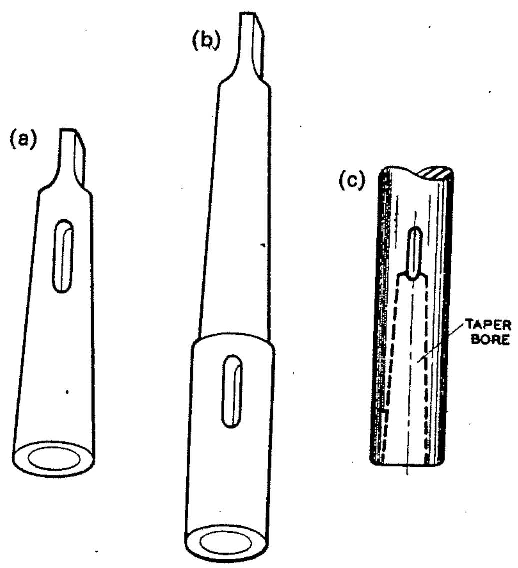
Fractional: From 1/64" (.015625') diameter to 3 inches in diameter by 1/64" increments.
Letters: Letter sizes from A (.2340" diameter) to Z (.4130" diameter).
Numbers: From the #1 (.228" diameter) to #80 (.0135" diameter)
Metric: From .5mm to 75mm in .05mm increments.

Fractional (1/64" to ½"), letter and number drills can be bought as having all the drills in the set.

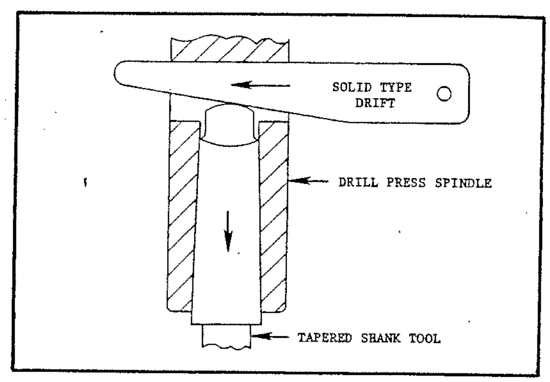
Morse Tapers

The Morse taper is the most common style taper used to hold twist drills in the drill press socket. (There are other tapers, such as the Jarno, Brown & Sharpe that used for different applications.)

The tang is what drives the drill and the taper holds and aligns the drill into the socket. That's why you must clean both the taper and socket before mating the two. To remove the drill from the socket, you need a "drift". It is a flat tapered piece of metal. To use, put the flat surface of the drift so it is against the flat of the drill and the round of the drift of the drift against the round of the socket. Tap with a hammer to remove. Make sure to catch the drill. The rule: "flat to flat, round to round".

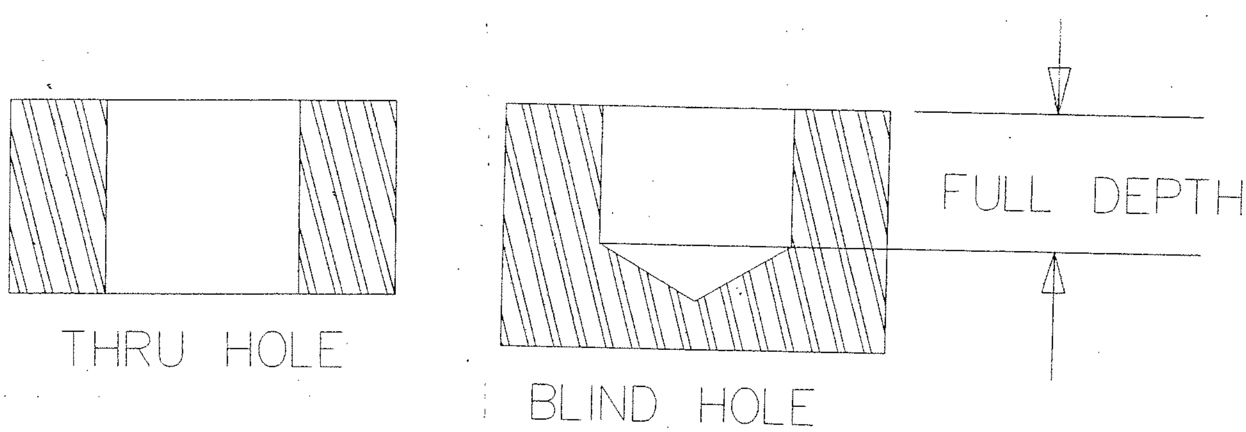


(a) Drill sleeve
(b) Drill socket
(c) Drill press spindle

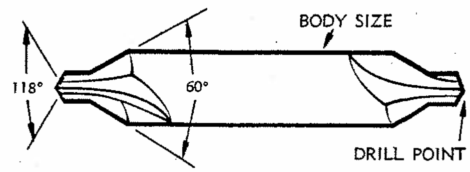


Hole Types

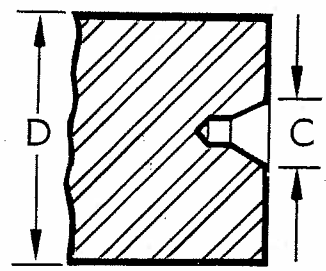
Basically, there are only two types of holes – those that go through the piece and those that don't, called a blind hole. The depth of a blind hole, unless noted, is always from the top to the full diameter at the bottom. See illustrations.



Center Drilling



Combination drill and countersink



Courtesy Cleveland Twist Drill (Canada) Ltd.
Cross section of a centre hole

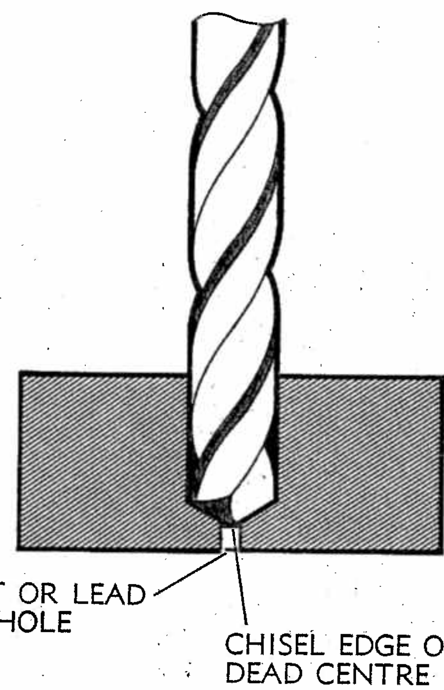
A center drill has a stub drill with a 60° cone at each end. The stub drill point prevents the drill from walking as you start the drill. The 60° cone serves two functions. The first is to create a cone for the drill to locate itself in so the hole is located in the right location. The second purpose is to serve as a means of work holding when turning with centers on a lathe. When center drilling, drill from 2/3 to ¾ the cone's length for depth.

A spotting drill has a short length and has a 118° drill point angle. Its purpose is to locate the drill location. Its drill point matches a regular drill point so, again, the drill doesn't walk.

Important. Before drilling a hole, the proper procedure is to center drill the hole location first, followed by the drill size.

Pilot Drilling

Pilot drilling is used to drill a small hole before drilling with the correct size drill. The pilot drill size is about 1/32" larger than the dead center. The reason to pilot drill is because the dead center does no cutting and the pilot drill helps' the larger drill through without the added cutting pressure. The machine horsepower helps determine whether you should pilot drill or not. A ½ horsepower machine might need a pilot while a machine with 5 HP can feed a 1-½ diameter drill with no problem.



Drilling a pilot hole first helps the larger drill to cut more easily

REAMING

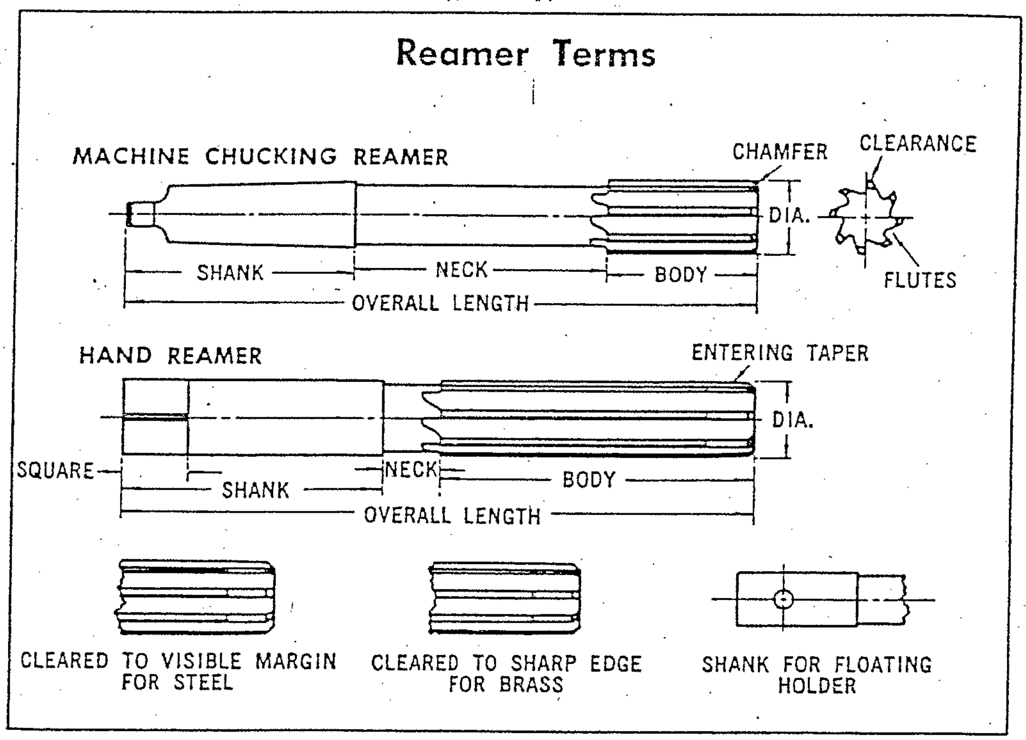
Reaming is the process of making a hole smooth, and to size. Reaming doesn't make a hole round or straight.

Machine Reamers

Machine reamers have a tapered or straight shank. All reamers have multiple cutting flutes. Machine reamers cut at their corners as they are fed through the work.

Hand reamers have a straight shank with a square tang on the end. The beginning of the reamer is tapered to about a third of its length at the beginning. A hand reamer is to scrape a hole because the hole is a little too small. Hand reamers remove about .002". They are generally used in assembly areas.

Shown below are reamer types and terms.



A reamer is used to finish a previously drilled hole. There are a few rules to remember.

For holes ½" diameter and smaller, drill the hole 1/64" smaller.
For holes ½" to 1" in diameter, drill the hole 1/32" smaller.
For holes 1" diameter and above, drill 3/32" smaller.

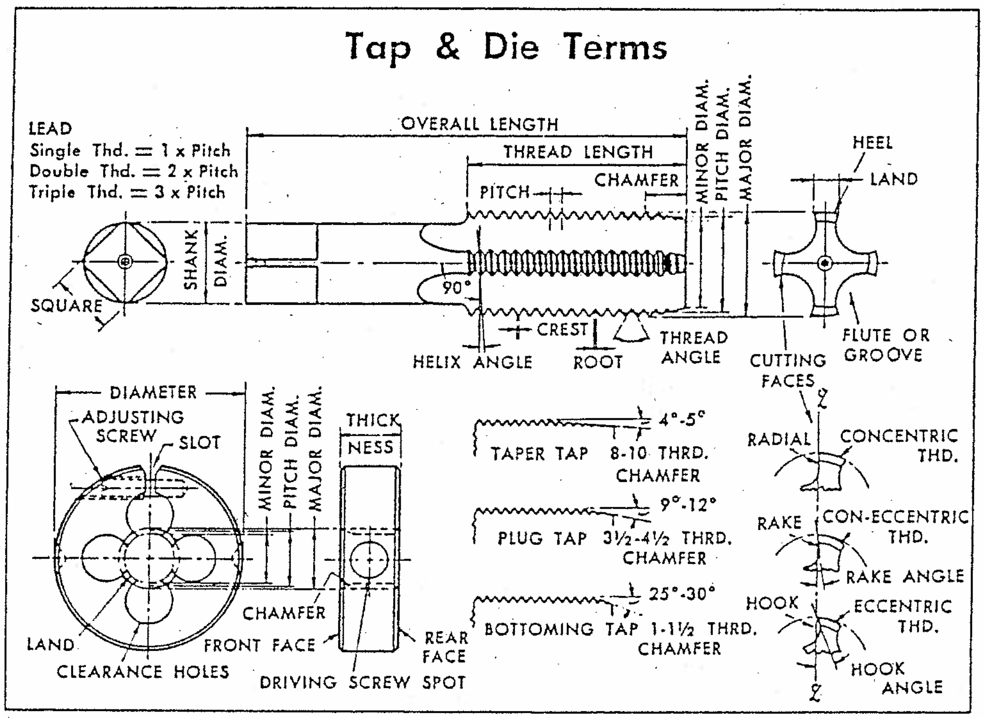
The reamer RPM is HALF the drill RPM and the feed is TWICE the drill feed. That's because you have more than two flutes doing the work.

Remember: NEVER run a reamer backwards, and finally, always USE cutting oil.

TAPPING

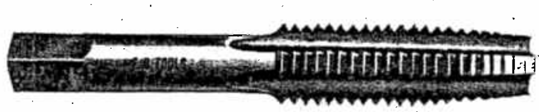
Tapping is machining internal threads into the work with a tool called a tap. Internal threads are found on nuts, hammer heads, engine parts and the like.

There are many styles of taps for many different purposes. Basically they all have the same geometry because cutting action is cutting action no matter the size. There are three shapes a tap can be depending upon the power used to feed the tap. The chart shows the common tap terminology.

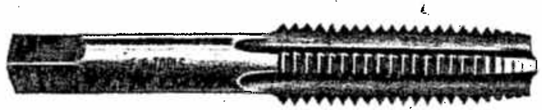


Tap Types

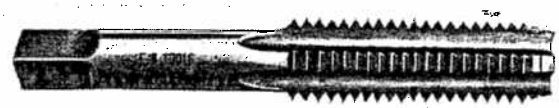
Taper: The first 8 to 10 threads are tapered. This taper type is used to start the tap square with the hole when hand tapping. Use a tap guide when hand tapping.



Plug: The first 3 to 5 threads are tapered. This style is used with power tapping. Power tapping holds the tap square to the work so there is no need for the long starting taper.



Bottom: The first 1 to 1-1/2 threads are tapered. This tap is used after a taper or plug tap. It is used to finish cutting to the bottom of a blind hole.



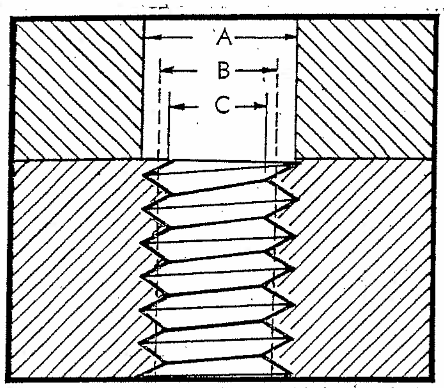
Gun Tap: A style of tap that pushes the chip ahead of the generated cut. It prevents the chip from clogging the tap flute and break the tap. It is used on through holes.

Spiral Tap: A style of tap has pulls the up and out of the hole. It is used on blind holes and helps to prevent the tap from breaking.

As a note, all taps have a square tang at the end of a straight shank. This allows either hand or power tapping.

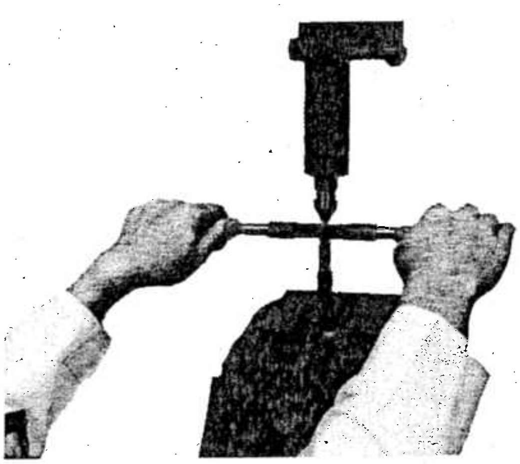
TAPPING A HOLE

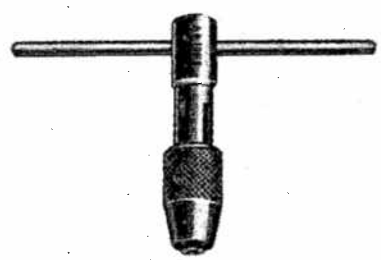
When tapping a hole, the hole must be smaller than the tap diameter. To determine tap drill size, look at a drill chart. When tapping the 3/8"-16 thread, find 3/8"-16. Look to the left and you'll find the size listed as 5/16" diameter. This drill will provide enough material for an adequate thread form to hold the fastener. Use the tap drill chart to find the right size for the tap you need. The illustration shows the difference between a clearance hole, A, for the bolt to fit through, a tap drill size hole, C, which leaves enough material to cut the thread form and B, which is the pitch diameter.



Cross section of a tapped hole: A = body size, B = tap drill size, C = minor diameter

After the hole is drilled, select either a taper or plug tap. If the work is still in the machine, put the tap in the chuck and turn by hand. Another choice is to use a spring loaded tap guide to keep the tap square to the work as the tap is fed in. To drive the tap, you can choose the adjustable tap wrench or the T-handle tap wrench.


Guiding a tap with a centre


Courtesy Firth-Brown Tools (Canada) Ltd.
T-handle tap wrench


Courtesy Firth-Brown Tools (Canada) Ltd
Double-end adjustable wrench

A tap guide is a must. If the tap is not held square to the work, it will bind and break. Not a good situation. When hand tapping,

1. Locate the position
2. Center drill and drill.
3. Using a tap guide,
   A. Put cutting oil on the flutes. This is a must.
   B. Turn and feed the tap two to three turns.
   C. Turn the tap backwards half a turn every two to three turns. This breaks the chip and helps to prevent the tap from breaking.
   D. Continue with this method until the hole is tapped through.

A Final CAUTIONARY note:
If the tap feels hard to turn or not going in straight, STOP. If you don't, you WILL BREAK the tap. You don't want to do that. Develop a feel as to how the tap is cutting. If it feels hard, stop. Don't try to be Superman and use lots of force. It will break.

Broken Taps

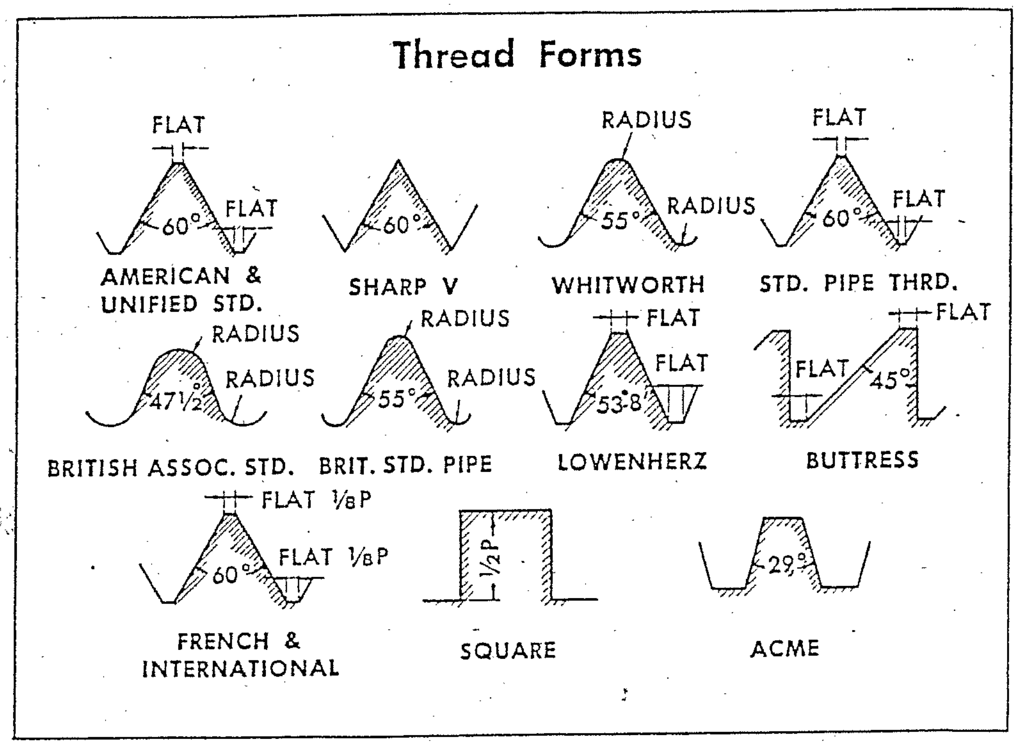
A broken tap is a bane to machinists, especially if it is in a blind hole. To remove it, you'll need small picks, a small hammer, pliers, lots of PATIENCE and lots of luck. Try to tap the broken tap forward and backwards to try and loosen it. Try to remove any broken pieces. Keep trying until you can back the tap out. The easiest way to remove a broken tap is to use a machine called a tap burner or an Electrical Dischage Machine (EDM).

Hand Threading Dies

Hand threading dies are used to cut threads on a rod. They generally work best to recut a thread that has been damaged. The first thing one needs to do is to grind a chamfer on the end of the rod. Using the proper size die and having mounted in the die stock, the die can be turned onto the rod. Note that the die has a long starting chamfer on one end. This end cuts first. Mount the rod securely in a vise. Use plenty of cutting oil. This process does not produce good external threads, especially if a good thread form is desired.

THREADS

As the illustration shows, there are many type of thread forms available to the designer. Modern manufacturing uses the ISO standard for metric threads. In the USA, there are primarily two standards found on threads. The first is the Unified National System and the Acme system.



The chart illustrating tap markings show other types used in varied assembly practices, liquid, gas fittings, microscope and the like.

Since 1947, all American threads came under the Unified System (UN). The Unified National System uses the coarse series, UNC (formerly the USS series). The UNF (formerly the SAE series) is used for fine threads and the extra fine, UNEF series. There are both a coarse and a fine thread series for each specified diameter. Use Machinery's Handbook to find all thread data for these and other thread forms including metric or ISO threads.

Metric threads are used worldwide, except in the USA. They have the same 60° angle, form and shape as a Unified Thread.

Threads are used for three reasons:

1. Fasten parts together, like a nut and a bolt.
2. Provide motion, like a house jack.
3. Provide a means of fine adjustment such as a micrometer or microscope adjustment.

THREAD TERMS

All thread terms have common terminology. The illustration shows some of them. We will define only those related to the hammer project. If you need to obtain more information, again, the first place to look is in your Machinery's Handbook.

Major Diameter: The largest diameter of the thread, the outside diameter.

Minor Diameter: The smallest diameter of the thread.

Pitch Diameter: The imaginary diameter of the thread usually found near the halfway point of the tread depth. It is the most important element of the thread because all calculations on strength, size, fit and inspection come from this imaginary diameter. The diameter lies at the point where the width of the groove equals where the thickness of the thread material lies.

Thread angle: This is the angle formed by the two sides of the thread form. Unified and metric threads have a 60° included angle with a flat crest (top, or major diameter), and/or a rounded root (bottom, or minor diameter).

Pitch: Defined as the distance from the point of one thread to the point of the next thread as measured axially parallel to the centerline. Our thread has 16 threads. Pitch would be 1 divided by 16 equaling 1/16" (.0625") distance from one thread to the next. Metric threads use pitch to identify the distance.

There are many thread forms, standards and terms. Again refer to the book.

Thread Designation, the 3/8"-16UNC-2B thread.

3/8: The nominal size of the thread. It is what we call it, even though the major diameter measures smaller. Our thread calls for a 3/8 OD, which is .375". The standard calls out the diameter to be between .3750 and .3595 diameter depending upon the fit needed. This smaller diameter is to insure that the major diameter will not rub or interfere with the minor diameter.

16: The number of threads per inch. This is the coarse series or 16 TPI. If it were the fine series, it would be 24 threads per inch.

UNC: This is the thread form series designating a 60° angle, flat crest, round root etc. from which all the calculations for cutting and inspection are done. If it were the fine series, it would be UNF.

2: This is the class of fit or in other words how much play or "slop" is there between the nut and bolt. Sometimes there's a lot, like the inexpensive stuff found in a hardware store and sometimes there not much like that found on a micrometer. There are five classes of fit:
1-loose, 2- standard commercial, and 3-close. Classes 4 and 5 are interference fit..

B: Locates where the form is on the cylinder. A is an EXTERNAL thread or typically a bolt.
B is an INTERNAL thread most often a nut.

STANDARD THREADS FOR EACH UNIFIED THREAD SERIES			
NOMINAL SIZE	UNC	UNF	UNEF
0		80	—
1	64	72	—
2	56	64	—
3	48	56	—
4	40	48	—
5	40	44	—
6	32	40	—
8	32	36	—
10	24	32	—
12	24	28	32
$\frac{1}{4}$	20	28	32
$\frac{5}{16}$	18	24	32
$\frac{3}{8}$	16	24	32
$\frac{7}{16}$	14	20	28
$\frac{1}{2}$	13	20	28
$\frac{9}{16}$	12	18	24
$\frac{5}{8}$	11	18	24
$\frac{3}{4}$	10	16	20
$\frac{7}{8}$	9	14	20
1"	8	12	20
UNC—UNIFIED COARSE UNF—UNIFIED FINE UNEF—UNIFIED EXTRA FINE			

Standard sizes for Unified thread series

Table 4 is from the Machinery's Handbook. It gives all the data and tolerances for threads. Note the various thread combinations available for a 3/8 diameter.

Threads less the ¼" diameter are called wire threads and are defined, for example, #10-32UNF-2A. All the terms and standards apply to this thread designation. The #10, as shown, shows it to be a #10 wire size a compared to a 10- designation which would mean it to be a ten inch diameter thread. Wire sizes range from #000 to a #12 size. Size is calculated by the formula

Wire OD = (wire size number x .013") + .060"

OD = (10 x .013) + .060 = .130" + .060" = .190" OD

Metric Threads

A metric thread is designated as a M8x1.0-2g.

M: designates the metric form and standard.
8: the 8mm nominal or OD size.
1.0: identifies the pitch, or the distance apart each thread is from the other.
2g: refers to the tolerance and class of fit. Metric threads have many of these. The lower case g identifies the thread as an external thread or bolt. An upper case letter, such as an H, would designate it as an internal thread or nut.

Fit

A word about fit. Fit is the intentional clearance between mating surfaces of an external and internal thread. On a bolt, this is easily accomplished on a lathe. Because the thread is external, its pitch can be closely cut and controlled by the pitch diameter. A pitch diameter can be measured with a pitch micrometer or the three-wire method and machined to tolerance. On an internal thread, this is very difficult to control and often the fit relies on how well the hole is tapped.

Checking for fit on a nut, we rely on a "Go-NoGo" gage of the correct size and required fit. The Go side enters completely through the hole. The NoGo side only enters the first 1-1/2 turn and no more. If it goes more than that, the thread must be rejected. The fit is controlled by the taps' pitch diameter.

If you refer to "Standard System of Marking Ground Taps", you'll see how a tap is marked. Note the pitch diameter limit symbols. The GH3 limit is the one commonly used in manufacture. For a tighter fit, use an L1, H1 or H2 tap. For looser fits, is the H4, H5 or H6 taps especially if the threads are to be plated. Remember the tap is what controls the internal thread pitch diameter.

Lastly, general practice is that ALL threads are assumed to be, unless duly noted:

A- right hand unless specified LH for Left Hand spiral, and

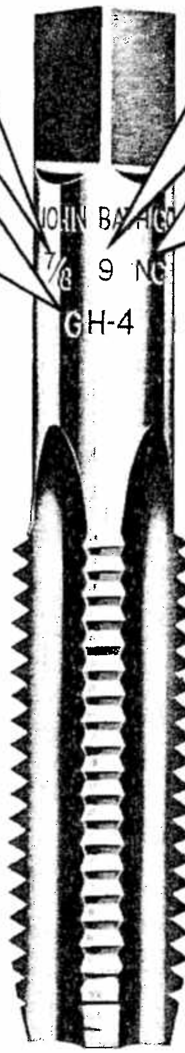
B- single start. This means only one thread is wrapped around the cylinder. Many items have more than on thread wrapped around its cylinder. Look at a peanut butter jar cover. You'll see three or four thread start points. This is called a multiple thread. It lets you close the cover without too much twisting.

Table 4 (*Continued*). Standard Series and Selected Combinations† — Unified Screw Threads

Nominal Size, Threads per Inch, and Series Designation[e]	External[c]								Internal[c]					
	Class	Allowance	Major Diameter			Pitch Diameter		UNR Minor Diam.[a]	Class	Minor Diameter		Pitch Diameter		Major Diameter
			Max[b]	Min	Min[d]	Max[b]	Min	Max (Ref.)		Min	Max	Min	Max	Min
1/4–36 UNS	2A	0.0009	0.2491	0.2436	—	0.2311	0.2280	0.2161	2B	0.220	0.226	0.2320	0.2360	0.2500
1/4–40 UNS	2A	0.0009	0.2491	0.2440	—	0.2329	0.2300	0.2193	2B	0.223	0.229	0.2338	0.2376	0.2500
1/4–48 UNS	2A	0.0008	0.2492	0.2447	—	0.2357	0.2330	0.2243	2B	0.227	0.232	0.2365	0.2401	0.2500
1/4–56 UNS	2A	0.0008	0.2492	0.2451	—	0.2376	0.2350	0.2280	2B	0.231	0.235	0.2384	0.2417	0.2500
5/16–18 UNC	1A	0.0012	0.3113	0.2982	—	0.2752	0.2691	0.2452	1B	0.252	0.265	0.2764	0.2843	0.3125
	2A	0.0012	0.3113	0.3026	0.2982	0.2752	0.2712	0.2452	2B	0.252	0.265	0.2764	0.2817	0.3125
	3A	0.0000	0.3125	0.3038	—	0.2764	0.2734	0.2464	3B	0.2520	0.2630	0.2764	0.2803	0.3125
5/16–20 UN	2A	0.0012	0.3113	0.3032	—	0.2788	0.2748	0.2518	2B	0.258	0.270	0.2800	0.2852	0.3125
	3A	0.0000	0.3125	0.3044	—	0.2800	0.2770	0.2530	3B	0.2580	0.2680	0.2800	0.2839	0.3125
5/16–24 UNF	1A	0.0011	0.3114	0.3006	—	0.2843	0.2788	0.2618	1B	0.267	0.277	0.2854	0.2925	0.3125
	2A	0.0011	0.3114	0.3042	—	0.2843	0.2806	0.2618	2B	0.267	0.277	0.2854	0.2902	0.3125
	3A	0.0000	0.3125	0.3053	—	0.2854	0.2827	0.2629	3B	0.2670	0.2754	0.2854	0.2890	0.3125
5/16–27 UNS	2A	0.0010	0.3115	0.3048	—	0.2874	0.2839	0.2674	2B	0.272	0.281	0.2884	0.2929	0.3125
5/16–28 UN	2A	0.0010	0.3115	0.3050	—	0.2883	0.2849	0.2689	2B	0.274	0.282	0.2893	0.2937	0.3125
	3A	0.0000	0.3125	0.3060	—	0.2893	0.2867	0.2699	3B	0.2740	0.2807	0.2893	0.2926	0.3125
5/16–32 UNEF	2A	0.0010	0.3115	0.3055	—	0.2912	0.2880	0.2743	2B	0.279	0.286	0.2922	0.2964	0.3125
	3A	0.0000	0.3125	0.3065	—	0.2922	0.2898	0.2753	3B	0.2790	0.2847	0.2922	0.2953	0.3125
5/16–36 UNS	2A	0.0009	0.3116	0.3061	—	0.2936	0.2905	0.2785	2B	0.282	0.289	0.2945	0.2985	0.3125
5/16–40 UNS	2A	0.0009	0.3116	0.3065	—	0.2954	0.2925	0.2818	2B	0.285	0.291	0.2963	0.3001	0.3125
5/16–48 UNS	2A	0.0008	0.3117	0.3072	—	0.2982	0.2955	0.2869	2B	0.290	0.295	0.2990	0.3026	0.3125
3/8–16 UNC	1A	0.0013	0.3737	0.3595	—	0.3331	0.3266	0.2992	1B	0.307	0.321	0.3344	0.3429	0.3750
	2A	0.0013	0.3737	0.3643	0.3595	0.3331	0.3287	0.2992	2B	0.307	0.321	0.3344	0.3401	0.3750
	3A	0.0000	0.3750	0.3656	—	0.3344	0.3311	0.3005	3B	0.3070	0.3182	0.3344	0.3387	0.3750
3/8–18 UNS	2A	0.0013	0.3737	0.3650	—	0.3376	0.3333	0.3076	2B	0.315	0.328	0.3389	0.3445	0.3750
3/8–20 UN	2A	0.0012	0.3738	0.3657	—	0.3413	0.3372	0.3143	2B	0.321	0.332	0.3425	0.3479	0.3750
	3A	0.0000	0.3750	0.3669	—	0.3425	0.3394	0.3155	3B	0.3210	0.3297	0.3425	0.3465	0.3750
3/8–24 UNF	1A	0.0011	0.3739	0.3631	—	0.3468	0.3411	0.3243	1B	0.330	0.340	0.3479	0.3553	0.3750
	2A	0.0011	0.3739	0.3667	—	0.3468	0.3430	0.3243	2B	0.330	0.340	0.3479	0.3528	0.3750
3/8–24 UNF	3A	0.0000	0.3750	0.3678	—	0.3479	0.3450	0.3254	3B	0.3300	0.3372	0.3479	0.3516	0.3750
3/8–27 UNS	2A	0.0011	0.3739	0.3672	—	0.3498	0.3462	0.3298	2B	0.335	0.344	0.3509	0.3556	0.3750
3/8–28 UN	2A	0.0011	0.3739	0.3674	—	0.3507	0.3471	0.3313	2B	0.336	0.345	0.3518	0.3564	0.3750
	3A	0.0000	0.3750	0.3685	—	0.3518	0.3491	0.3324	3B	0.3360	0.3426	0.3518	0.3553	0.3750
3/8–32 UNEF	2A	0.0010	0.3740	0.3680	—	0.3537	0.3503	0.3368	2B	0.341	0.349	0.3547	0.3591	0.3750
	3A	0.0000	0.3750	0.3690	—	0.3547	0.3522	0.3378	3B	0.3410	0.3469	0.3547	0.3580	0.3750
3/8–36 UNS	2A	0.0010	0.3740	0.3685	—	0.3560	0.3528	0.3409	2B	0.345	0.352	0.3570	0.3612	0.3750
3/8–40 UNS	2A	0.0009	0.3741	0.3690	—	0.3579	0.3548	0.3443	2B	0.348	0.354	0.3588	0.3628	0.3750
0.390–27 UNS	2A	0.0011	0.3889	0.3822	—	0.3648	0.3612	0.3448	2B	0.350	0.359	0.3659	0.3706	0.3900
7/16–14 UNC	1A	0.0014	0.4361	0.4206	—	0.3897	0.3826	0.3511	1B	0.360	0.376	0.3911	0.4003	0.4375
	2A	0.0014	0.4361	0.4258	0.4206	0.3897	0.3850	0.3511	2B	0.360	0.376	0.3911	0.3972	0.4375
	3A	0.0000	0.4375	0.4272	—	0.3911	0.3876	0.3525	3B	0.3600	0.3717	0.3911	0.3957	0.4375
7/16–16 UN	2A	0.0014	0.4361	0.4267	—	0.3955	0.3909	0.3616	2B	0.370	0.384	0.3969	0.4028	0.4375
	3A	0.0000	0.4375	0.4281	—	0.3969	0.3935	0.3630	3B	0.3700	0.3800	0.3969	0.4014	0.4375
7/16–18 UNS	2A	0.0013	0.4362	0.4275	—	0.4001	0.3958	0.3701	2B	0.377	0.390	0.4014	0.4070	0.4375
7/16–20 UNF	1A	0.0013	0.4362	0.4240	—	0.4037	0.3975	0.3767	1B	0.383	0.395	0.4050	0.4131	0.4375
	2A	0.0013	0.4362	0.4281	—	0.4037	0.3995	0.3767	2B	0.383	0.395	0.4050	0.4104	0.4375
	3A	0.0000	0.4375	0.4294	—	0.4050	0.4019	0.3780	3B	0.3830	0.3916	0.4050	0.4091	0.4375
7/16–24 UNS	2A	0.0011	0.4364	0.4292	—	0.4093	0.4055	0.3868	2B	0.392	0.402	0.4104	0.4153	0.4375
7/16–27 UNS	2A	0.0011	0.4364	0.4297	—	0.4123	0.4087	0.3923	2B	0.397	0.406	0.4134	0.4181	0.4375
7/16–28 UNEF	2A	0.0011	0.4364	0.4299	—	0.4132	0.4096	0.3938	2B	0.399	0.407	0.4143	0.4189	0.4375
	3A	0.0000	0.4375	0.4310	—	0.4143	0.4116	0.3949	3B	0.3990	0.4051	0.4143	0.4178	0.4375
7/16–32 UN	2A	0.0010	0.4365	0.4305	—	0.4162	0.4128	0.3993	2B	0.404	0.411	0.4172	0.4216	0.4375
	3A	0.0000	0.4375	0.4315	—	0.4172	0.4147	0.4003	3B	0.4040	0.4094	0.4172	0.4205	0.4375
1/2–12 UNS	2A	0.0016	0.4984	0.4870	—	0.4443	0.4389	0.3992	2B	0.410	0.428	0.4459	0.4529	0.5000
	3A	0.0000	0.5000	0.4886	—	0.4459	0.4419	0.4008	3B	0.4100	0.4223	0.4459	0.4511	0.5000
1/2–13 UNC	1A	0.0015	0.4985	0.4822	—	0.4485	0.4411	0.4069	1B	0.417	0.434	0.4500	0.4597	0.5000
	2A	0.0015	0.4985	0.4876	0.4822	0.4485	0.4435	0.4069	2B	0.417	0.434	0.4500	0.4565	0.5000
	3A	0.0000	0.5000	0.4891	—	0.4500	0.4463	0.4084	3B	0.4170	0.4284	0.4500	0.4548	0.5000
1/2–14 UNS	2A	0.0015	0.4985	0.4882	—	0.4521	0.4471	0.4135	2B	0.423	0.438	0.4536	0.4601	0.5000

All dimensions in inches. † Use UNS threads only if Standard Series do not meet requirements (see p. 1479). See footnotes *a*, *b*, *c*, *d*, and *e* at end of table.

STANDARD SYSTEM OF MARKING GROUND THREAD TAPS



NOMINAL SIZE

THREADS PER INCH

PITCH DIAMETER LIMIT SYMBOLS

"G" signifies Ground Thread.
"H" (High) signifies the pitch diameter is above basic.
(**"L"** (Low) replaces the letter "H" to signify the pitch diameter is below basic.)

The numeral following "H" or "L" signifies the number of .0005″ steps above or below the basic pitch diameter.

Standard taps 1″ diameter and smaller are ground to a .0005″ tolerance on the pitch diameter and are marked with one of these corresponding pitch diameter limits:

- **L1** = Basic to basic minus .0005″.
- **H1** = Basic to basic plus .0005″.
- **H2** = Basic plus .0005″ to basic plus .0010″.
- **H3** = Basic plus .0010″ to basic plus .0015″.
- **H4** = Basic plus .0015″ to basic plus .0020″.
- **H5** = Basic plus .0020″ to basic plus .0025″.
- **H6** = Basic plus .0025″ to basic plus .0030″.

Standard taps larger than 1″ diameter are ground to a .0010″ tolerance on the pitch diameter and are marked with this pitch diameter limit:

- **H4** = Basic plus .0010″ to basic .0020″.

SPECIAL TAPS

Special Ground Thread taps made to the pitch diameter limits as shown above will also be marked with the corresponding limit number.

Where special taps are ordered without a pitch diameter or limit number given, the pitch diameter will normally be determined from Table 331 and marked with the appropriate "H" or "L" limit number.

When the tap P.D. is specified to be a certain amount oversize or undersize, this amount is added or subtracted from the basic pitch diameter of the nominal size tap. This dimension becomes the minimum pitch diameter to which the standard tolerance for the nominal size is added. Oversize or undersize taps are marked with the nominal size and pitch and the amount the minimum pitch diameter is over or under basic. For example: ½-13 + .010.

Taps having multiple threads are marked with the nominal size, number of threads per inch, form of thread, and lead designated in fractions; also double, triple, etc. For example: A special tap, 1″-8, double thread with National Form of Thread will be marked:

1″-8 NS Double
¼″ Lead

THREAD FORM SYMBOLS

NC	American National Coarse Thread Series.
UNC	Unified Coarse Thread Series.
NF	American National Fine Thread Series.
UNF	Unified Fine Thread Series.
NEF	American National Extra Fine Thread Series.
UNEF	Unified Extra Fine Thread Series.
N	American National 8, 12 and 16 Thread Series.
UN	Unified Constant-Pitch Thread Series.
UNR	Unified Constant-Pitch Thread Series with a 0.108P to 0.144P Controlled Root Radius.
UNJ	Unified Thread Series with a 0.15011P to 0.18042P Controlled Root Radius.
UNM	Unified Miniature Thread Series.
NS	American National Thread—Special.
UNS	Unified Thread—Special.
NPT	American Standard Taper Pipe Thread.
NPTF	Dryseal American Standard Taper Pipe Thread (Fuel).
PTF	Dryseal SAE Short Taper Pipe Thread.
ANPT	Aeronautical National Form Taper Pipe Thread (MIL-P-7105).
NPS	American Standard Straight Pipe Thread.
NPSC	American Standard Straight Pipe Thread in Pipe Couplings (Tap marked NPS).
NPSF	Dryseal American Standard Internal Straight Pipe Thread (Fuel).
NPSH	American Standard Straight Pipe Thread for Hose Couplings and Nipples.
NPSI	Dryseal American Standard Intermediate Internal Straight Pipe Thread.
NPSL	American Standard Straight Pipe Thread for Loose Fitting Mechanical Joints with Locknuts.
NPSM	American Standard Straight Pipe Thread for Free Fitting Mechanical Joints for Fixtures (Tap marked NPS).
NPTR	American Standard Taper Pipe Thread for Railing Joints (Tap marked NPT).
NGO	National Gas Outlet Thread.
NH	American National Hose Coupling and Fire Hose Coupling Threads.
AMO	American Standard Microscope Objective Thread.
ACME C	Acme Thread—Centralizing.
ACME G	Acme Thread—General Purpose.
N BUTT	American Buttress Screw Thread.
STI	Special Threads for Helical Coil Wire Screw Thread Inserts.

Symbols used for British Threads are:

BSW	British Standard Whitworth Coarse Thread Series.
BSF	British Standard Fine Thread Series.
BSP	British Standard Taper Pipe Thread.
BSPP	British Standard Pipe (Parallel) Thread.
WHIT	Whitworth Standard Special Thread.
BA	British Association Standard Thread.

Courtesy of John Bath & Company, Inc.

THREAD INSPECTION

Threads are inspected in many ways. The inspection method is dependent upon thread use, the required precision and the process used in creating it. Basically, all threads and threads properties are measured from the pitch diameter. Lets look at some commonly used methods.

Mating Part

If two parts need to assemble and do not need to be interchangeable with other parts, machine either the internal or external thread first, then machine the other to fit.

Nut/Bolt

When machining an external tread and a precise fit is not a major criteria, as on the hammer handle, cutting the thread until the nut screws on freely is an acceptable inspection method.

Thread Gage

Thread gages are precise thread forms that screw onto the mating form. Gages are either "GO" or "NO-GO". That means that the GO side will screw in while the NO-GO won't. They are good for high volume mass production work. These gages have two drawbacks. The first is they don't show you if and how the thread profile is deformed, but more importantly, they don't show you how much material you need to remove in order to bring the pitch diameter to tolerance.

Three-wire Method

This simple, effective and inexpensive thread inspection method. It requires three wires of a specific size, a regular micrometer, a thread table and solving two formulas. This method is 95% accurate. After measuring the pitch diameter, the measurements show you how much the thread form is in or out of tolerance. If you're machining an external thread, you know exactly how much material needing to be machined. The following page shows the calculations for the 3/8"-16b thread.

Other Methods

There are special gages that can measure pitch and thread deformities. These are used on thread requiring extreme precision. These gages are very expensive and can measure a small range of diameters.

Optical comparators can be used, but are generally used to verify the thread form, but have limited use in inspecting pitch and pitch diameter.

Pitch micrometers directly measure the pitch diameter. Each micrometer can measure a few thread pitches within its anvil geometry.

THREE-WIRE THREAD INSPECTION

The use of three wires and a standard micrometer is a simple and effective means of inspection. The best part is that as the machinist, you can measure exactly how much material needing to be machined in order to machine the thread to tolerance. Its only drawback is that you can't measure deformation. Deformation is how much the thread is misshape from the perfect geometric shape. In order to use the three-wire method, some math is necessary. You'll need Machinery's Handbook and a calculator. Let's calculate the required values for the 3/8"–16 thread.

From the handbook, we find the table for the three wires giving us the formulas. Note that there are formulas for different thread forms. We will use the 60° thread from formula. This is also the formula used for metric threads as its thread from is also 60°.

The Wire Size formula is:

W = .57735 * pitch, where W is the needed wire diameter and p is the pitch, 1/16 = .0625.

W = .57735 * .0625 = .0360 diameter wire size. Note: when calculating MOW, use the ACTUAL wire diameter, otherwise your calculations will be in error for the wire diameter used.

The Measurement Over the Wires formula

There are two formulas you can use to measure a UNC thread. The pitch diameter is the most controlled form of the thread therefore it should be the formula to use. Another reason is that many other variables can be eliminated.

M = E – (.86603 * P) + (3 * W)

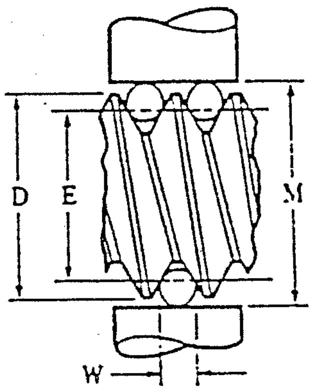
M: Measurement Over the Wires
E: pitch diameter as found in the handbook for the required thread and fit.
P: pitch, one divided by the number of threads per inch.
W: Wire size, use the actual size of the wires.

E: .3331 max, .3287 min for a 2A fit.
P: .0625
W: .0360

Calculating Max M;
M = .3331 – (.86603* .0625) + (3 * .0360)
.3331 – .0541 + .108
M = .387 Max diameter
-.0044 tolerance (.3331 - .3287 = .0044)
.3826 Min diameter

Now, using the three wires and a 0-1" micrometer, the thread can be precisely measured. The machinist knows exactly how much material can be removed in order to machine it to tolerance.

Formulas for Checking Pitch Diameters of Screw Threads



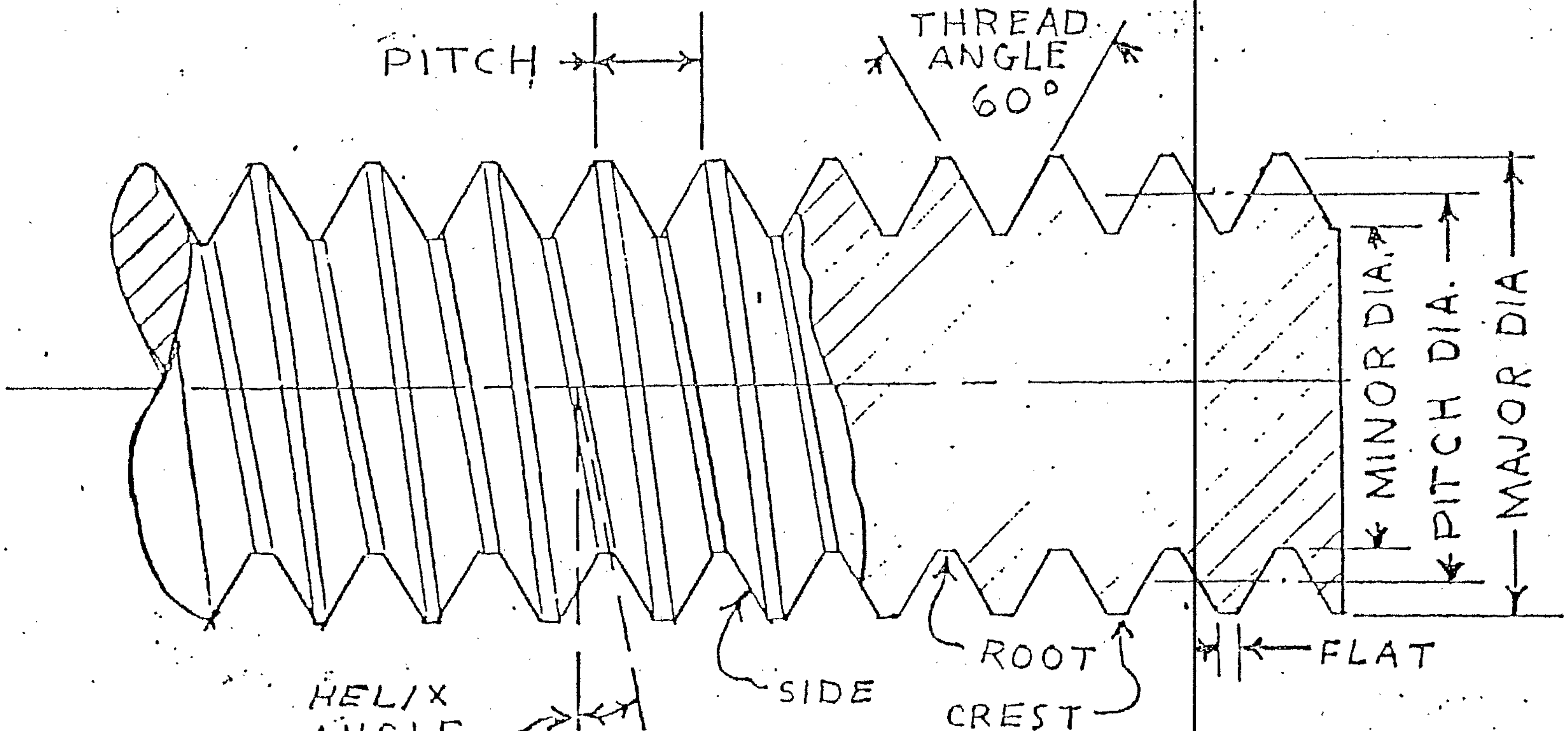
The formulas below do not compensate for the effect of the lead angle upon measurement M, but they are sufficiently accurate for checking standard single-thread screws unless exceptional accuracy is required. See accompanying information on effect of lead angle; also matter relating to measuring wire sizes, accuracy required for such wires, and contact or measuring pressure. The approximate best wire size for pitch-line contact may be obtained by the formula

$$W = 0.5 \times \text{pitch} \times \sec \tfrac{1}{2} \text{ included thread angle}$$

For 60-degree threads, $W = 0.57735 \times \text{pitch}$.

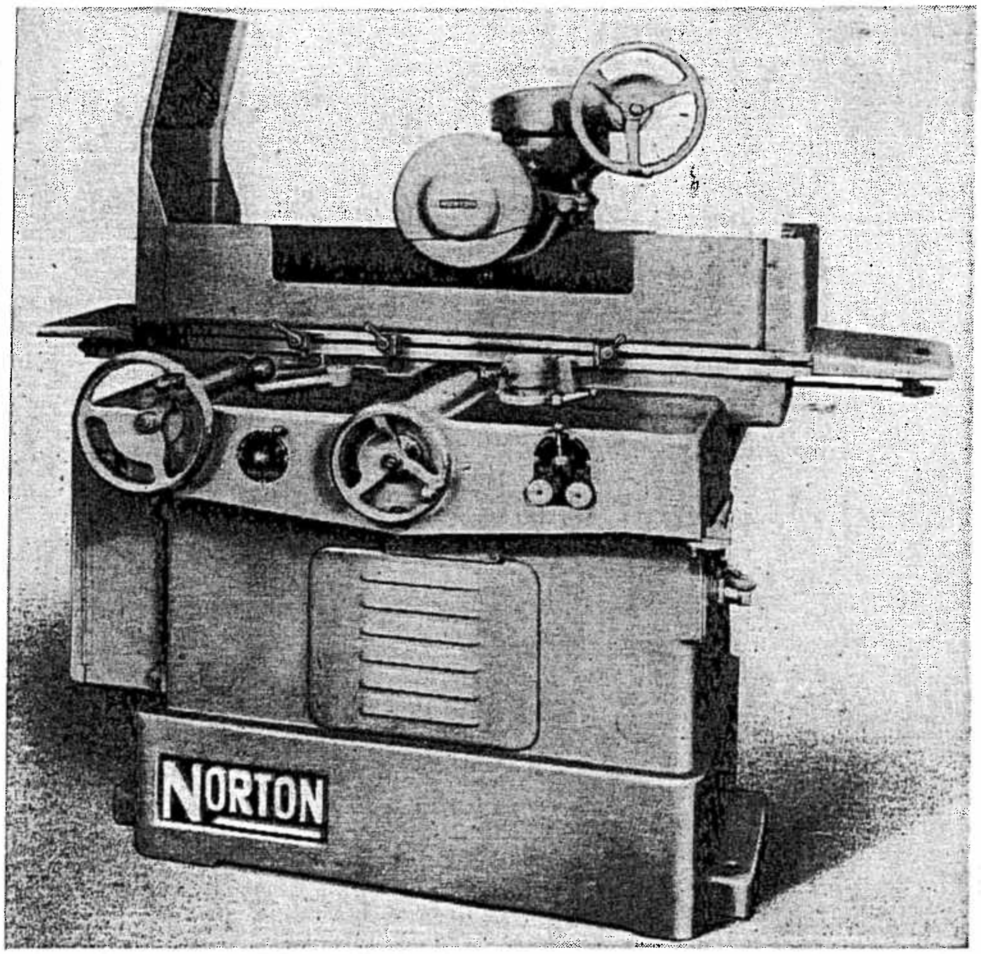
Form of Thread	Formulas for determining measurement M corresponding to correct pitch diameter and the pitch diameter E corresponding to a given measurement over wires.*
American National Standard Unified	When measurement M is known. $E = M + 0.86603P - 3W$ When pitch diameter E is used in formula. $M = E - 0.86603P + 3W$ The American Standard formerly was known as U.S. Standard.
British Standard Whitworth	When measurement M is known. $E = M + 0.9605P - 3.1657W$ When pitch diameter E is used in formula. $M = E - 0.9605P + 3.1657W$
British Association Standard	When measurement M is known. $E = M + 1.1363P - 3.4829W$ When pitch diameter E is used in formula. $M = E - 1.1363P + 3.4829W$
Lowenherz Thread	When measurement M is known. $E = M + P - 3.2359W$ When pitch diameter E is used in formula. $M = E - P + 3.2359W$
Sharp V-Thread	When measurement M is known. $E = M + 0.86603P - 3W$ When pitch diameter E is used in formula. $M = E - 0.86603P + 3W$
International Standard	Use the formula given above for the American National Standard Unified Thread.
Pipe Thread	See accompanying paragraph on Measuring Taper Screw Threads by Three-wire Method.
Acme and Worm Threads	See Buckingham Formulas; also Three-wire Method for Checking Thickness of Acme Threads.
Buttress Form of Thread	Different forms of buttress threads are used. See paragraph on wire method applied to buttress threads.

* The wires must be lapped to a uniform diameter and it is very important to insert in the rule or formula the wire diameter as determined by precise means of measurement. Any error will be multiplied. See paragraph on Wire Sizes for Checking Pitch Diameters.



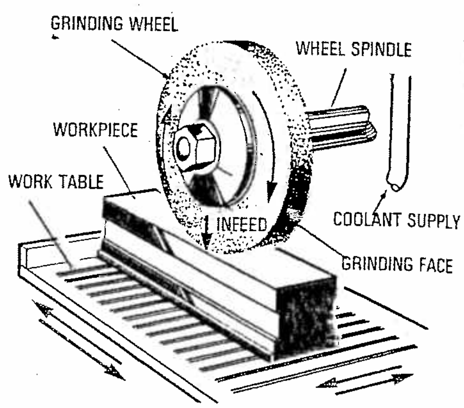
$$PITCH = \frac{1}{\#\,Thds./Inch}$$

THE SURFACE GRINDER



THE SURFACE GRINDER

The surface grinder is generally used to make flat surfaces flat, smooth, to precision size and to a high surface finish. There are many types of grinders, such as cylindrical, tool and cutter, pedestal, internal and external grinders and the like. It is used to grind soft or hardened surfaces, steel or non-ferrous metals. The cutting action occurs because an abrasive wheel rubs against the material and removes little amounts from it. A typical surface grinder is shown below. The work is usually held a magnetic chuck. Turn the chuck on and magnetism holds the ferrous (steel) work securely. It doesn't work on metals like aluminum, brass and all non-ferrous material.



Courtesy The Carborundum Company

Horizontal spindle with reciprocating table

Base: Holds all the major parts. It must be massive and solid so it isn't affected by heat and vibration, it can absorb cutting forces and that it has a reservoir for coolant.

Table: It holds the magnetic chuck. It reciprocates (goes back and forth) and transverses (moves in and out) allowing the work to be ground.

Magnetic chuck: The holding device that secures the work for grinding. Make sure you turn it on!

Grinding wheel: Is the cutting tool that does the work.

Wheel Feed Handle: brings the wheel down into the work for cutting.

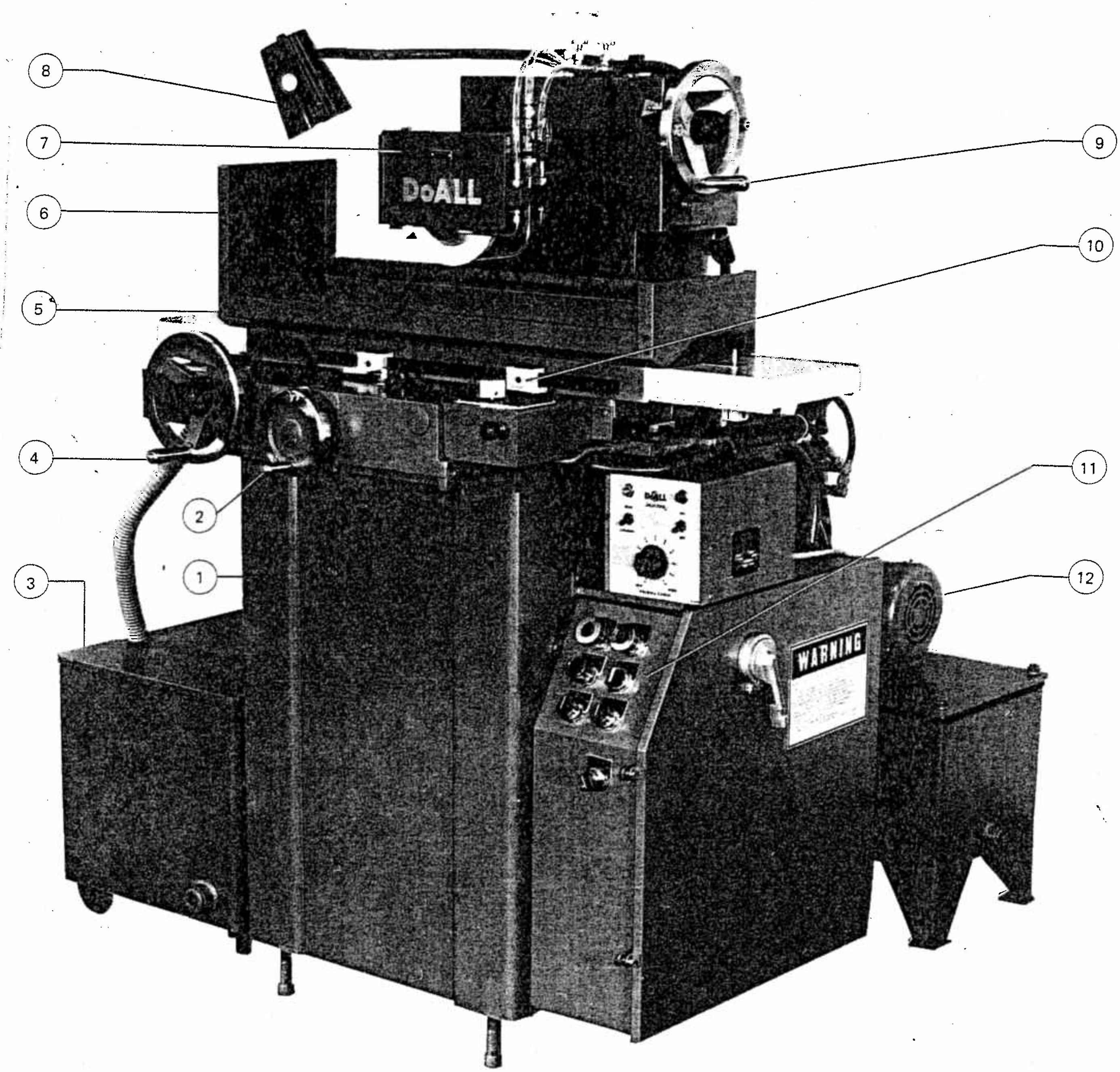
Table Traverse Wheel: Allows hand motion of the wheel to reciprocate.

Cross-feed Handwheel: Allows in and out motion by hand.

Trip Dogs: Sets the reciprocating distance of the table. Allows the table to automatically reverse direction.

Coolant: A liquid cooling medium needed to remove the heat generated by the grinding action. It is necessary so the generated heat isn't allowed to expand the metal being ground. If that happens, the metal shrinks upon cooling and leaves a depression in the steel.

SURFACE GRINDER



1 BASE
2 CROSSFEED HANDWHEEL
3 COOLANT TANK
4 TABLE FEED HANDWHEEL

5 TABLE
6 SPLASH GUARD
7 WHEEL GUARD
8 WORKLAMP

9 DOWNFEED HANDWHEEL
10 TABLE TRAVERSE DOG
11 CONTROL PANEL
12 HYDRAULIC MOTOR

Courtesy of DoAll Company

WHEELS

The grinding wheel is what cuts the material. Wheels are made of two major components – the grit, which does the cutting and the bond or glue, which holds the grit together. Wheels are made from either Aluminum Oxide or Silicon Carbide grit material. There are special grits for special grinding operations. Depending upon the application, the bond becomes important. It must withstand coolant, rough handling, high RPM's and finish required.

A typical wheel marking could be A 60 – J8V. Let's see what they mean.

A represents the grit type. A is Aluminum Oxide. C is Silicon Carbide. Others are for special operations. Aluminum oxide is used to grind hard metals and steels. It is softer than silicon carbide. Silicon carbide is used to grind cast iron and non-ferrous metals, like aluminum and brass. Notice that a soft wheel is used on hard materials while a hard wheel is used on soft materials. As a wheel grinds, it is forced against the work. Heat is created and particles are removed from the work. This causes heat and friction. Particles imbed themselves between the spaces in the wheel and dull the wheel. As the wheel dulls, it breaks down and exposes new sharp grit particles. This is a function of grit and bond. When the wheel can no longer break down, it must be dressed, that is use a diamond to remove the embedded particles.

60 represents grit size. The higher the number, the finer the wheel, think of sandpaper. Fast grinding uses a 40 to 60 grit wheel while fine finish and precision requires a wheel from 80 grit and higher.

J is the wheel grade. Grade is how hard the wheel is. This is a function of bond holding the grit together. The stronger the bond, the harder the wheel, the harder to remove a lot of work fast.

8 represent structure. When a wheel is manufactured, the mixture of grit particles and bond, is mixed together then squeezed together in a mold. Depending on the ratio of grit to bond and pressure used to squeeze the mold determines how far apart the grit spacing is. Spacing is important is important because it holds/releases particles and has a spot to hold some coolant. The mixture of released grinding particles and coolant is called "swarf". (If nothing else, a great scrabble word.)

V is the bond or glue. Most wheels used are vitrified bond. There are also rubber, resinoid, shellac and metal bonds. The bond has to withstand coolant, abuse, high RPM's and need to be selected for the operation being done.

Selecting a wheel becomes a compromise of give and take. Consult a manufacturer's representative or a manual, such as the Machinery's Handbook.

Other markings that you need to pay attention to are these: 7 x ½ x 1-1/4. This identifies the wheel size. Wheels come in many different shapes, but we'll pay attention to a straight wheel.

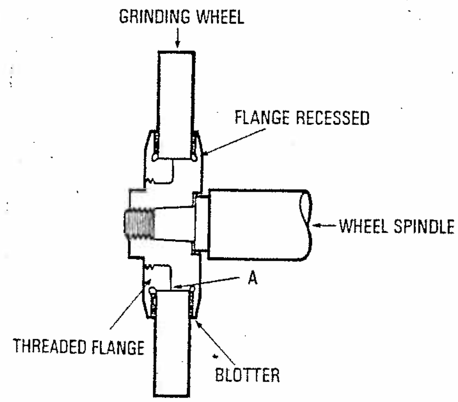
The 7 is the wheel Outside Diameter. The ½ is the wheel's width, while the 1-1/4 is the hole diameter. If the hole diameter is too big for the shaft, spacers are used.

Whether the wheel is used on a pedestal grinder or a surface grinder, you need to pay attention to these details.

Ring Test

The first rule in mounting a wheel is to test for cracks – no matter what size the wheel or application. The wheel is held through the hole by one finger. A wooden hammer handle is used to <u>gently</u> hit the side of the wheel. If the wheel is safe and not cracked, it will ring like a bell. If the wheel is cracked, is won't ring at all. It would sound more like a "thug". That means it is cracked, unsafe and could explode on you if you mount the wheel and turn the spindle on. Break up the wheel so no one uses it.

Courtesy The Carborundum Company
Testing a grinding wheel



Courtesy "Precision Surface Grinding," Delmar Publishers Inc.

A grinding wheel properly mounted on the grinding spindle

Mounting the Wheel

After the wheel is checked for cracks, it is ready to be mounted. First, remove the old wheel. Remember that some arbors have LEFT hand threads. This prevents the nut from coming loose by using centrifucal force to help keep the nut tight. Wedge the wheel with a stick and turn the nut in the correct direction. Clean the spindle, flanges and nut. Mount a felt washer on each side of the wheel. This does a couple of things. If the wheel is not perfectly smooth, or has a high spot, the flange, when tightened, sets up a stress area and could cause the wheel to crack. The felt washer helps to minimize this effect. Put the flanges on. They help to equalize the holding forces. Use the correct spacer between the wheel ID and the spindle. Mount the wheel onto the spindle. Tighten the nut firmly. You don't have to Superman, but the nut needs to be tight. Replace the wheel guard. It is important.

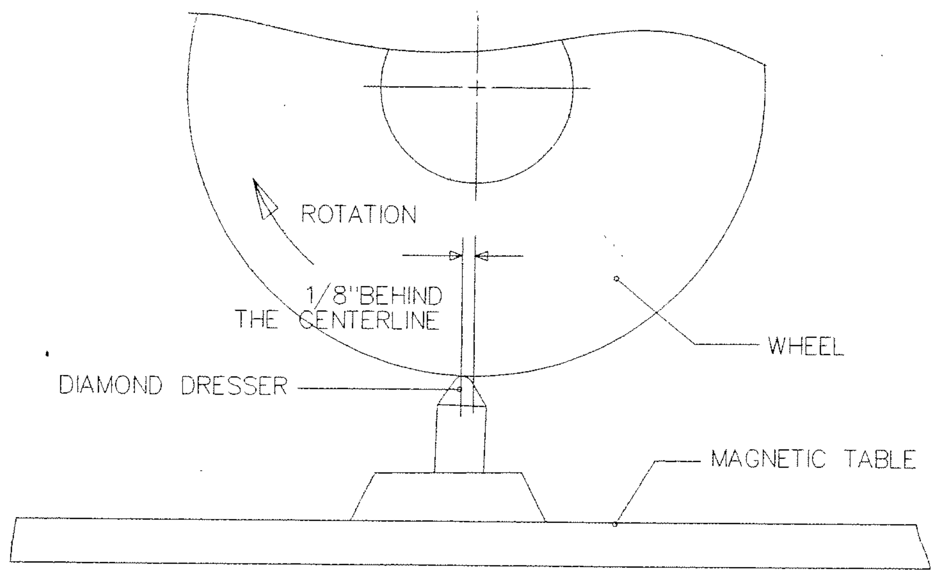
Truing and Dressing

After the wheel is mounted, the circumference has to run true with the spindle centerline. On a pedestal grinder, run a mechanical wheel dresser back and forth across the face to make the wheel run the true with the spindle. On a surface grinder, a diamond dresser is used. See further on.

During grinding, metal particles imbed themselves between the spaces of the wheel. If not removed, the wheel burns the metal and dulls up. Taking a wheel dresser or a diamond, clean the surface to expose new and sharp particles. When you have problems, this is the first step in problem solving. Most times, you'll dress and true at the same time.

Diamond dressing

Dressing and truing on a surface grinder is done with a diamond. Clean the chuck, place the diamond on the chuck over a piece of paper. Turn the chuck on. IMPORTANT. Position the diamond from 1/8" to ½" to the LEFT of the wheel centerline. If the wheel pulls the diamond off the chuck, it will send it flying to the rear, it won't go under the wheel and cause you great problems. It is for SAFETY. Bring the wheel so it touches the diamond. Take cuts .001" to .002" deep per pass. Move the diamond in/out to clean the face. Continue until the face is clean. If you have coolant, use coolant when dressing the wheel. This conditions the wheel utilizing the same parameters that the wheel will be used when grinding.



GRINDING THE WORK

Once the wheel has been dressed, you can grind the work. First, if the work has been hardened, you must remove the scale. Scale prevents the work from sitting flat on the magnet and fails to provide full power to hold the work down. Use sandpaper or emery cloth to clean up two adjacent sides. Measure the work so you'll know how much stock to remove per side. On the hammerhead, this would be about .005" per side. Place the work so it crosses as many field lines as possible. If the work is short, place thin pieces of metal around the part to "nest" it securely. Make sure to turn the magnet "ON" before proceeding. Grab the work and see if you can move it. If necessary, put more metal to better nest the part.

Turn the machine on. Move the table by hand and set the trip dogs to the point near where they will reverse the table direction. The wheel should travel beyond the work by an inch or so at each end. Turn on automatic travel and see where the reverse occurs. Adjust if necessary.

Bring the wheel down so it just makes contact with the work. Turn on the coolant and take your first cut at this height. If you have a high spot, you'll take off only a small cut rather than a big chunk and send the work flying. The rear shield is there to stop flying objects. Grind the work until the wheel goes off either side. Lower the wheel by .001" maximum per cut. Grinding is a slow process. Continue until you've removed the required amount per side. Use a depth mike to measure the thickness or the depth slide of your vernier. Don't take the part off to measure because you'll end up grinding the dimension undersize. Stop. Remove the work. Clean off the swarf from the work and table. Turn the part on its side. Repeat the process. Once two sides are done, you can grind the opposite side parallel to the other and to finish size. Clean up the swarf.

If you decide to grind the angled surfaces of the head, use a precision vise. Set the work to the correct angle and tightened the work. Choke up the work as much as practical. Use a square and align the vise near square to the chuck. Turn the chuck on and grind the angle clean. Use vises, angle plates or fixtures when grinding features you can't hold directly on the chuck. Think and work SAFE on this machine.