MAKING LOADING DIES & BULLET MOLDS

By Harold Hoffman

H & P Publishing 7174 Hoffman Rd. San Angelo, Tx. 76905

INTRODUCTION

I have had numerous request from my readers to include a book on making loading dies as well as bullet molds. What this involves is basic Tool & Die work. Making loading dies as well as other tools are simple once you learn the fundamentals of tool making. There are quite a few books out, but I have found that few go into the step-by-step process of leading the individual through the actual process of machining tools that are needed for making various items.

Though this book is basic, I try to lead you through the different processes. You will find that all the machining operations are about the same, so once you have mastered the basic, you can go on to the next stage.

I cover in this book turning of materials, milling operations for making tools, and the final heat treating of the part to be able to use it. Many of my other books go into much greater detail and information, but they are for craftsperson that has gained more experience.

Once you have learned the basic, you will be able to do many of the other operations described in these books. If you learn to do what is covered in this book, you will then be able to make just about any type of metal tools that is needed to make loading dies, etc.

This book may seem to be a little vague at time for the inexperience reader, but once the reader starts making the parts described, it all falls in place. I welcome any suggestions on how to improve the books and ideas on other books that they feel that is needed

If the reader follows the instructions, he or she will be able to become a good machinist in time. All it takes is much time patience, and practice to become proficient, and this is true of any new skill. I have many readers tell me it sounded too simple. Well, becoming a machinist is simple, but it takes time. So don't expect to become an expert machinist over night.

Harold Hoffman

TABLE OF CONTENTS

CHAPTER 1 EQUIPMENT AND TOOLS

In the introduction I listed a few machines that are needed, to make what you need. What is needed will allow you to machine and make parts.

LATHE

Your lathe should have at least a 2 foot bed. The hole through the head stock should be at least 1½ inch, as you will need to center the steel in the head stock if it sticks all the way through.

Lathes used in manufacturing can be classified as engine, and toolroom. On an engine lathe, the mechanisms for holding and rotating the workpiece include the headstock and the tailstock. The bed and ways are horizontal, and the headstock is mounted at one end of the bed. When standing in front of an engine lathe, the headstock is always located at the operator’s left. The headstock houses and supports the spindle and the means for rotating the spindle.

Engine Lathes are the type most frequently used in manufacturing. They are heavy-duty machine tools with all the components described previously and have power drive for all tool movements except on the compound rest. In most cases the bed is mounted on two pedestal legs. They range in size from 12 to 24 inches swing and from 24 to 48 inches center distance. Most have chip pans and a built-in coolant circulating system.

The axis of rotation of the spindle is accurately parallel to the ways of the bed. The spindle rotates the workpiece and often completely supports it. The spindle must be supported in excellent bearings, which will keep all radial and axial movements to a minimum. It is necessary to be able to change the rpm of the spindle to obtain different needed cutting speeds. With modem engine lathes, this is usually accomplished by shifting gears with the aid of levers that extend to the outside by the head stock. The spindle is hollow so that a bar being machined at one end may extend through the spindle. The inner end of the spindle, or the end that is closest to the tailstock, is provided with means for fastening certain work holding devices to it.

Tool Room Lathes have greater accuracy and, usually, a wider range of speeds and feeds than ordinary engine lathes. They are designed to have greater versatility to meet the requirements of tool and die work and they often have a continuously variable spindle speed range. They have shorter beds than ordinary engine lathes of comparable swing, since they are generally used for machining small parts. They may be either bench or pedestal type.

Supporting work in lathes. Three methods commonly are used for supporting workpieces in lathes:

1. Held between centers.
2. Mounted on a face plate.
3. Held in a chuck.

In the above methods the workpiece is rotated during machining. The tailstock is movable on the ways, and it can be clamped to the bed at various locations. The tailstock contains a sliding cylindrical member, known as a quill. This quill is sometimes called the tailstock spindle, even though it cannot rotate.

Movement of the tailstock quill is accurately parallel to the rotational axis of the spindle in the headstock. This movement is limited to a few inches, depending upon the size of the engine lathe. The movement of the tailstock quill is controlled by a sufficiently large screw, which is turned by a handwheel at the outer end of the tailstock. Once set, the quill may be clamped to remain in a desired position. At its inner end, a tailstock quill has a tapered hole for supporting a center, which in turn may be used to locate and support a workpiece.

The axis of this tapered hole must be accurately parallel to the rotational axis of the spindle, and for the majority of engine lathes work these two axes must be accurately in line. It is possible to adjust the upper portion of the tailstock so that the axis of the tailstock quill can be moved horizontally out of line. It always remains parallel to the rotational axis of the spindle.

This adjustment is useful for turning tapers by the method in which the tailstock center is purposely offset. Since drills, reamers, and other cutting tools are commonly located and supported in the tapered hole of the tailstock quill, a tailstock serves in a dual capacity. Movement of these cutting tools is controlled by the handwheel that moves the quill.

ENGINE LATHE SIZE

Several features are used to find the size of an engine lathe. When considering purchase of a lathe are (1) the largest workpiece diameter that can be rotated without striking the ways and (2) the overall length of the bed. For some applications, the largest workpiece diameter that can be rotated without striking the top of the cross slide may be the most important consideration. The diameter of the hole through the hollow spindle will be the limiting feature for diameters of bars that can be passed through it. The electric motor, which furnishes power for the lathe, limits the rate of metal removal.

Operations commonly performed on a lathe include turning, facing, boring, use of form tools, parting, grooving, knurling, drilling, taper turning, taper boring, and thread chasing. When Milling machines are not available, an engine lathe with certain accessories such as a Milling Vise, or Tool Post Grinder may also perform some milling and grinding operations.

There will need to be a collar on each end of the head stock so the blank can be centered. The collars will need to be tapped for 4-1/4 inch set screws, which will be used to center the blank.

The lathe should be able to turn at least 2000 rpm. It should have tapered bearings in the head stock spindle.

You will have to get extra gears for your lathe so you can slow down the feed to give you the minimum of ·001·5 inch of feed per revolution.

OIL PAN

There should be some type of oil pan under the ways to catch the returning cutting oil, so it can be strained before it is returned to the oil reservoir, if you plan to use a coolant. This tray should extend full length of the lathe.

TOOL POST GRINDER

In my opinion the Tool Post Grinder is the most important piece of equipment or accessory that you will need for your lathe. With the Heat Treat Furnace, Milling Machine, lathe, and Tool Post Grinder you make all the tools that you need in the shop. If you are going to make your tools, such as reamers, and other special tools or cutters, a tool post grinder is necessary. With a tool post grinder you can cut your expenses down to a very small percent of what it would be if you had to buy them or have them special made.

MILLING MACHINE

You will need a milling machine with an indexing attachment for making rifling buttons, and reamers. A vertical mill would be the best choice, as you

can do much gun work with it. You will also need a coolant pump. This can be from an air conditioner pump, the evaporative type.

The coolant that you should is a water soluble type that can be found at any machine supply house or oil bulk plants.

A good small mill can be bought from Wholesale Tools. See listing at back of manual under suppliers.

DRILL PRESS

Most shops have these. You will need a drill press for most of your fixture making. There will be quite a few fixtures to be made to drill the special parts.

SHAPER

A shaper is not a necessary item to have but it will save quite a bit of time in making the necessary fixtures that will be needed.

Most of the work can be done on a shaper can be done on a milling machine. However some special shapes can best done with a shaper. It is easy to shape a lathe bit to what you want rather than to try to reshape a milling cutter.

SAWS

A good band or cut off saw is necessary when you are working with barrel steel. It gets old very quick cutting off a 1¼ bar steel with a hack saw. It will come in handy also in the fixtures that you will be making.

Wholesale Tool has a good one that works as a cut off saw or a vertical band saw:

HEAT TREAT FURNACE

This is absolutely necessary to have if you want to make your tools. There are many small furnaces available on the market that would work for what we want. It should go up to at least 2000 degrees, if you are planning working with high speed steel. In this book I give complete instructions for making gas furnaces for as little as one hundred dollars and a little time and effort.

I have found that an oil hardening tool steel (01) works just about as good. You will need to have good control to hold precise temperatures of the oven.

This can be used to draw the temper of the reamers and cutters also. The furnace can be made fairly easy, and a blower from a vacuum cleaner can provide the air. More on this later.

MEASURING AND LAYOUT TOOLS

The following listing includes all the tools and instruments of this category that are essential to good gunsmithing and tool making. Some of these precision items are a bit on the expensive side when one has to go out and buy them all at once.

Considering the years of good service they will render, if properly taken care of, one can scarcely consider them as being really costly.

MICROMETER

You will need a micrometer from 0 to one inch, and one to two inches. They should be of a type so you can read down to ten thousandths of an inch.

MICROMETER (DEPTH)

Most of these come equipped with three interchangeable rods giving a range of measurement from 0-3 inch by thousandths of an inch.

MICROMETER (INSIDE AND OUTSIDE)

These should have a capacity of at least 6″ and equipped to give a reading in thousandths.

ANGLE AND RADIUS GAUGE

Another of the gauges that you will need will be angle and radius gauges. These are not used to often, but they do come in handy when you need them. You will need a thread gauge, as in every barrel you pull you will have to know how many threads per inch there is.

LEVELS

You will need a very accurate machinist level, one that will have the adjustable degree base, so correct angles can be achieved.

TOOL STEEL

You will need a good supply of tool steel, (oil Hardening) for your reamers. You can experiment with different makes till you find what will fit your needs.

In over 30 years I have found 01 hard to beat. More on this later.

Reamers can if you want to made from worn out hand reamers. All that is necessary is to regrind them to the size needed. All sizes and dimensions will be given in later chapters, along with all other information and sketches.

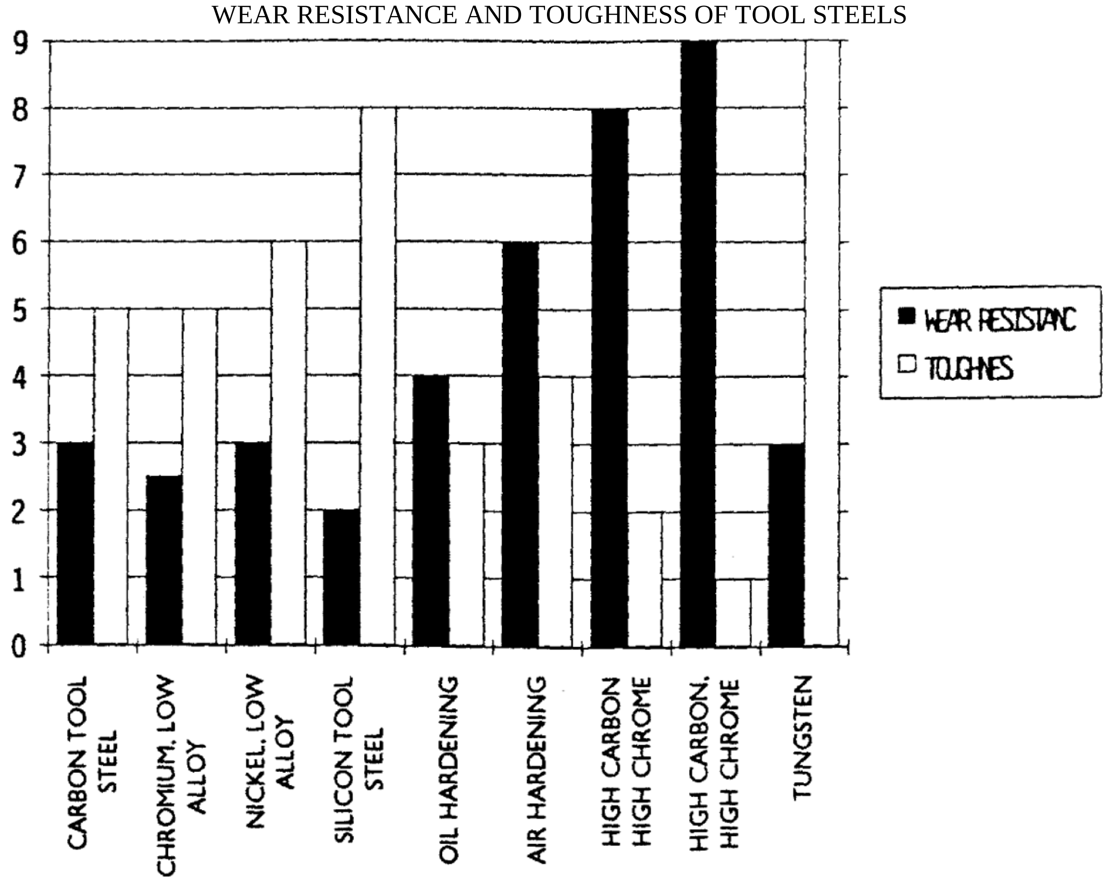
CHAPTER 2 HEAT TREATING TOOL STEEL

There is a lot of different tool steels available to the machinist today. I will list some of the more common types that you can use. A good source of tool steel if you live in a fanning community is old farming equipment. Much of the steel used in plows, springs, and other types of farming equipment is made of some type of tool steel.

When you use the plow items, they will be too hard to machine. To use these, you will need to cut them to the size that will fit in your furnace and anneal them. To do this all that is necessary is to bring them up to about 1500° and either let them cool in the oven, or bury them in lime. When cool they can be processed much as any other steel.

Much of items you make will not need to be heat treated, as they can be used as they are. If you need to heat treat other parts other than tools, treat them the same as a carbon steel. Drawing the parts back to a working hardness is important as they are glass hard after quenching.

CHANGES IN TOOL STEEL



Wear Resistant & Toughness Chart

Now for the heat-treatment of the tools and cutters that you will be making. To prevent the spoilage of tools during heat-treatment, you must understand the changes that take place in the steel when it is heated, quenched and tempered.

This chapter deals with the heating cycles for typical tool steels, the quenching or cooling procedure, and the effects of single tempering operations on the structure and properties of the steel.

LIFE OF TOOLS

The life of tools if proportional to their hardness can be developed in a plain carbon steel by heat-treatment if the steel contains over about 0·50% carbon, provided the section is not very large.

When tools are made in large sections, a plain carbon steel cannot be hardened adequately, and it is necessary to add allowing elements in order to increase harden-ability.

The elements usually used for this purpose are chromium, manganese, and molybdenum.

Steel becomes hard during heat treatment because of the formation of a microstructure called martensite. For some tool applications, the wear resistance provided by the martensite structure alone is not sufficient, and therefore, tungsten and vanadium as well as chromium and molybdenum, are introduced into tool steel. This is important to take into consideration when purchasing steel.

These elements combine with some of the carbon in the steel to form very hard particles of carbides. This gives the heat treated steel much better abrasion resistance than can be developed without the presence of alloy carbides.

When the tool operates at high speed or under high pressure or is in contact with hot metal, as in forging, etc., special tool steels must be used that are resistant to the high temperatures encountered.

Vanadium, cobalt, and chromium combined with tungsten or molybdenum give the steel the necessary resistance to softening at high temperatures.

Finally, in some applications, the life of the tool is more dependent on toughness than on hardness. If you are using farm equipment, most of the steel will fall into this group.

TYPICAL TOOL STEELS

The compositions of tool steels that we use for tool making are as follows.

GENERAL PROPERTIES AND HEAT TREATING DATA FOR CARBON TOOL STEELS						
Type	AISI	Carbon	Mn	Si	Cr	V
110	W1	0·60-1·400	0·25	0·25		
120	W2	0·60-1·40	0·25	0·25		0·25
121	W3	1·00	0·25	0·25		0·50
122		0·90	0·40	0·55		0·10
CARBON-CHROMIUM TOOL STEELS						
130		1·00	0·25	0·25	0·10	
131	W4	1·00	0·25	0·25	0·25	
132	W5	1·00	0·25	0·25	0·50	
133	W4	0·90	0·70	0·25	0·25	
140	W7	1·00	0·25	0·25	0·35	0·20

Heat Treating Data For Carbon Steels

PLAIN CARBON TOOL STEELS

Carbon 1·14, Manganese 0·22, Silicon 0·16 per cent.

OIL HARDENING TOOL STEEL

Carbon 0·85, Manganese 1·18, Silicon 0·26, Chromium 0·50, Tungsten 0·44 per cent.

HIGH CARBON HIGH CHROMIUM TOOL STEEL

Carbon 1·55, Manganese 0·27, Silicon 0·45, Chromium 11·34, Vanadium 0·24, Molybdenum 0·53 per cent.

GENERAL PROPERTIES OF W SERIES						
TYPE	AOSI	WEAR RESISTANCE	TOUGHNESS	USUAL WORKING HARDNESS Rockwell C	SURFACE HARDNESS AS-QUENC Rockwell C	CORE HARDNESS Rockwell C
110	W1	2-4	3-7	58-65	65-67	38-43
120	W2	2-4	3-7	58-65	65-67	38-43
121	W3	4	3-7	58-65	65-67	38-43
122		3	3-7	58-65	65-67	38-43
130		3-4	3-7	58-65	65-67	38-43
131	W4	3-4	3-7	58-65	65-67	38-43
132	W5	3-4	3-7	58-65	65-67	38-43
133	W4	3	3-7	58-65	65-67	38-43
140	W7	3	3-7	58-65	65-67	38-43

HEAT TREATING DATA FOR CARBON TOOL STEELS									
TYPE	AISI	AVAILABILITY	QUENCHING MEDIUM	HARDENING TEMPERATURE, F°	APPROX HARDNESS AS ROLLED OR FORGED BHN	ANNEALED HARDNESS, Bhn	ANNEALING TEMPERATURE F°	TEMPERING RANGE, F°	FORGING TEMPERATURE, F°
110	W1	4	W	1400 TO 1550	275	159 TO 202	1360 TO 1450	300 TO 650	1800 TO 1950
120	W2	4	W	1400 TO 1550	275	159 TO 202	1360 TO 1450	300 TO 650	1800 TO 1950
121	W3	1	W	1400 TO 1550	275	163 TO 202	1360 TO 1450	300 TO 650	1800 TO 1950
122		1	W	1400 TO 1550	275	163 TO 202	1360 TO 1450	300 TO 650	1800 TO 1950
130		1	W	1400 TO 1555	275	159 TO 202	1360 TO 1450	300 TO 650	1800 TO 1950
131	W4	1	W	1400 TO 1500	275	159 TO 202	1360 TO 1450	300 TO 650	1800 TO 1950
132	W5	2	W	1400 TO 1550	275	163 TO 202	1360 TO 1450	300 TO 65c	1800 TO 1950
133	W4	1	W	1400 TO 1550	275	159 TO 202	1360 TO 1450	300 TO 650	1800 TO 1950
140	W7	1	W	1400 TO 1550	275	163 TO 202	1360 TO 1450	300 TO 650	1800 TO 1950

Carbon Steel Data

Steels referred to as Oil Hardening, and differs from the first type primarily in harden ability. This is the basic type of tool steel that I have used for over 30 years. Because of its greater harden-ability, large tools made of this steel can be hardened by an oil quench rather than a water quench.

Tools made of the oil hardening tool steels do undergo as much size change and distortion during hardening as those made from plain carbon steels.

CARBON STEELS

Plain carbon steels are among the least expensive of tool steels, and their properties provide a convenient base line against which all other tool steels may be compared, Since the wear resistance of plain carbon tool steel is lower than that of alloy steels of equivalent carbon content, the useful life of a plain carbon tool will normally be shorter than that of a tool made from a higher-alloy steel. One notable exception is found in cold header dies where, owing to their combination of hard case and tough core, plain carbon tool steels are unsurpassed in performance. Other applications for plain carbon tool steels include shear blades, blanking dies, reamers, threading dies, taps, twist drills, lathe tools and wiredrawing dies.

A portion of the iron-iron carbide phase diagram is a useful guide to understanding of the heat treatment and properties of plain carbon tool steels. Although the alloying elements, other than carbon, have a significant effect on the position of the phase boundaries, the iron-iron carbide diagram is sufficiently precise to warrant its use in making qualitative conclusions.

HOW HEATING CHANGES THE STRUCTURE

The first step in the heat treatment of steel is the heating. The purpose of the heating is to form austenite and to dissolve carbon in the austenite. The solution of the carbon is necessary so in the second step of heat treatment, when the transformation of austenite takes place, the steel will develop the needed hardness.

You must remember that even though a large amount of carbon is in the steel, it is not effective in developing hardness unless it is first dissolved in the austenite. The hardness increases rapidly up to 60 Rc as the carbon increases to 0·40 to 0·70%. Above about 0·70% carbon, the hardness remains practically constant.

LOW ALLOY TOOL STEELS							
TYPE	AISI	C	MN	SI	CR	V	MO
210	LI	1·10	0·30	0·25	1·25		
211	L2, L3	0·65 TO 1·10	0·10 TO 0·90	0·25	0·70 TO 1·70	0·20	
212	L7	1·00	0·35	0·25	1·40		0·40
213		1·00	1·60	1·60	2·75		

Low Alloy Tool Steels

For maximum hardness in the steel, therefore, approximately 0·70% of carbon must be dissolved in the austenite. All of the steels being discussed except the chromium molybdenum hot work steel have sufficient carbon in the analysis to attain a hardness of 65 Rc.

GRAIN SIZE AND TOUGHNESS

Two other factors are involved in the heating of tool steels, besides the formation of austenite and the solution of carbides.

These are grain coarsening and melting of the steel. In tool steels, the grain size should be as small as possible, because a fine grained hardened steel is inherently tougher than a coarse grained steel.

LOW ALLOYS PROPERTIES						
TYPE	AISI	WEAR RESISTANCE	TOUGHNESS	USUAL WORKING HARDNESS ROCKWELL C	SURFACE HARDNESS AS QUENCHED, ROCKWELL C	CORE HARDNESS 1 INCH, ROCKWELL C
210	LI	3	4	56 TO 63	58 TO 67	54 TO 60
211	L2, L3	2 TO 3	4 TO 6	45 TO 63	58 TO 67	54 TO 60
212	L7	3 TO 4	4	56 TO 64	58 TO 67	54 TO 60
213		4	4	56 TO 64	58 TO 67	58 TO 64

Low Alloys Properties

Usually there is little concern about coarse grain in a tool steel because coarsening of the steel does not occur until the temperature is well above the usual austenitizing temperatures.

WATER HARDENING

AISI-SAE W2-1·00 Carbon-Vanadium. This an all purpose water hardening Carbon-Vanadium Tool Steel with remarkable hardening characteristics that permit a wide variation in treatment with uniformly good results. Used for punches, blanking and forming dies, shear blades, bending dies, etc.

LOW ALLOY HEAT TREAT DATA									
AVAILABILITY	MACHIN ABILITY	QUENCHING MEDIUM	HARDENING TEMPERATURE, F°	SUSCEPTIBILITY TO DECARBURIZATION	APPROX HARDNESS AS ROLLED OR FORGED Bhn	ANNEALED HARDNESS Bhn	ANNEALING TEMPERATURE F°	TEMPERING RANGE, F°	FORGING TEMPERATURE F°
1	8	W, O	1450 TO 1550	M	350	179 TO 207	1425 TO 1475	350 TO 600	1800 TO 2000
2	8	W, O	1400 TO 1500	M	350	163 TO 202	1400 TO 1500	350 TO 1000	1800 TO 2000
1	8	O	1500 TO 1600	M	400	183 TO 212	1450 TO 1500	300 TO 600	1800 TO 2000
1	8	O	1500 TO 1600	M	450	192 T0 228	1475 TO 1575	300 TO 600	1800 TO 2000

Low Alloy Heat Treat Data

Effects of tempering on a 2 inch disc 5/8 inch thick, water quenched at 1450 degrees.

TEMPERING TEMPERATURE

Rockwell C

As Quenched	100°	200°	300°	400°	500°	600°
67	67	67	65	62	59	55

Hardening - AISI SAE 01

This is one of the main tool steels that I used. This is an oil hardening tool steel that combines high hardness and deep hardness with minimum distortion, freedom from cracking and good machine-ability. Used for cams tolls, thread rolling dies, thread chasers, blanking dies and punches, bushings, etc.

WORKING TEMPERATURE

Forging, 1800 to 1925 degrees F., never below 1550 degrees F.

OIL-HARDENING COLD WORK TOOL STEELS								
TYPE	AISI	C	MN	SI	CR	V	W	MO
410	01	0·95	1·20	0·25	0·50	0·20	0·50	
411	02	0·95	1·60	0·25	0·20	0·15		030
412		1·00	1·10	0·60	1·00			
413	07	1·20	0·25	0·25	0·60	0·20	1·60	025
414		1·05	0·70	0·25	1·60		0·50	

Oil Hardening Tool Steel

Hardening, 1400 to 1475 degrees F. Annealing, 1375 to 1425 degrees F.

Tempering Temperature

As Quenched	300°	400°	500°	600°	700°
63	62·5	60	57	53·5	51·5

HEAT-TREATMENT OF TOOL STEEL

Hardening

The rate of heating for hardening should be slower for alloy steels than for plain carbon steels. The higher the alloy content, the slower the heating rate should be. Much difficulty with warping or size change can be reduced or eliminated by slow uniform heating.

HEAT-TREATING PROPERTIES FOR OIL HARDENING						
TYPE	AISI	WEAR RESISTANCE	TOUGHNESS	USUAL WORKING HARDNESS, ROCKWEL C	SURFACE HARDNESS AS QUENCHED ROCKWEL C	CORE HARDNESS 1-INCH ROCKWEL C
410	01	4	3	57 TO 62	61 TO 64	59 TO 61
411	02	4	3	57 TO 62	61 TO 64	59 TO 61
412		3	3	57 TO 62	61 TO 64	59 TO 61
413	07	5	3	58 TO 64	61 TO 64	59 TO 61
414		4	3	58 TO 62	62 TO 64	62 TO 64
Oil Hardening Heat Treat Data						

If you have a gas furnace start the heat low, and bring up the temperature in steps. This takes about 30 minutes. Always put the tools in the oven before it is started. This preheating is not always necessary for the water-hardening or oil hardening groups, but I have found that this almost always eliminates warping. I have found also that this preheating will reduce the time in a hardening furnace without atmosphere control and reduce scaling and decarburization.

Commercial compounds can be bought for temperatures up to 1550 degrees, to eliminate decarbonizing of the surface. Brass brazing flux can be used also, and it works up to 2200 degrees.

OIL HARDENING TOOL STEELS									
AVAILABILITY	MACHLNABILITY	QUENCHING MEDIUM	HARDENING TEMPERATURE F°	SUSCEPTIBILITY TO DECAR-BURIZTION	APPROX HARDNESS AS ROLLED OR FORGED, BHN	ANNEALED HARDNESS, BHN	ANNEALING TEMPPERATURE, F°	TEMPERING RANGE, F°	FORGING TEMPERATURE, F°
4	8	O	1450 TO 1500	M	325	183 TO 212	1400 TO 1450	300 TO 500	1800 TO 1950
3	8	O	1400 TO 1475	M	325	183 TO 212	1375 TO 1425	300 TO 500	1800 TO 1925
2	8	O	1450 TO 1600	M	350	183 TO 212	1425 TO 1475	300 TO 500	1800 TO 1950
2	7	W. O	1450 TO 1625	M	325	192 TO 217	1450 TO 1500	300 TO 500	1800 TO 2000
1	6	O	1500 TO 1550	M	325	179 TO 207	1500 TO 1550	300 TO 500	1800 TO 1900

Oil Hardening Tool Steel

The steel must be held at a temperature long enough to ensure uniform temperature throughout the entire piece. Longer time at temperatures is required for the high alloy steels. When heating 01 steel it is best to hold a slightly rich flame on the gas furnace to keep from getting a soft skin on the surface after quenching.

When the steel has reached the proper temperature, let it soak at this temperature for a few minutes. You will need a long nose tong tool to be able to reach into the oven to remove the tool. When ready to remove the part, open the oven and grasp the end of the tool with the tongs. Now carefully and quickly remove the part from the oven and plunge it straight into the quenching medium that is correct for the steel you are using.

Go straight in when you quench, as if you quench the tool on the angle it will more than likely warp and make the tool worthless. Hold the tool in the quenching medium for 10 to 20 seconds without moving, then move it up and down to finish cooling it.

QUENCHING MEDIA

I have used quite a variety of liquids to quench the tools in over the years, and when you are ready to heat-treat the tool make sure that you use the right quenching medium.

WATER

Water as it comes from the faucet, but have not got to good results, as there seems to be too much air in the water. If you use water, keep the temperature at 60 to 80 degrees.

BRINE

This will produce a more uniform quench if not over 10% salt by weight is added to the water. I have found that it will produce a much cleaner and more uniform surface on the tools. As with tap water, temperatures should be held at 60 to 80 degrees. After quenching, dip in oil to eliminate rusting.

OIL

I normally use a 5 weight oil for all my quenching, but any mineral oil 5 weight or less will work fine. The temperature of the oil should be held 100 to 130 degrees F.

LYE

Carbon Steels may be quenched in a 5% lye solution with very good results. I have found that it will eliminate soft spots in the steel. It will give a bright finish, plus it will not rust the tools if they are not dipped in oil.

TEMPERING

Now comes the most important part of heat treating process and that is of tempering of the metal. Harden steel can be tempered or made softer and less brittle by reheating it to a pre-determined temperature. Depending upon the nature of the steel and its intended use, and then the cooling.

When the tempering is done by the color method, the temper is gauged by the colors formed on the surface of the polished metal as the heat increases. The color method of gauging temperatures is not dependable, as the color is affected, to some extent, by the composition of the metal. Tempilaq is a compound that you can buy, (Brownells caries all the temperature ranges) in any temperature range, to put on the metal.

When it melts you have the metal at the proper temperature. It is better to have a temperature gauge to get the correct temperature. After you have worked with a certain type of steel for a while, you will be able to judge the correct temperature by the color. It is best to polish the metal to a bright finish so the color can be seen clearly.

COLORS FOR TEMPERING

430 degrees, Very pale yellow, extra file hard, dies, milling cutters, cut off tools.
440 degrees, Light Yellow, file hard, reamers, thread chasers, fly cutters, hollow mills.
450 degrees, Pale straw yellow, profile cutters for milling machines, rolling dies, knurling tools.
460 degrees, Straw yellow, knife hard, swages.
470 degrees, Deep straw yellow.
480 degrees, Dark yellow, cutting dies.
490 degrees, Yellow brown, extra hard, taps, dies.
500 degrees, Brown yellow, thread dies for general work.
510 degrees, Spotted red brown.
520 degrees, Brown purple, hard.
530 degrees, Light purple.
540 degrees, Light purple.
550 degrees, Dark purple.
560 degrees, Full purple.
570 degrees, Dark blue, half hard.
620 degrees, Blue gray, spring temper.

HIGH TEMPERATURES BY COLOR

752 degrees, Red heat, visible in the dark.
885 degrees, Red heat, visible in the twilight.
975 degrees, Red heat, visible in the daylight.
1077 degrees, Red heat, visible in the sunlight
1292 degrees, Dark red.
1472 degrees, Dull cherry red.
1652 degrees, Cherry red.
1832 degrees, Bright cherry red.
2015 degrees, Orange red.
2192 degrees, Orange Yellow.
2372 degrees, Yellow white. .
2552 degrees, White welding heat.
2732 degrees, Brilliant white.
2912 degrees, Dazzling white, bluish white.

In tempering I let the oven cool down to the temperature that I need to draw the temper for the job that I need. I then put the part in the oven, then let it cool down over night.

FORGING

Plain carbon steels are among the easiest tool steels to forge. When heated to recommended forging temperatures (1800° to 1950° F), the structure is free of excess carbides that may decrease forgeability. No special precautions are required, although prolonged soaking at forging temperature should be avoided to minimize surface scaling and decarburization. The best forging temperature for a given steel depends on the carbon content; the lower the carbon content, the higher the forging temperature. For example, a 0·70% Carbon steel may be forged at 1925° F, while 1825° F is best for a 1·20% Carbon type. The recommended finishing temperature depends on carbon content in a similar manner, decreasing from 1500° F at the 0·70% Carbon level to 1450° F at the 1·20% level. After forging, pieces may be air cooled.

Although not absolutely necessary, forgings may be normalized from a temperature slightly above the recommended temperature before annealing to obtain a more uniform structure. Normalizing produces a "soft" structure predominantly pearlitic but with some tendency for cementite at prior austenitic grain boundaries if the carbon is over about 1·10%.

ANNEALING

Sometimes you may need to anneal the tool to do some machining or changing. The annealing of steel consists in heating it slightly above the critical temperature range and cooling it slowly in the oven. The steel is then totally annealed. The steel is held at this temperature usually not less than one hour for each inch of the thickest section of the part being annealed.

Annealing is necessary after forging and normalizing, or after cold working, to ensure grain refinement and to place the structure of the steel in the proper state for machining and heat treating. The advantages of this treatment may be summarized as follows:

1. Softens the steel for ease in machining
2. Changes the structure, thereby altering the machining characteristics
3. Relieves stresses induced by mechanical working
4. Produces a structure that will respond uniformly to hardening.

To minimize scaling and decarburization, the material to be annealed should be packed in an inert substance such as dry sand and heated to the annealing ~ range. Packing ensures a slow cooling rate from the annealing temperature, but packing materials such as charcoal or cast iron chips should be avoided since they may cause slight decarburization. Time at temperature during the anneal should be controlled according to section thickness of the steel; up to a 1 inch section size, 15 minutes at temperature is adequate, while an 8 inch section would require about 2½ hour.

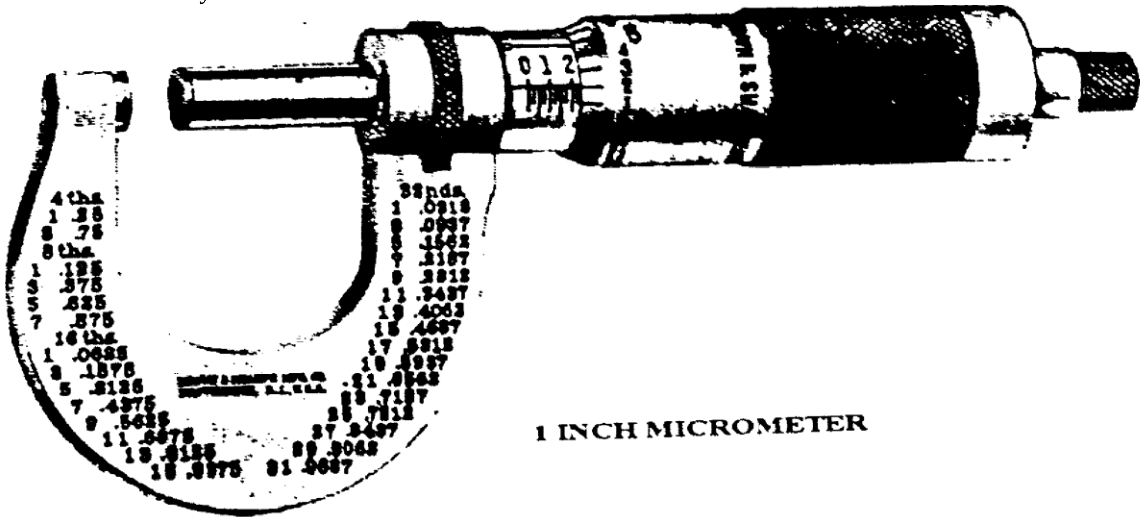
Cooling from the annealing temperature should be slow (maximum rate of 50° F per hour) down to 1000° F, at which point more rapid cooling is permissible. By annealing after machining this will also remove any stress or hard spots caused by machining. The temperature required for annealing varies for different steels. Low carbon steel may be annealed at about 1650 degrees F. The temperature should be maintained long enough to heat the entire piece evenly throughout.

Care should be taken not to heat the steel much above the decalescence or hardening point. When steel is heated above this temperature, the grain assumes a definite size for that particular temperature, the coarseness increasing with an increase of temperature.

Moreover, if the steel that has been heated above the critical point is cooled slowly, the coarseness of the grain corresponds to the coarseness at the maximum temperature. Therefore the grain of annealed steel is coarser, the higher the temperature to which it is heated above the critical point.

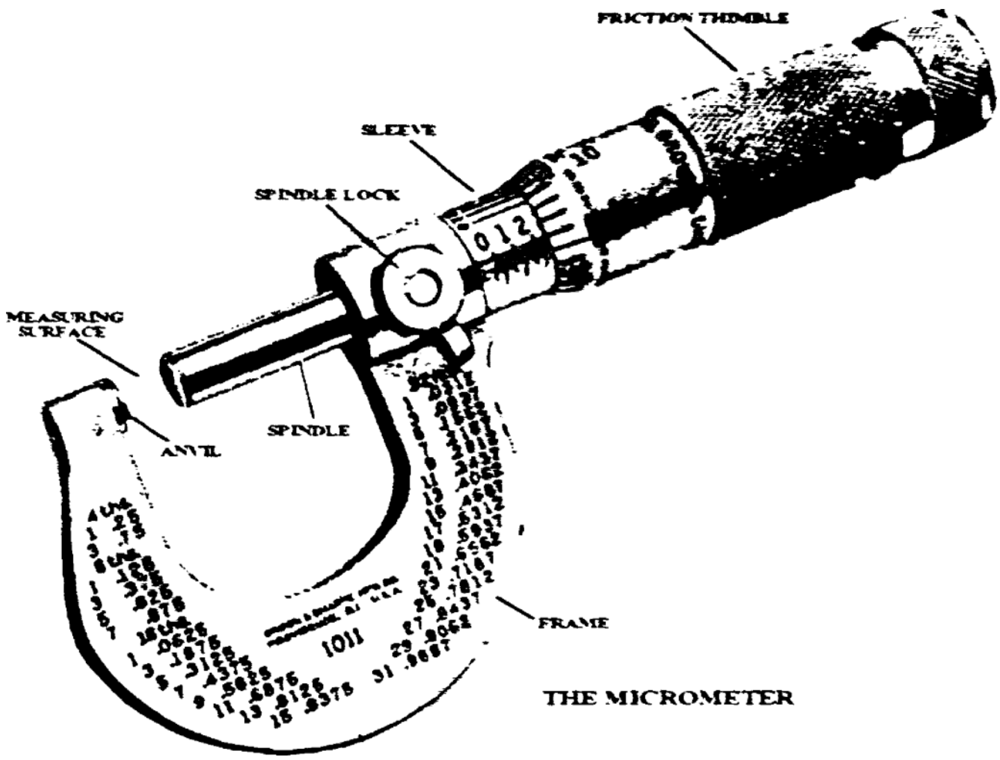
CHAPTER 3 MICROMETER CALIPER

The first thing that you should know is how to read measuring tools such as micrometers and calipers. These are the handiest to use and will give you accurate measurements of your work



STANDARD 1 INCH MICROMETERS

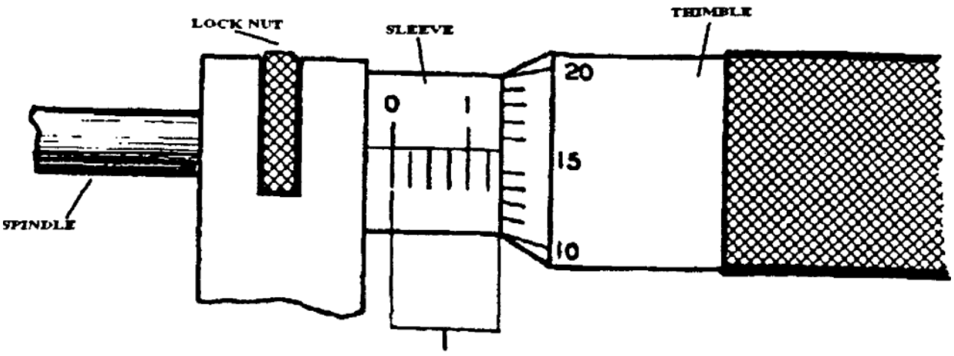
There are many types, and I prefer the one with the dial indicators on it. You will need two sizes of calipers should be provided. One set should be about 4 inch, and the other set 8 inch. They should be of the spring-opening type with screw adjustment. The quick-opening spring nut is a good time saver, as it is possible to open the nut and make a rough adjustment of the calipers without having to run the nut slowly along the thread.



PARTS OF THE MICROMETER

MICROMETER DEPTH GAGE

A micrometer depth gage is used to measure the depth of such work as holes, slots, recesses, and keyways. The tool consists of a hardened, ground and lapped base combined with a micrometer head. Measuring rods with individual length adjustment are inserted through a hole in the micrometer screw and brought to a positive seat by a knurled nut. The screw has a 1″ movement.



0·125 - 0·0150-
0·140 THOUSANDS TOTAL

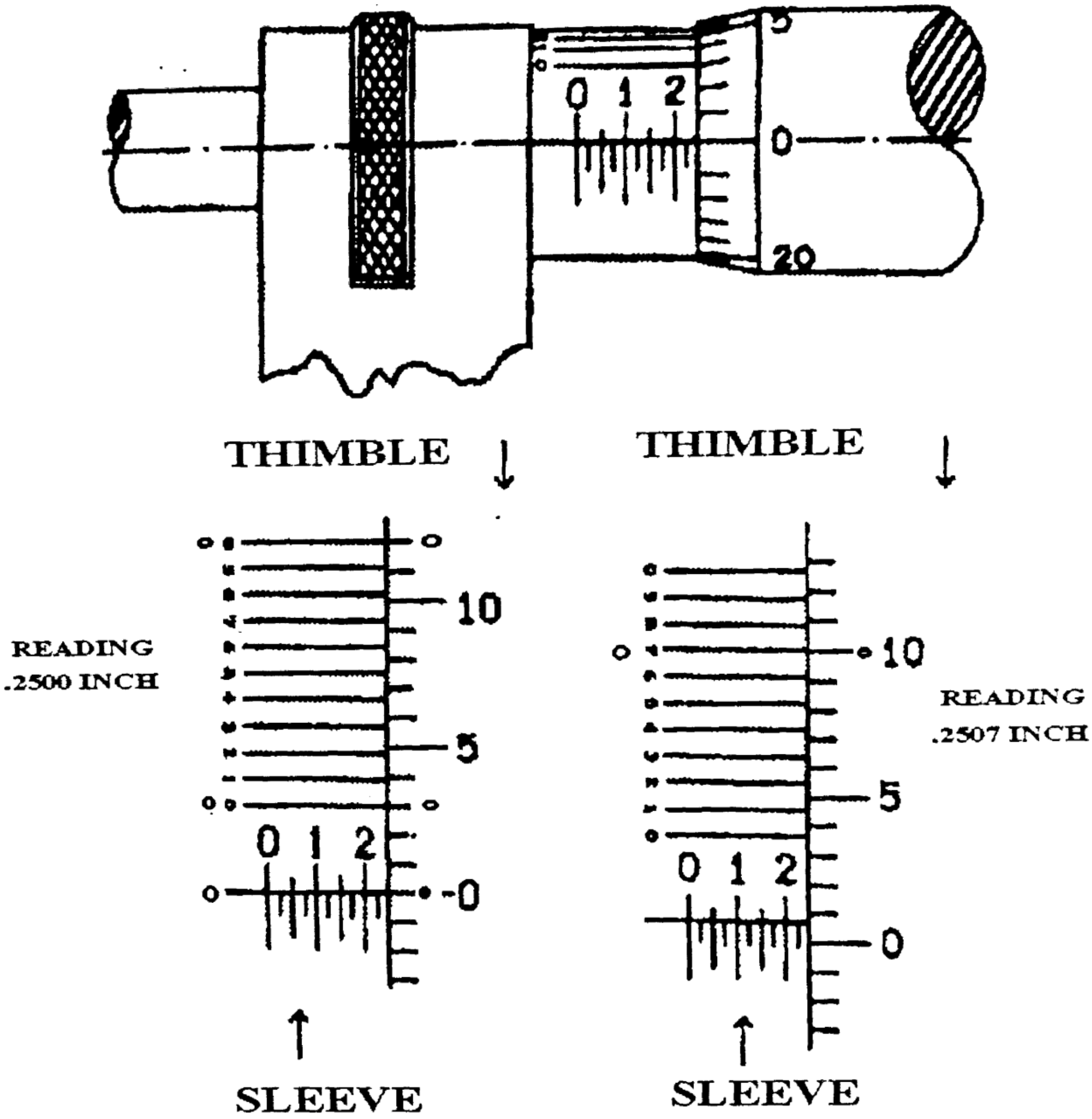
HOW TO READ A MICROMETER

For accurate work you will need a good micrometer. These are rather expensive, but a 1 inch size and a 2 inch size will be used often and are well worth the price. To go with these you should have a set of inside micrometers. These come with adjusting rods to measure from 2 inches up.

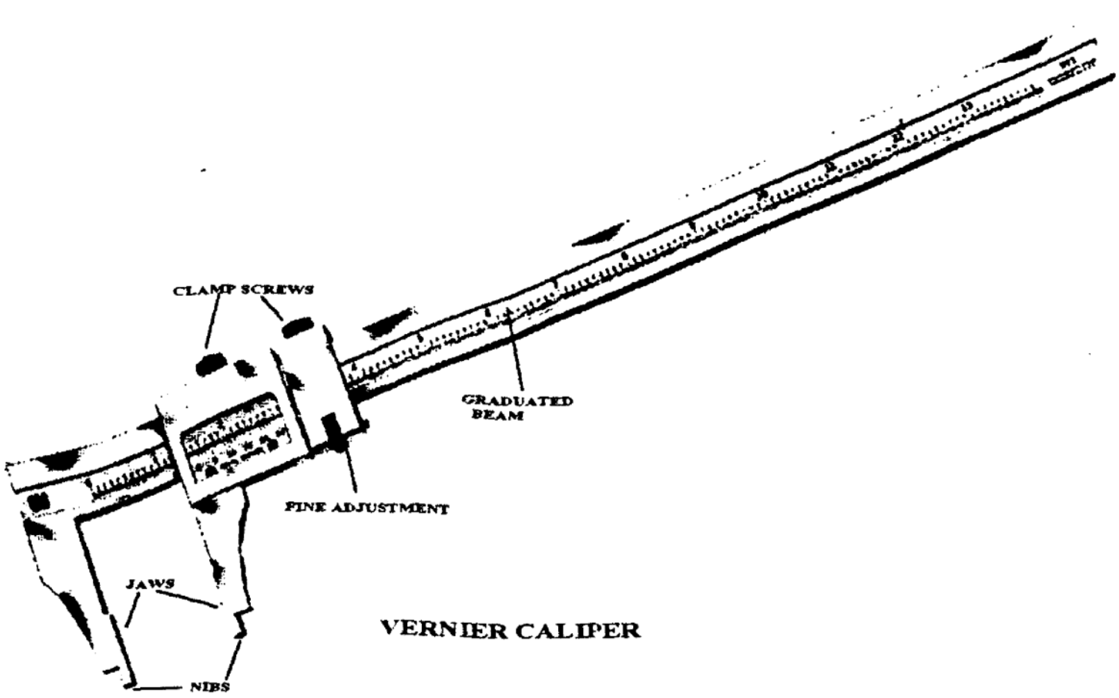
For accurately measuring holes smaller than 2 inches across, a telescoping gauge should be used. This has a handle with a telescoping head operated by a spring. Several lengths of pins fit the sliding part of the head, and there is a locking nut at the end of the handle.

For measuring the depth of holes, a depth gauge is necessary. For very accurate work, a micrometer depth gauge should be used. The usual run of work in a home workshop will not require the use of a micrometer depth gauge, so one with a sliding head is good enough.

The smallest measurement in common fractions that can be made with the fixed caliper and steel rule is 1/64″ To measure in thousandths and ten- thousandths, you will need a micrometer.



READING A MICROMETER GRADUATED IN TEN THOUSANDTHS OF AN INCH



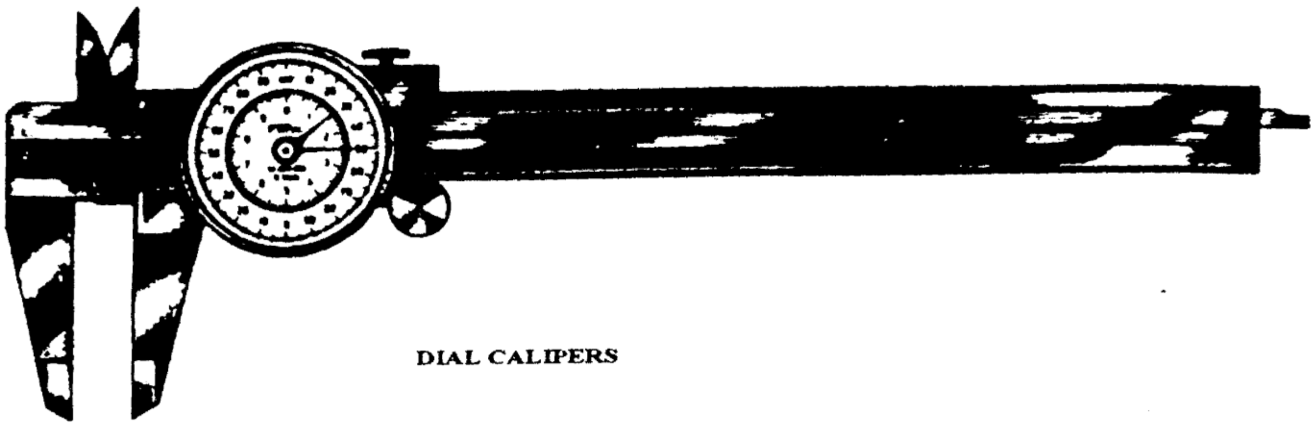
VERNIER CALIPER

READING THE MICROMETERS

To use these you must understand that the pitch of the screw thread on the spindle of a micrometer is 1/40″ Or 40 threads per inch. So one complete revolution of the thimble advances the spindle face toward or away from the anvil face 1/40″ or ·025″.

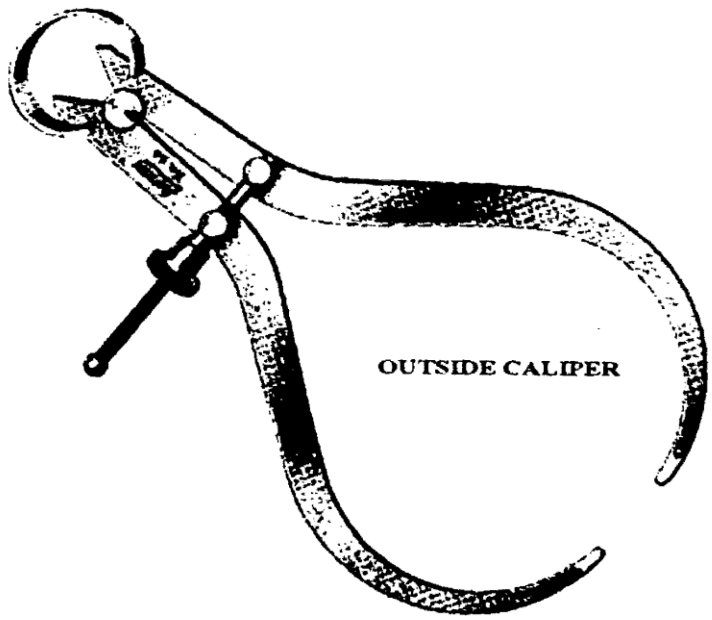
On the longitudinal line on the sleeve it is divided into 40 equal parts by vertical lines that correspond to the number of threads cut on the spindle. On the spindle each vertical line designates 1/40″ or ·025″ and every fourth line designates ·100″. You will see that the line marked "1" will represents ·100″, the line marked "2" will represents ·200″ and so on.

The beveled edge of the thimble is divided into 25 equal parts with each line representing ·001″ and every line numbered consecutively. By rotating the thimble to the next higher line moves the spindle 1/25 of ·025″ or ·001″. Twenty-five divisions shows a complete revolution, ·025″ or 1/40″.



What would be the answer if the reading if the edge of the thimble is between the ·125″ and the ·150″ lines, and the line on the thimble is the coinciding line? The answer is: Micrometer reading = sleeve + thimble or ·125 + ·015 = ·140 of an inch.

It is very important to keep your micrometers clean as dirt between the anvil and spindle will cause the micrometer to read incorrectly. If you want to test to see if the micrometer is accurate, clean and bring the anvil and spindle together carefully. If the zero line on the thimble and the axial (longitudinal) line on the sleeve fail to coincide, wear has taken place either in the screw or contact surfaces.



SPRING CALIPERS

Micrometers are made in a wide range of sizes and in matched sets. The ratchet on the micrometer is used to rotate the spindle when taking a measurement and insures consistent, accurate gauging by limiting the spindle pressure on the workpiece.

There is a locknut that makes it possible to lock the micrometer spindle at any desired setting. A slight turn of the knurled locknut ring contracts a split bushing around the spindle and makes the micrometer a fixed gage.

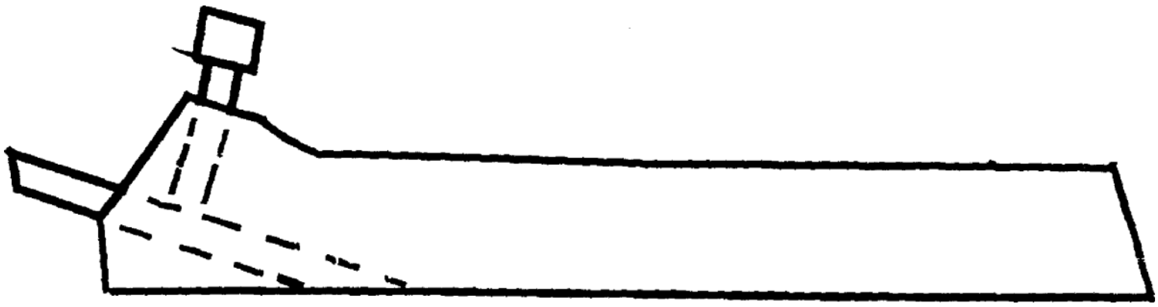
If you want very accurate measurements are required, a micrometer that has an extra scale added to the sleeve is used, enabling the micrometer to be read in ten-thousandths of an inch. This scale consists of a series of lines on the

sleeve parallel to its axis.

Ten divisions on the sleeve mark the same spaces as nine divisions on the beveled edge of the thimble. The difference 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, 1/10 of a division would be ·0001 or one ten-thousandth.

CHAPTER 4 CUTTING TOOLS AND TOOLHOLDERS

Because cutting-tool materials are expensive, it is desirable to use as small amounts as possible. It is necessary that the cutting tool should be supported in a strong, rigid manner to minimize deflection and possible vibration. Lathe tools are supported in various types of heavy, forged toolholders. The tool bit should be clamped in the tool post with minimum overhang. If there is much overhang, tool chatter and poor surface finish may result. Most lathe work is done with high-speed steel, carbide, or ceramic tools.



CUTTING TOOL HOLDER

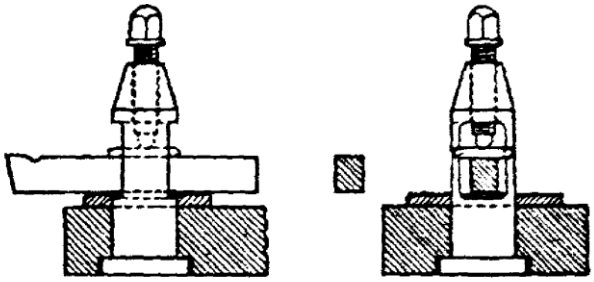
Where large tool bits are required, the heavy type of forged toolholder is used. It provides adequate method of clamping and supporting the tool than is provided by an ordinary tool post. The tools used in such cases have a heavy shank of Tool Steel or hot-rolled bar stock in which a carbide tip is brazed.

Most all of the lathe operations are done with simple, single point cutting tools. On right-hand and left-hand turning and facing tools, the cutting takes place on the side of the tool so that the side rake angle is important so deep cuts can be made. On the round-nose turning tools, cutoff tools, finishing tools, and some threading tools, cutting takes place on or near the end of the tool, so that the back rake is of importance, and are fed with light depths of cut.

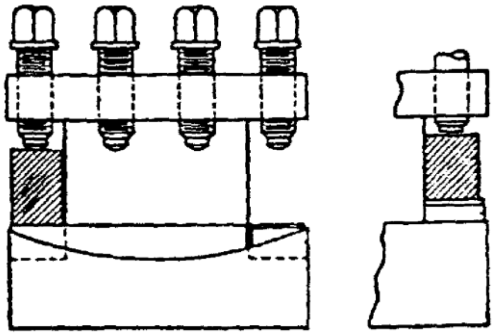
Where there are more than 1 operation on a lathe are performed repeatedly in sequence, the time required for changing and setting tools may make up as much as 50 per cent of the total time. Quick- change toolholders are being used to reduce the time of tool changing.

The individual tools, preset in their holders, can be interchanged in the special tool post in a few seconds. With some systems a second tool may be set in the tool post while a cut is being made with the first tool, and then be brought into proper position by rotating the post.

In most lathe work, the nose of the tool should be set exactly at the same height as the axis of rotation of the work. Any setting below the axis causes the work to tend to climb up on the tool, and in most operations you will need to set your tool a few thousandths of an inch above the axis, except for cutoff, threading, and some facing operations.



SINGLE SCREW TOOL POST

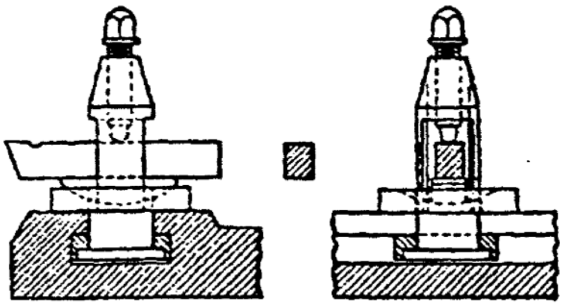


FOUR WAY TOOL POST

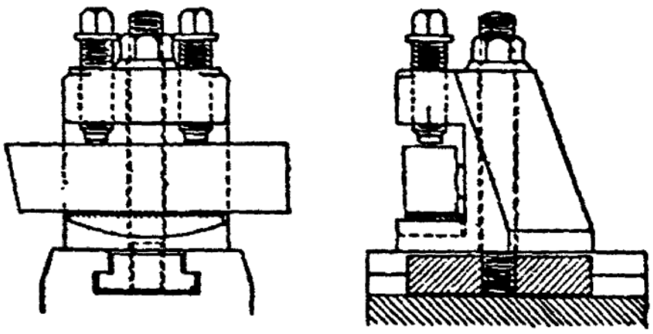
TOOL HOLDERS

SETTING A SINGLEPOINT CUTTING TOOL.

Single point cutting tools for the lathe are ground sharpened for the on center settings. Tilting the cutting tool to some angle other than the angle for which it was ground will affect the cutting action. A Single point cutting tool should be set in the position for which it is ground. For a heavy cut, the cutting edge of a cutting tool should be arranged so that it will deflect under heavy forces away from the workpiece surface instead of into the work piece surface.
The cutting force may become great enough to deflect the cutting tool about the vertical axis of the tool post.
For light cuts, the cutting edge may be set ahead of the tool post in the direction of feeding.



SINGLE POINT TOOL & TOOLPOST



OPEN SIDE TOOL POST

TOOL HOLDER TYPES

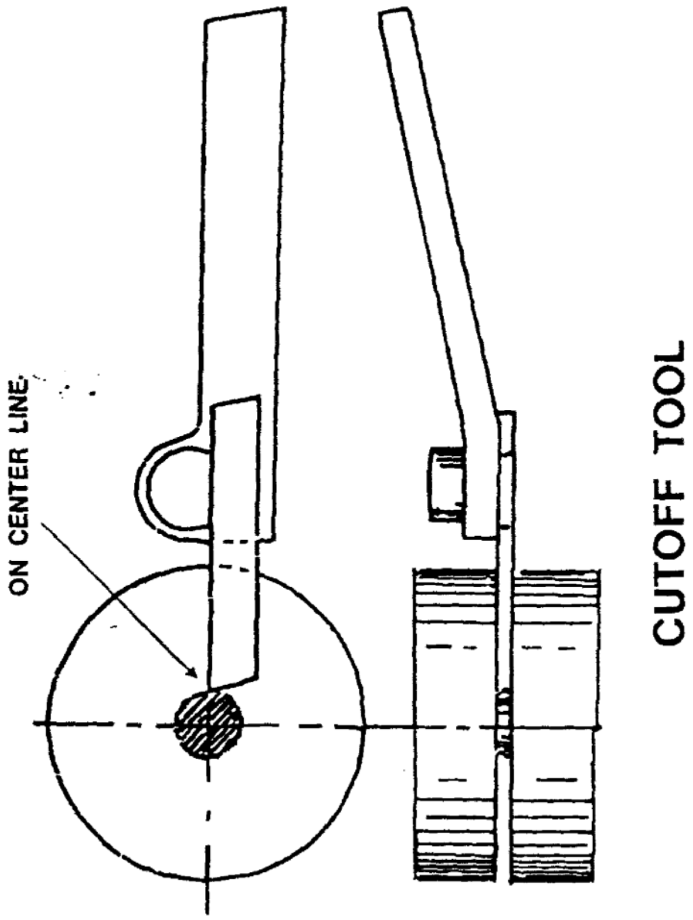
SHARPENING THE BIT

You will need some type of bench grinder to sharpen the tool bits when they get dull. In most cases the bit can be sharpened without a gauge, after you get some experience. When sharpening the bits, be sure that you have enough clearance so the tool doesn't rub on the work.

This can be checked with the tool in the holder and the point on the center of the workpiece, or it can be set on the center that is in the tail stock.

The type of fixture for holding and controlling the movement of the cutting tool include the tailstock, carriage, cross slide, and compound rest. The carriage, like the tailstock, is movable on the ways of the bed. The motion is parallel to the rotational axis of the spindle, and this motion is commonly used for producing cylindrical surfaces.

A compound rest is used for producing some tapered surfaces, for helping in the cutting of threads by thread chasing, and for several other types of applications. Mounted above the compound rest, the tool post serves to clamp the cutting tool, or tool holder, in a desired position. At the base of the tool post the rocker plate has a concave spherical surface facing upward.



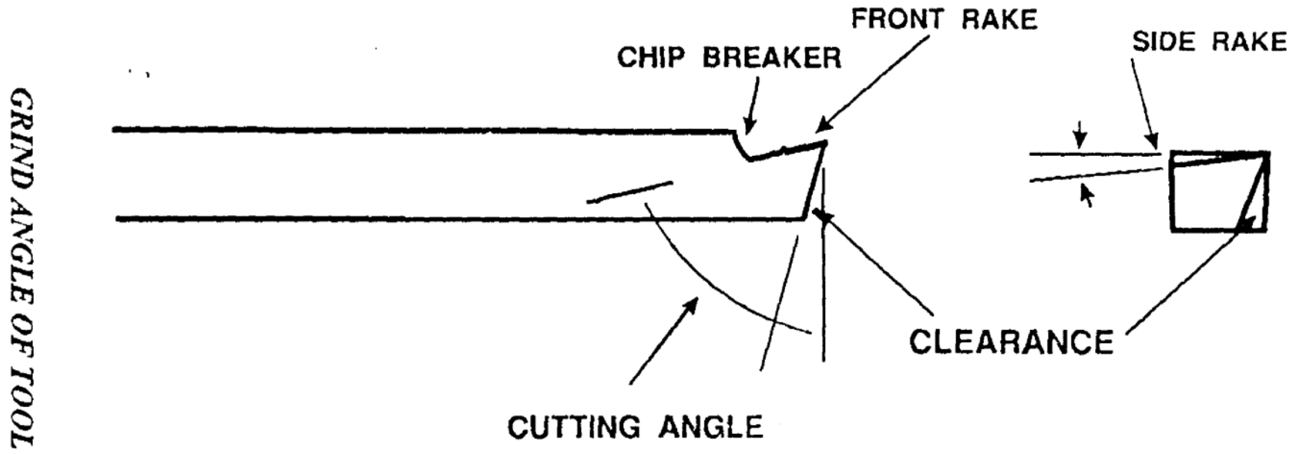
CUT OFF TOOL

Usually the cutting tool consists of a toolholder, which carries a tool bit. Different toolholders are available to hold tool bits at various desired horizontal angles. A rocker for mounting the tool holder that has a mating convex spherical surface fits above the rocker plate, and the cutting tool is clamped above this rocker. The rocker plate and rocker thus permit the cutting edge of the cutting tool to be raised or lowered. When the cutting tool has been properly positioned, a set screw is tightened to clamp the cutting tool rigidly.

USE OF FORM TOOLS

In lathe work, form tools are used for form turning, producing short tapers, thread chasing, and other applications. If a form tool is to reproduce the special contour of its cutting edge on the workpiece surface, it is important that the cutting edge should be adjusted so that it is at the same height as the rotational axis of the workpiece. This is known as adjusting the cutting edge on center. If a form . tool is set above or below center, a contour differing from that of the cutting edge will be produced on the surface of the workpiece. For the feed, a form tool is moved into the workpiece with the cross slide.

PARTING OR CUTTING OFF



TOOL GRIND ANGLE

Parting is the operation of severing a workpiece from a bar. It is done with a narrow cutting tool, which cuts only at its end. A parting tool should be adjusted so that its cutting edge is on center if it is to cut to the center of a bar. It should also be set so that no rubbing or cutting will occur along its sides.

GROOVING

A groove can be produced with a parting tool or with a form tool.

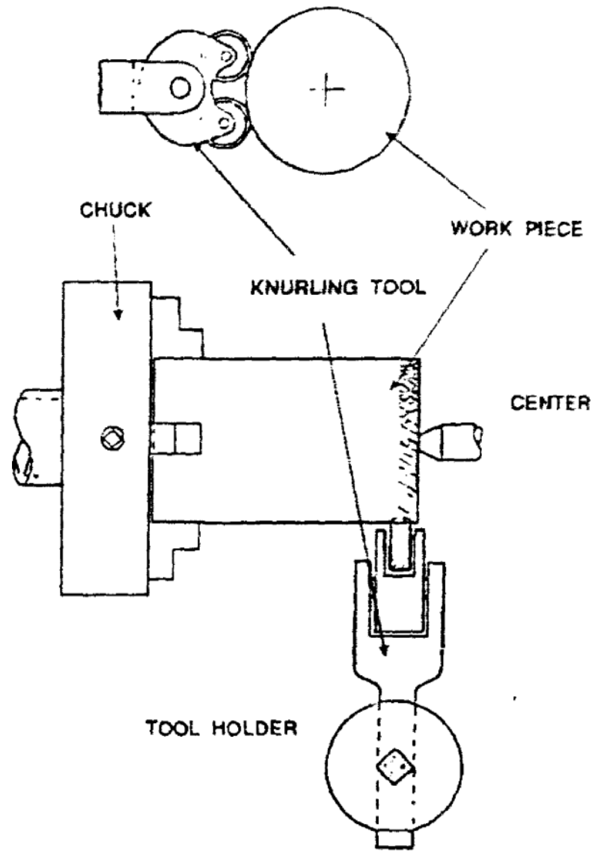
KNURLING

Making a raised diamond-shaped pattern on the cylindrical surface of a workpiece is known as “knurling”. Surfaces are often knurled so that they may be more easily gripped with the fingers or by hand. A knurling tool has two serrated hardened-steel rollers with the serrations slanted at 4-5 degrees from the horizontal.

The serrations of the roller are slanted at 90 degrees from those of the other roller. Both rollers are mounted on a floating head, which permit the rollers to exert equal pressure as they are pressed against a rotating workpiece. Knurling is a mechanical-working process, in which the metal on the surface of the workpiece is deformed rather than removed. Since greater forces are used in knurling, the workpiece should be well supported.

The workpiece, although held in a chuck, is given extra support by the center of the tailstock. When knurling, the knurling tool is forced to a workable depth into the workpiece surface with the cross slide, and then it is fed for the length of the desired knurl with the carriage.

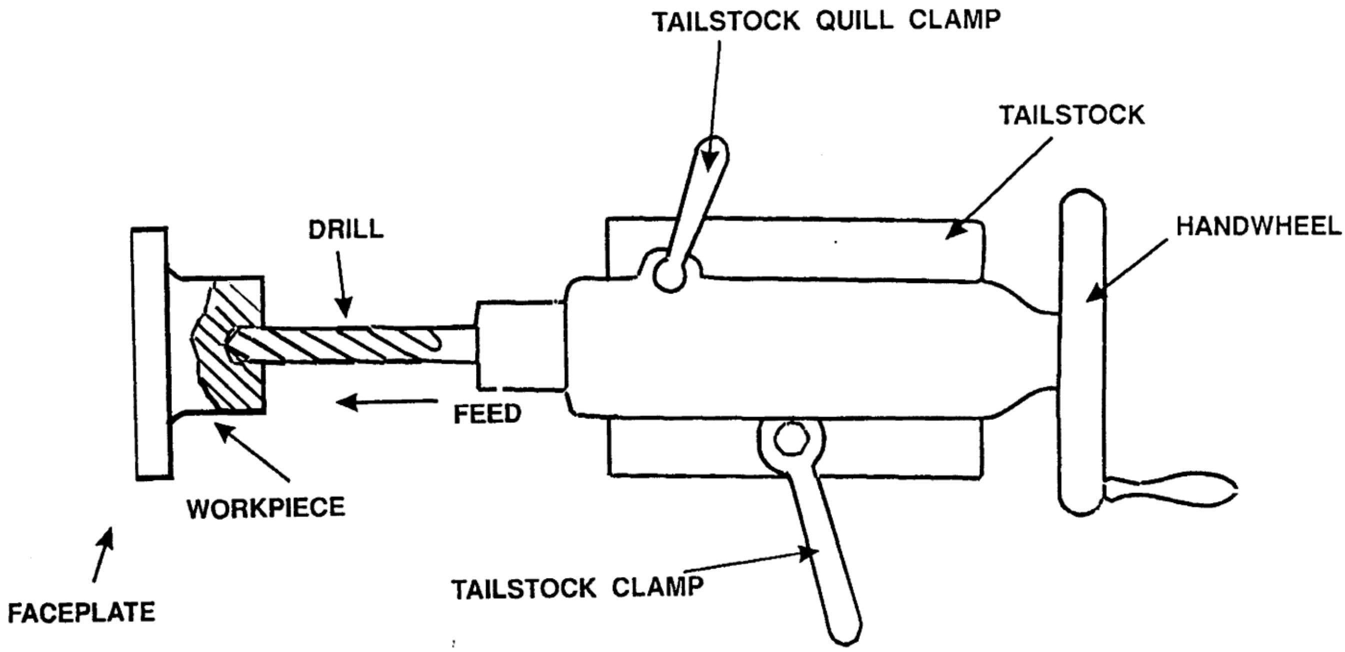
This is repeated until final depth and a finished knurl is obtained. Since workpieces with various diameters may be knurled, the circumference will seldom be equal to a whole number of the diamond shaped patterns Thus after the first complete revolution, some fraction of a diamond ordinarily remains. With continued knurling, this fraction of a diamond distributes itself evenly over the complete circumference, so that it cannot be detected.



KNURLING OPERATION

Knurling Steel

Drilling On A Lathe



DRILLING ON THE LATHE

DRILLING WITH A LATHE

A drill is held in the tapered hole of the tailstock quill, which should locate it concentric with the rotational axis of the spindle. The workpiece to be drilled must be positioned so the desired hole is concentric with the rotational axis of the spindle. Reamers, counterbores, and other cutting tools also may be held by the tailstock quill. They may be used after drilling.

PRODUCING TAPERS

The method to be used for producing a taper depends upon the length of the taper, included angle of the taper, number of workpieces to be produced, and the available tooling and attachments. A taper may be produced with the use of the compound rest. Since the liner movement of a compound rest slide is limited, this method is suitable only for tapers no longer than this movement.

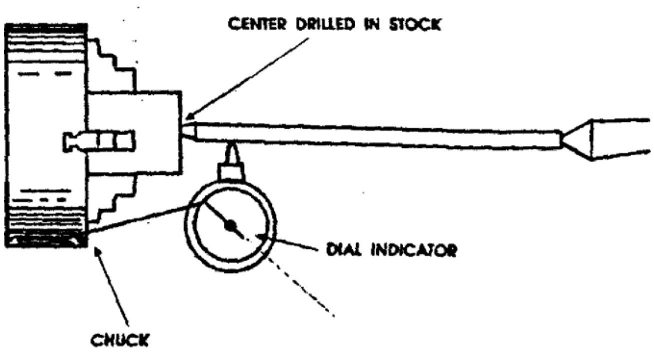
CHAPTER 5 TURNING STEEL

Turning stock usually makes up the majority of lathe work. The work usually is held between centers or in a chuck, and a right-hand turning tool is used, so that the cutting forces, resulting from feeding the tool from right to left, tend to force the workpiece against the headstock and thus provide better work support.

TEST BAR

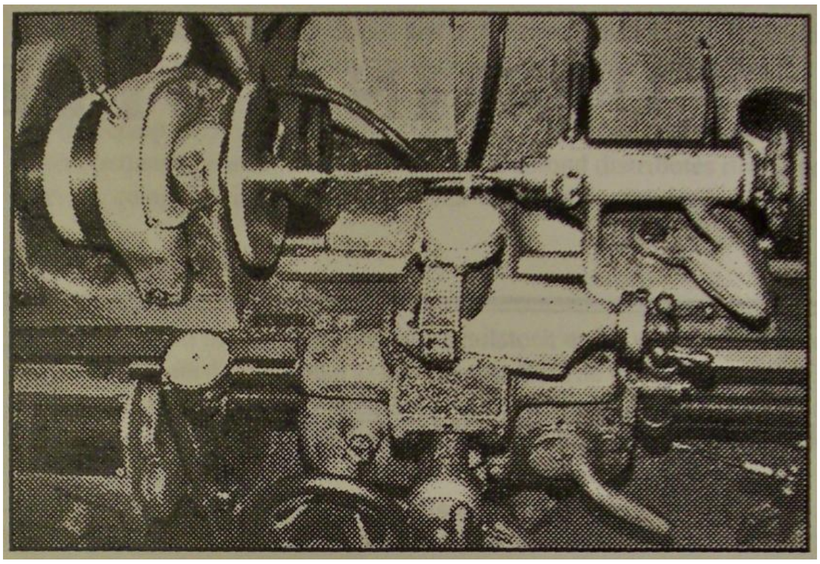
Before you start the turning operation, set the tail stock back to 000 using an 18 inch bar that is turned to exactly the same diameter on each end. To make this bar, get a 1 inch bar 18 inches long, center it and set it up between centers.

Make a light pass and check both ends to see if they measure the same. If not, adjust the tailstock and make another pass. Repeat the above operations until the bar measures the same on both ends.



SETTING ANGLE FOR TURNING TAPER

MAKING A TEST BAR



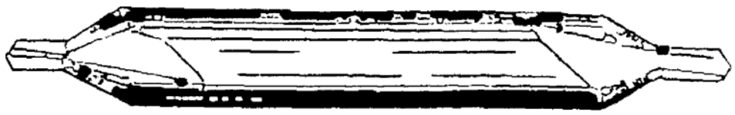
SETTING THE TAIL STOCK

This bar, you save as you will be using it again each time you true up the tailstock. Once you have the bar completed, all that is necessary is to put the bar between centers. Clamp a dial indicator to the carriage on the lathe.

With the plunger of the indicator on the bar, start from the headstock end (without the lathe being turned on) and move the carriage to the tail stock end. If there is any difference in size, adjust the tail stock and repeat until the dial indicator reads the same on each end.

MAKING A KEYWAY CUTTER

We will start out by making a ¾ inch keyway cutter. By doing so, this project though a simple one will go through all the operations that is normally done in a machine shop.



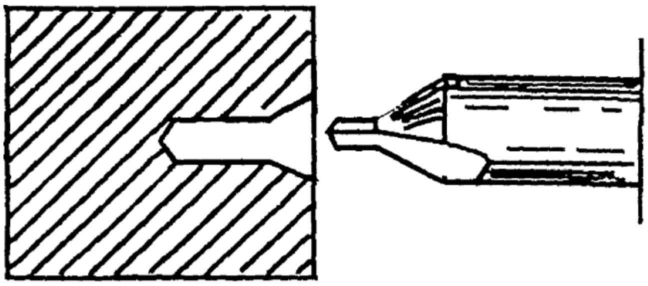
CENTER DRILL

High Speed Center Drill

First, you will need a piece of tool steel, preferably 01, which can be purchased from about any machine shop supply house. If you do not have one in your area, in the Appendix in the back of the book will give you the location of these. Wholesale Tool will have most all of the tools and supplies that you will need.

Once you have the 1 inch tool steel, cut off a piece 2 inches long. You will next need to cut centers in both ends for turning. Before a workpiece can be mounted between lathe centers, a 60° center hole must be drilled in each end. This can be done in a drill press, or in a lathe by holding the work in a chuck.

A combination center drill and countersink is used, taking care that the center hole is deep enough so that it will not be machined away in any facing operation, and yet is not drilled to the full depth of the tapered portion of the center drill. To cut the center in the lathe, chuck up the stock in a three jaw chuck, or if you are using a four jaw chuck center the stock using a Dial Indicator.



THE CENTER DRILL AND CENTER

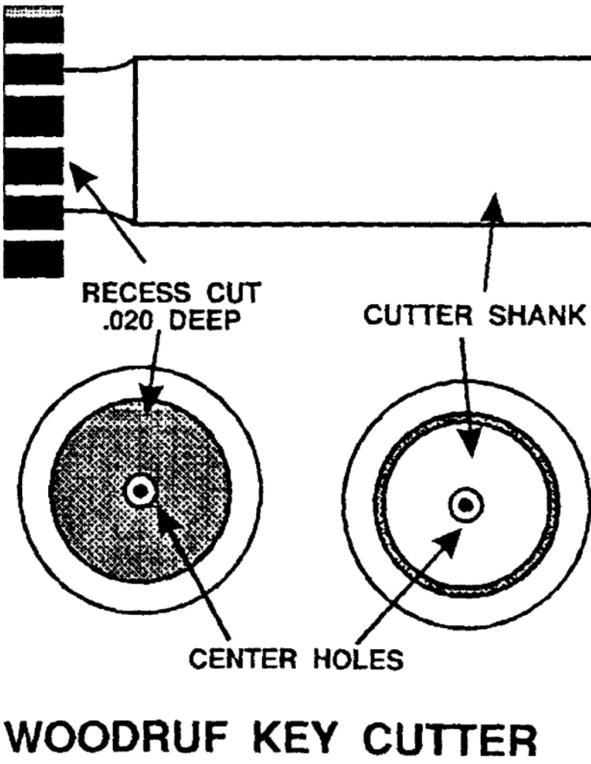
Center Drilled Stock

Set up a cutting tool in the lathe and face off each end of the tool steel stock. This is necessary so that when you center drill the stock the center drill will not cut off center. Once you have the stock trued, place a drill chuck in the tail stock of the lathe (the taper on the chuck will match the taper of the tail stock center.)

Centers have shanks with self-holding tapers, and they fit accurately into the tapered holes in the spindle and tailstock quill. When inserting a center, both the tapered shank and hole must be clean, because small chips or dirt particles will cause misalignment. The centers of a lathe must be accurately concentric with the rotational axis of the spindle, if accurate cylindrical surfaces are to be turned. Otherwise, a slight taper will be turned.

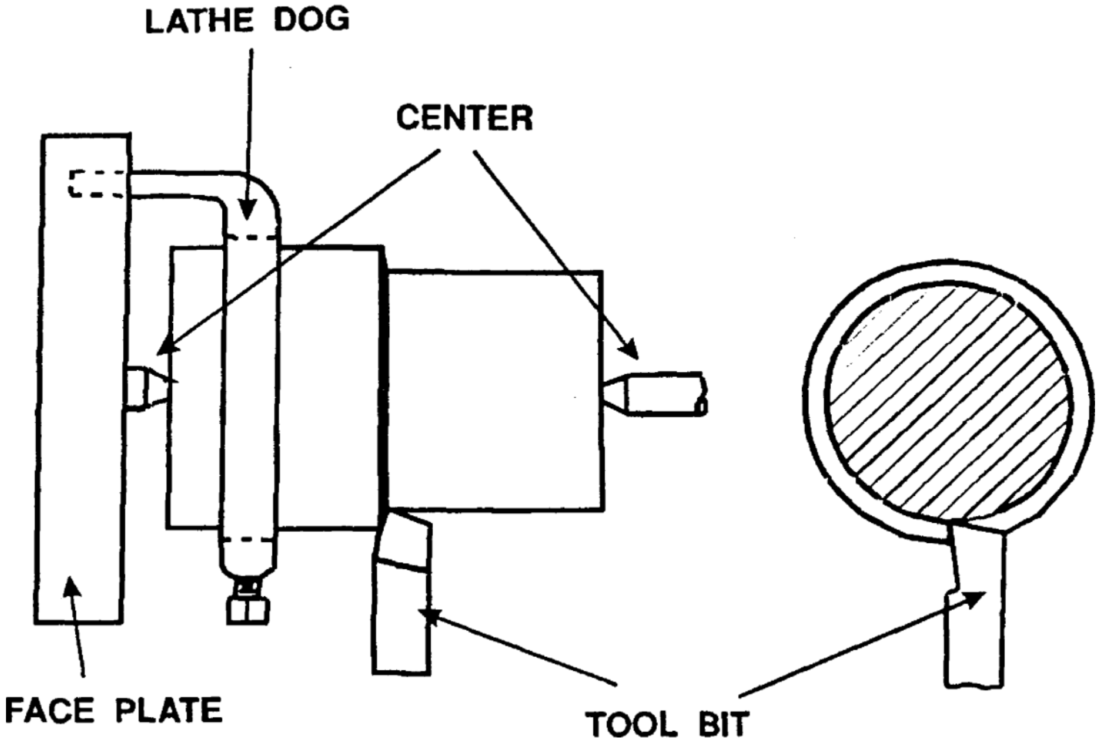
Use a small center drill to cut these centers. Put the center drill in the chuck in the tailstock and clamp it tight. Turn the lathe on with the speed set on the lowest speed (not in back gear) and move the tailstock so that the center is within 1/2 inch of the stock. Clamp the tailstock, and drill a center just deep enough so there is just about a 3/32 inch bevel on the edge of the center hole.

When you have the center cut on one end, turn off the lathe and turn the stock around and repeat the process on the other end. When both centers are cut, you can start to turn the tool steel to its correct shape.



WOODRUF CUTTER

A Face Plate can now be installed in place of the three or four jaw chuck and centers set in place. The work and the center at the headstock end rotate together, so no lubricant is needed in the center hole at this end. However, because the center in the tailstock quill is dead with respect to the rotating workpiece, adequate lubrication must be provided. This usually is accomplished by putting a mixture of white lead and oil, or with another type of lubricant in the center hole before the dead center is tightened in the hole. If you do not provide proper lubrication at all times, you will result in scoring of the workpiece center hole and the center, and inaccuracy and serious damage may occur to the centers.

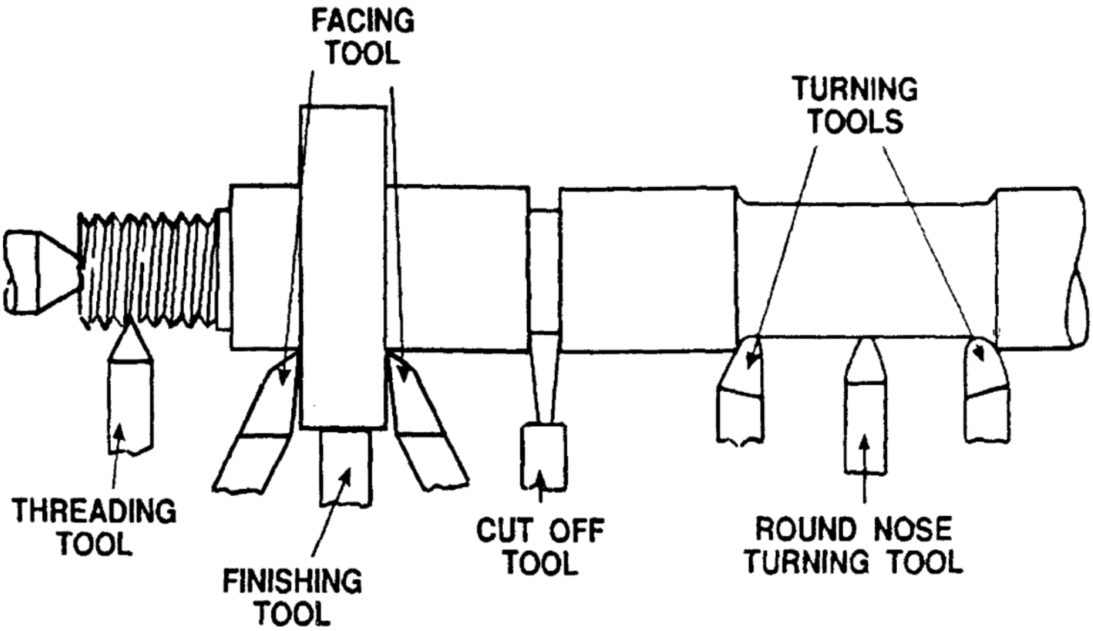


TURNING WORKPIECE WITH CENTERS

TURNING WORKPIECE

A connection must be provided between the spindle and the workpiece to cause it to rotate. This is accomplished by a lathe dog and a face plate. For this project you will need a 1 inch, and a 1/2 inch lathe dog. The dog is a forging that fits over the end of the workpiece and is clamped to it by means of a setscrew. The tail of the dog enters a slot in the dog plate, which is rigidly attached to the lathe spindle in the same manner as a lathe chuck. If the dog is attached to work that has a finished surface, a piece of soft metal, such as copper or aluminum, can be placed between the work and the setscrew to avoid marring.

Proper tightness must be maintained between the centers and the workpiece. The workpiece must rotate freely, yet no looseness should exist. Looseness usually will be first noticed by chattering of the material during cutting.



DIFFERENT TYPES OF CUTTING TOOLS

CUTTING TOOLS USED IN MACHINING

Tightness of the centers should be checked after cutting has been done for a short time. The resulting heating and thermal expansion of the workpiece will increase the tightness.

Live Centers are sometimes used in place of the dead center in the tailstock quill. The end that fits into the workpiece is mounted on ball or roller bearings so that it is free to rotate; thus no lubrication of the center hole is required. In most cases they may not be as accurate as the solid type, so they often are not used for precision work.

Clamp the 1 inch lathe dog to the stock. It is best to leave the dog loose around the stock for now. Put the stock between centers, and adjust the centers to where they are just snug. Now clamp the lath dog as far back on the stock as you can. These reason for this is that you need to turn the shank end down to ·0500 inch and 2 inches long. If you can't turn the stock to 2 inches long, turn to 1½ inches in length. The other 1/2 inch can be turned later.

If good finish and accurate size are needed, one or more roughing cuts usually are followed by one or more finish cuts. Roughing cuts may be as heavy as proper chip thickness, tool life, and lathe capacity permit. Large depths of cut and smaller feeds are preferred to the reverse procedure, because fewer cuts are required and less time is lost in reversing the carriage and resetting the tool for the following cut.

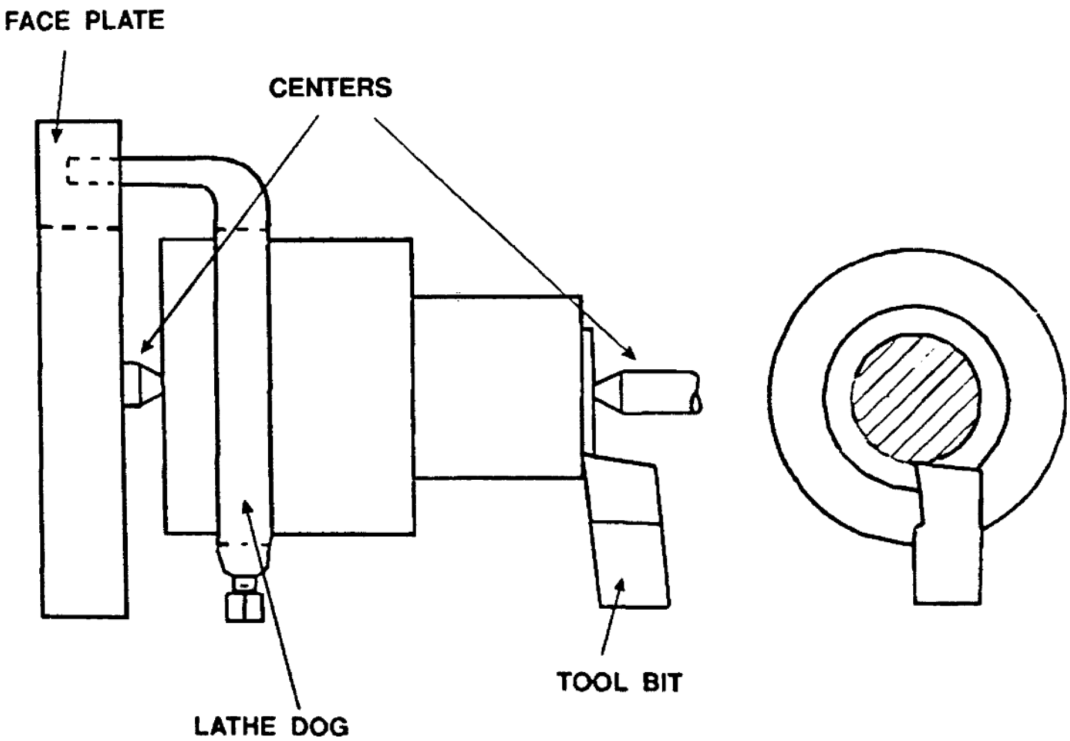
CUTTING SPEEDS	
Cast Steel	50 fpm
Bronze	70 fpm
Cast Iron	70 fpm
Malleable Iron	100 fpm
Mild Steel	100 fpm
Soft Brass	200 fpm
Aluminum	300 fpm

CUTTING SPEEDS

TURNING THE STOCK

Mark on the stock from the end (tailstock end) a mark with a file at 2 inches. Set the cutting tool for turning and just touch the point on the stock. Move the carriage back far enough to clear the work piece and move the compound feed in ·020 of an inch. In most cases this will remove ·040 from the stock on each pass.

Having the lathe running at the slowest speed in direct belt drive, engage the feed lever and start removing the stock. When the cutting tool reaches the mark you put on the stock, disengage the feed. Now, run the carriage back to where it just clears the work, set it in another ·020 and repeat the process.



FACING THE END OF BAR
Facing Stock

In turning operations, diameters usually are measured with micrometers, although spring calipers may be used to check roughing cuts or where close accuracy is not required. The method of making length measurements is made by spring, hermaphrodite, vernier, or micrometer calipers or micrometer depth gages can be used.

The shank will be finished to ·500 or 1/2 inch, however when you get the stock down to within ·030 you need to take a ·010 pas at a slower feed to clean up the work, If you have a Tool Post Grinder and plan to finish the work by grinding, stop about ·010 to ·015 oversize. If you do not have a grinder, you can take a fine cut at the slowest feed and finish the work to ·003 to ·005 oversize and use as is.

Remove the turned part around, remove the 1 inch dog and put the 1/2 inch dog on the turned end. Set the work piece back between the centers, adjust the dog and tighten it. Touch the point to the stock and move it slightly past the work, and set it in ·020. Engage the feed and take the stock down to ·780, then take the finer feed to finish it to ·015 over finish size. If you are going to use without grinding, take the stock down to about ·005, or ·755.

Next we have to finish the width to ·187 for the cutter. You need make a groove or recess on the shank next to the large end. This can be done with a cut off tool, or a square end cutting tool. Run the recess or groove to a depth of ·025 per side. Use a right hand tool holder and face off the inside of the large end to get a thickness of ·190. An undercut is also made on both sides for clearance.

Finishing cuts are light, usually being less than ·005 inch in depth, with the feed as fine as necessary to give the needed finish. Sometimes a special finishing tool is used, but often the same tool is used for both roughing and finishing cuts. In most cases one finishing cut is all that is required.

Where exceptional accuracy is required, two finishing cuts are usually used. If the diameter is controlled manually, make a short finishing cut about 1/2 inch long and check the diameter before completing the cut. Because the previous micrometer measurements were made on a rougher surface, some readjustment of the tool setting may be necessary in order to have the final measurement, made on a smoother surface, check exactly.

Turning is a lathe operation in which an external cylindrical surface is produced by cutting action of the tools. The cutting tool is first adjusted for the desired depth of cut, using the cross slide. Then, as the workpiece rotates, the cutting tool is advanced slowly in a direction parallel to the rotational axis of the spindle and this motion is known as the feed.

By adjusting the feed so that the helical path if the tool tip overlaps sufficiently, the cutting tool removes the excess material and generates a cylindrical surface on the workpiece, A spindle rpm that gives a desired cutting speed at the circumference of the cylindrical surface should he selected. This may be calculated using the following:

Spindle rpm = 12 X cutting speed, ft/min / pie x workpiece diameter (in.)

Feed is measured in thousandths of an inch advance of the cutting tool per revolution of the workpiece.

FACING.

Facing is the removing of metal on a flat surface as the result of the tool being fed across the end of the rotating workpiece. The work may be held in a chuck, a face plate, or between centers. Both ends of the work that is to be faced, must be turned end for end after the first end is completed and the facing operation repeated.

Because most facing operations are performed on surfaces that are away from the headstock, a right-hand tool is used most frequently. The spindle speed should be determined from the largest diameter of the surface to be faced. Facing may be done either from the outside inward or from the center outward. The point of the tool must be set exactly at the height of the center of rotation. Because the cutting force tends to push the tool away from the work, it usually is desirable to clamp the carriage to the lathe bed during each facing cut to prevent it from moving and thus producing a surface that is not flat.

CHAPTER 6 MILLING MACHINES

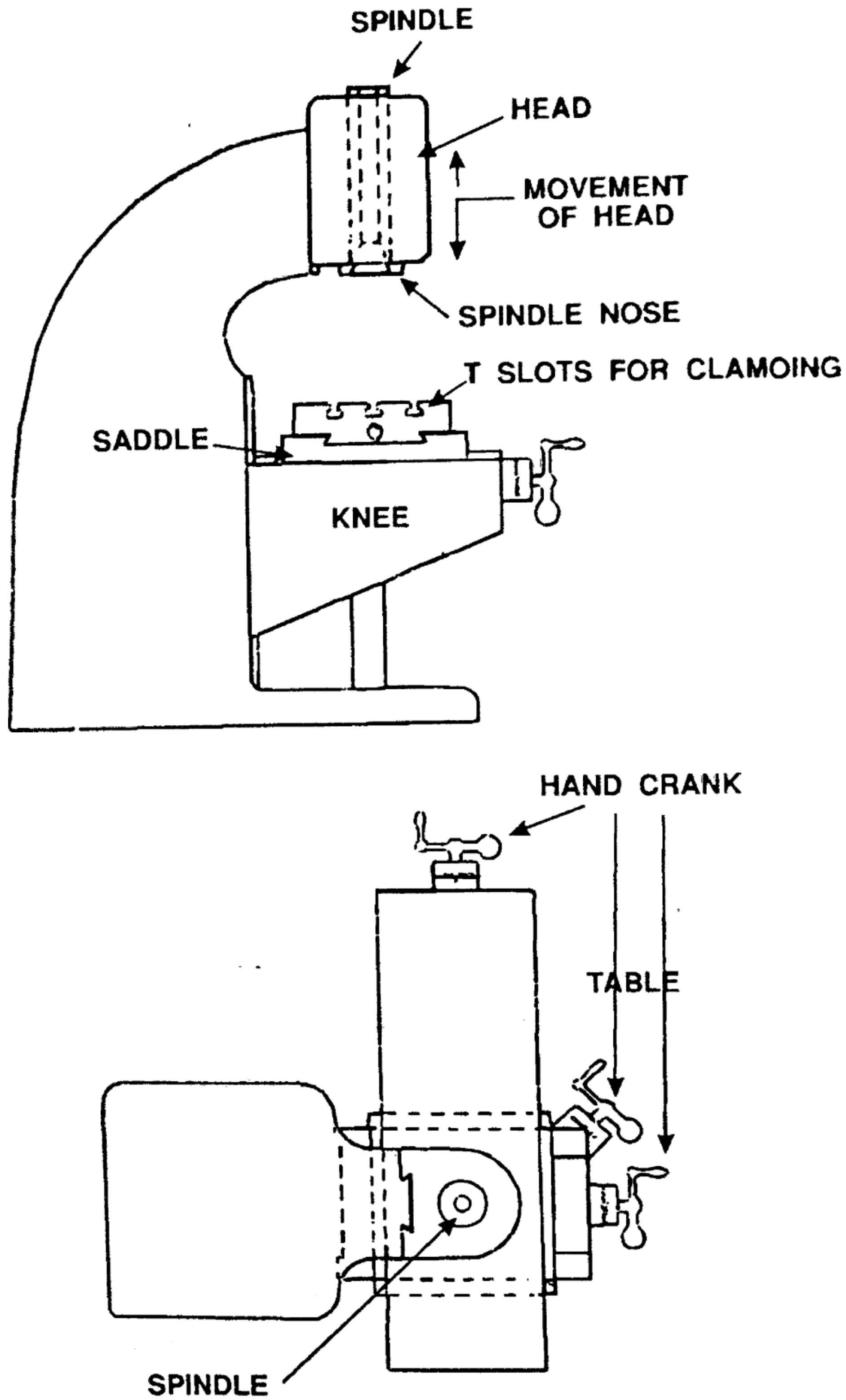
GENERAL TYPES

Hand milling machines may be of the column and knee type or constructed with a table mounted on a fixed bed. This type of machine is intended for small work only. The hand feed operates by means of levers or a hand screw for work such as slotting and cutting grooves and keyways.

The machine is provided with a horizontal spindle with speeds of 75 to 4,000 rpm (4- ranges). The worktable has longitudinal and vertical feeds also a crossfeed. A machine of this type can be used for production work if provided with stops and specially designed fixtures where parts can be rapidly loaded and unloaded.

Milling machines are designed to hold and rotate a milling cutter, hold a workpiece, and feed the workpiece to the milling cutter in one of several directions. The workpiece may be held directly, or indirectly, on the table of the milling machine.

The table and the workpiece may be moved or adjusted about the rotating milling cutter, in three directions, that is vertical, horizontal, parallel to the rotational axis of the spindle, and horizontal, perpendicular to the rotational axis of the spindle. Along any one of these three directions feeding may be accomplished.



VERTICAL MILLING MACHINE

Movements along the other two directions then are used for locating the cut, that includes obtaining the depth of cut. Adjusting the movements along these three directions can be controlled to within 0·001 inch.

Milling machines are available in several different types, and can be used for making a large variety of machining cuts. Milling machines with a horizontal spindle for rotating the milling cutter are called horizontal milling machines. Milling machines with a vertical spindle are called vertical milling machines that we will be using here.

VERTICAL MILLING MACHINES

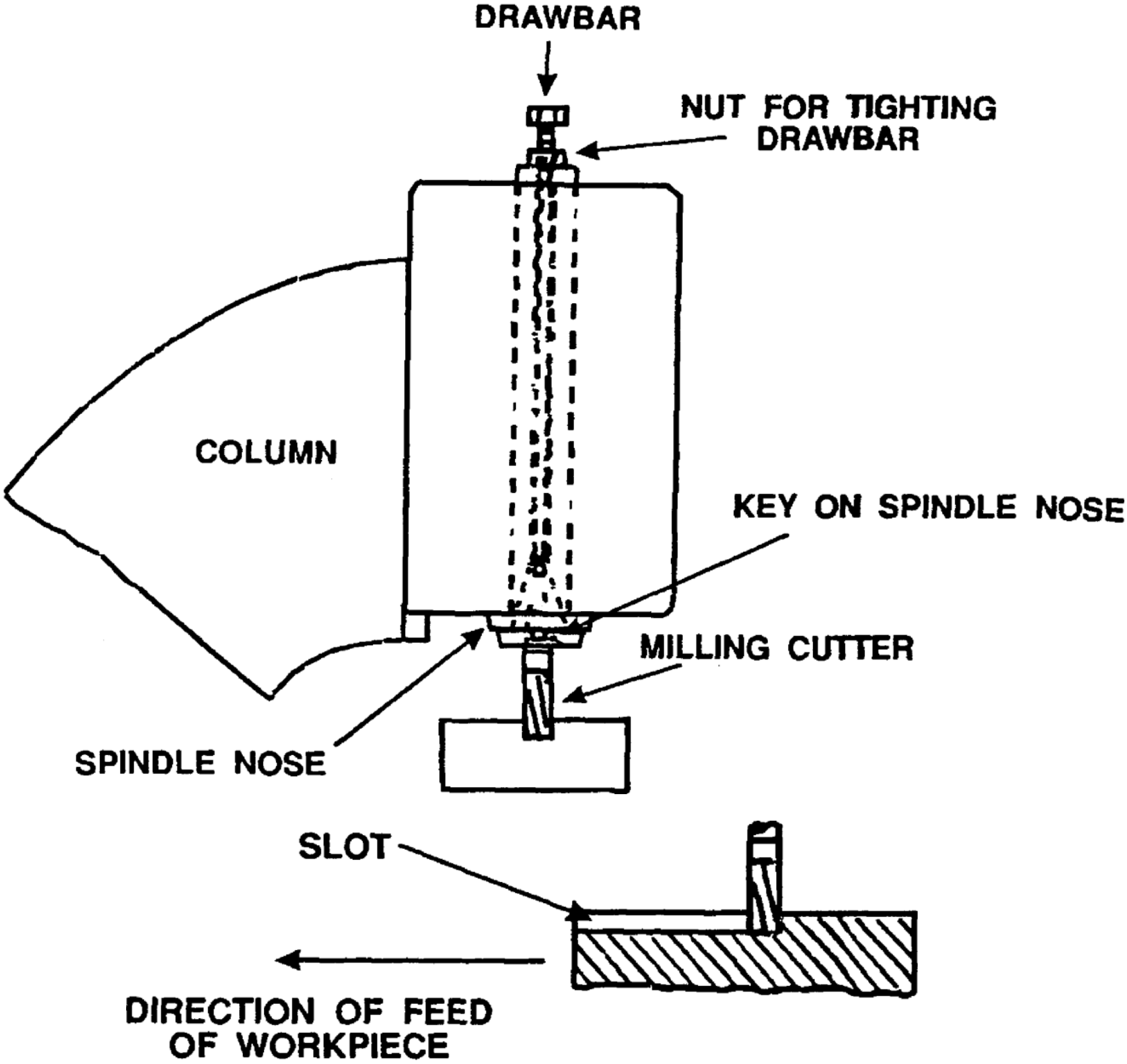
A vertical milling machine has the same table movements as a horizontal machine. It is called a vertical milling machine because the spindle is located vertically and at right angles to the top of the table. The head may be swiveled for angular or bevel milling operations.

Vertical milling machines use end-milling cutters of various types and sizes depending upon the kinds of operations to be performed. These operations consist of milling horizontal surfaces, angular surfaces, milling grooves, keyways, T-slots, and dovetails.

Vertical milling machines can also be used for drilling and boring operations where it is necessary to space a number of holes accurately. In this type of operation, dial gages, vernier scales, precision measuring pins, and rods can be used advantageously for producing precision holes.

The table, saddle, and knee portion of a vertical milling machine is the same as that of a horizontal milling machine. A vertical milling machine is not suitable for using arbor-mounting milling cutters, that must be mounted on an arbor. There is no provision for supporting the outer end of an arbor. Compared with a horizontal milling machine, a vertical milling machine can use shank- mounted milling cutters easier.

Milling Machine R. P. M. Necessary to Give a Desired Cutting Speed						
	Cutting Speeds in Feet per Minute					
Diameter (Inches)	40	50	60	70	80	90
	Revolutions per Minute					
¼	611	764	917	1,070	1,222	1,375
5/16	489	611	733	856	978	1,100
⅜	407	509	611	713	815	917
7/16	349	437	524	611	698	786
½	306	382	458	535	611	688
⅝	244	306	367	428	489	550
¾	204	255	306	357	407	458
⅞	175	218	262	306	349	393
1	153	191	229	267	306	344
1⅛	136	170	204	238	272	306
1¼	122	153	183	214	244	275
1⅜	111	139	167	194	222	250
1½	102	127	153	178	204	229
1⅝	94	117	141	165	188	212
1¾	87	109	131	153	175	196
1⅞	81	102	122	143	163	183
2	76	95	115	134	153	172
2¼	68	85	102	119	136	153
2½	61	76	92	107	122	137
2¾	56	69	83	97	111	125
3	51	64	76	89	102	115
3½	44	55	65	76	87	98
4	38	48	57	67	76	86
4½	34	42	51	59	68	77
5	31	38	46	54	61	69
5½	28	35	42	49	56	63
6	25	32	28	45	51	57
7	22	27	33	38	44	49
8	19	24	29	33	38	43
9	17	21	25	30	34	38
10	15	19	23	27	31	34
11	14	17	21	24	28	31
12	13	16	19	22	25	29
13	12	15	18	21	24	27
16	10	12	14	17	19	22
17	8	11	13	15	17	19
48						



VERTICAL MILL AND END MILL

Milling Machine Head

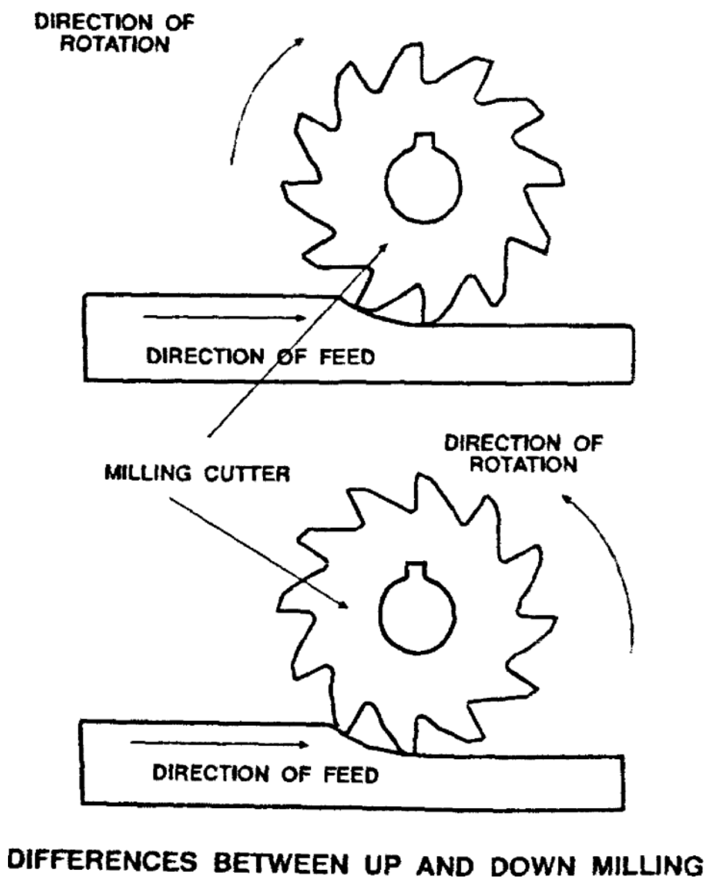
Using a shank-mounted milling cutter on a vertical milling machine, the operator can more easily set up the workpiece and observe the machining. On some vertical milling machines the head, that contains the spindle, may be swiveled about a horizontal axis. The milling cutter may then be set at any angle in a vertical plane parallel to the direction of table movement.

All milling machines have an electric motor, housed in the column, to provide power, through suitable gearing and a clutch, for rotating the spindle. The gearing provides a means for obtaining different speeds (rpm * s) for the spindle for different cutter diameters and machining conditions. Power from the electric motor, through the gear train, can be used for moving the table, saddle, or knee. This gearing may be quickly changed to get a variety of desired movement velocities. When used during machining, this is called power feeds. A more rapid movement of the table, saddle, or knee, a rapid traverse is available. This is used when setting up a milling machine. Accurate positioning of the table, saddle, and knee during setup is set by hand, using the hand cranks and micrometer dials.

CUTTER TEETH

Cutters with comparatively few widely spaced teeth have distinct advantages over fine-toothed cutters.,

A coarse-toothed cutter with few widely spaced teeth can remove a maximum amount of metal, without distressing the cutter or overloading the machine. These cutters have a free cutting action, largely because a smaller amount of cutting is required to remove a given amount of metal.



Up And Down Milling

SPEEDS				
Material	Brn. Range	H.S.S. Cutter S.F.M. Range	Cast Alloy Cutter S.F.M. Range	Carbide Cutter S.F.M. Range
Aluminum	100—150	1000—550	2000—1100	4000—2200
Brass	100—175	650—250	1300—500	2600—1000
Low-Carbon Steel	100—200	325—100	650—200	1300—400
Free-Cutting Steel	150—200	250—150	500—300	1000—600
Alloy Steel	150—250	175—70	350—140	700—280
Alloy Steel	250—350	70—40	140—80	280—160
Cast Iron	125—175	100—40	200—120	400—240
Cast Iron	175—200	60—45	120—90	240—180
Cast Iron	200—225	45—40	90—80	180—160
Cast Iron	225—250	40—35	80—70	160—140

Cutting Speeds For Mills

Other advantages are:

The rake and increased spiral of the teeth gives a shearing action. Wide spacing decreases the tendency of the cutter to slide over the surface. Less friction is created, resulting in cooler teeth and consequently decreasing the necessity of regrinding operations. There is decreased power consumption. Increased production is possible.

Positive radial rake angles of 10° to 15° are used on high-speed steel cutters. These angles serve in machining most materials and give good cutting ability to the cutter without sacrificing strength of the cutter. In milling softer materials, a greater rake angle can be provided to improve cutting ability.

Negative rake angles are provided on carbide-tipped cutters for high-speed milling operations. Since the angles are both radial and axial, tool life can be increased by increasing the lip angle. For softer steels, a negative rake angle of 5° to 10° is provided on plain milling cutters with teeth on the periphery. This angle is increased when medium-carbon and alloy steels are being machined.

Clearance angles are kept on the small side to avoid weakening the cutting edge of the tooth. With a minimum amount of material in back of the tooth, the strength of the tooth is diminished. Clearance angles of 3″ to 5″ are used on cutters over 3″ in diameter. This is increased on smaller diameter cutters to prevent the teeth from a rubbing instead of a cutting action.

The type of material being machined affects clearance angles. If cast iron is being machined, 4° to 7° might be used; non-ferrous materials require clearance angles of 10° to 12°. The land on a cutter can be from 1/32″ to 1/16″ in width, with a secondary clearance back of the land.

ARBOR-MOUNTING MILLING CUTTERS

A milling machine arbor has a shank with a locating taper for locating it so that it will rotate concentrically with the spindle. The arbor shank is driven by a key on the spindle nose. It is held to the spindle nose by a draw bar that extends through the hollow spindle. After screwing the drawbar into the end of the arbor shank for at least four full threads. The nut is then tightened to hold the arbor firmly in the taper of the spindle nose.

An arbor-mounting milling cutter has a central hole that closely fits an arbor diameter. A nut at the outer end of the arbor is turned for tightening all collars and cutters on the arbor. Running the length of the accurate cylindrical portion upon that the milling cutters are located, milling machine arbors have a keyway, and thus a key may be used to drive a milling cutter.

Often with lighter cuts and especially with hand feeding, a key is not used. If a milling cutter driven without a key slips when power feed is being used, the amount of material to be removed by the next cutter tooth may be increased. This may cause either more slippage or possible breakage.

PLAIN MILLING CUTTERS

Plain milling cutters are cylindrical with teeth on the periphery only. The periphery of a milling cutter is the imaginary cylindrical surface enveloping the outer ends of the peripheral teeth and determining the diameter of the cutter. These cutters are used primarily for milling flat surfaces. They can be combined with cutters of other types to produce surfaces with various forms. The teeth may be either straight or helical, depending upon the width of the cutter. Plain milling cutters with helix angles of 45° to 60° and higher are called helical cutters.

ANGULAR MILLING CUTTERS

Angular milling cutters are used for operations such as: cutting V-grooves, notches, dovetails, flutes on milling cutters, and reamer teeth. Single-angle cutters have one angular surface while double-angle cutters are made with Y-shaped teeth. These cutters, with equal conical angles on both faces, are made with an included angle of 45°, 60°, or 90°.

PLAIN MILLING CUTTERS

Arbor-mounting cutters are cylindrical in form and provided with cutting edges of their outer cylindrical surfaces. There are no cutting edges on either side of a plain milling cutter. Plain milling cutters are used normally for machining flat surfaces.

Arbor-mounting cutters with small widths, ranging from a few thousandths up to 3/16 inch, are called slitting saws. They are used for cutting off and narrow slotting operations. Most slitting saws are similar to plain milling cutters as they have cutting edges only on their outer cylindrical surfaces. These slitting saws are ground slightly concave on their sides to provide side clearance so that their sides will not rub. Some slitting saws, especially those with greater widths, nearer 3/16 inch, are used as side milling cutters.

FLY CUTTERS

A fly cutter consists of one or more single-point tool bits mounted in a bar of some type that can be attached to the spindle of the milling machine. Its principle in operation is quite like that of a boring tool. Set screws are used to hold the tool bit in place, This type of tool is used for special applications.

T-SLOT CUTTERS

T-slot cutters are a special type of end mill having either straight or tapered shanks and designed for cutting T-slots in machine tables and similar applications.

NOTE: In producing a T-slot, a groove for the narrow portion of the slot is first machined with an end mill or side mill and then finished with the T-slot cutter.

WOODRUFF KEY SEAT CUTTERS

These cutters are of special design for cutting key seats for Woodruff keys (that have the shape of a half circle). These are available in all sizes and are of two types, end mill and arbor cutters. The end mill is available in diameters from 1/4″ to 1½″; the arbor type, in diameters from 2⅛″ to 3½″.

SIDE MILLING CUTTERS

Cylindrical in form, side milling cutters have cutting edges on one or both sides also on their outer-cylindrical surface. Side milling cutters are quite similar to plain cutters. They also have teeth on one or both sides. In milling operations where two cutters are placed side by side, they have teeth on only one side. The teeth can be straight, helical, or staggered.

Slots machined with side milling cutters have smoother and more accurate sides than those machined with plain milling cutters. Rake angle for the cutting edges at the sides of a side milling cutter is called the axial rake angle. It is the angle at the cutting edge between the tooth face and the machined surface.

METAL-SLITTING SAWS

Metal-slitting saws are designed for cutoff operations and for cutting narrow slots. The sides are slightly tapered toward the hole to prevent binding. Like other milling cutters, they can be plain or made with side teeth or with staggered teeth.

HOLDING THE WORKPIECE ON THE TABLE

Since more than one cutting edge of a milling cutter is cutting, the total cutting force of the workpiece can be large. A machinist needs considerable skill and experience to enable him to securely clamp some types of workpieces. A workpiece must be held securely so that it cannot shift during a cut. A workpiece should also be supported to prevent any springing due to the cutting force, the clamping, or its own weight. A workpiece is usually clamped to the table using the T slots. Smaller workpieces can be held in a vise bolted to the table. There are several types of vises that can be used including the plain vise, swivel vise, and the toolmaker's universal vise.

Most milling vises have two keys on their bases for fitting into a T slot for locating the vise on the milling machine table. Standard vise jaws are flat. They can be removed and replaced with special vise jaws, designed for locating and holding the workpiece. These special vise jaws have locating stops, that make possible easy location of the workpieces. The special jaws convert a vise into a milling fixture.

UP MILLING AND DOWN MILLING

DOWN MILLING

If down milling is used, all looseness must be eliminated in the table feed screw, as the motion of the cutter tends to pull the workpiece into the cutter. The machine must be designed with special features, adapting it to down milling if this type of milling is to be used.

In down milling, the maximum chip thickness is obtained close to where the tooth contacts the workpiece. No built-up pressure is developed in down milling, and, therefore, no heavy burr (a protruding, ragged metal edge) forms on the surface of the metal.

Down milling that is being done depends upon the side from that the workpiece is fed to the rotating milling cutter. In down milling, the portion of the tooth contact with the workpiece shows a very good finish. An element of the final milled surface is produced at the end of the tooth travel when the built-up edge is completely developed. This could mean that the finish of the final surface might be of poorer quality than produced by up milling.

UP MILLING

In up milling, the cutter rotates against the direction of feed as the workpiece advances toward it from the side where the teeth are moving upward. The separating forces produced between cutter and workpiece oppose the motion of work.

In up milling, since the cutter teeth come up from the bottom of the cut, the chip is very thin. It the beginning where the tooth first contacts the workpiece. Gradually, the chip increases in thickness, reaching its maximum thickness where the tooth leaves the workpiece.

In up milling, the material removed by each tooth starts with a minimum thickness and ends with a maximum thickness.

The chip should form at the center, but due to the resistance of the material to penetration, the cutting action is delayed somewhat and cutting starts slightly ahead of the center. The cutter slides over the workpiece to be machined until sufficient pressure has been built up to force it to bite into the surface of the workpiece to produce a chip.

The starting of the cutting in up milling is not recommended as the cutting edge of a tooth rubs along the workpiece surface at the start, and the beginning of the cutting is difficult. The opposite cutting condition, or down milling, is better, since cutting edges remain sharp longer, and smoother surfaces can usually be obtained.

Up milling is more commonly used because it is safer. With down milling, damage may be caused to the milling cutter, workpiece, and milling machine. Down milling the resultant force of the cutter upon the workpiece is directed toward and under the cutter.

This pulls the workpiece under the cutter. It is better to have this resultant force directed in the opposite direction as with up milling. If the workpiece is not securely held, it will be drawn into the cutter so fast that the cutter teeth are unable to make the cut and something will be damaged. If the workpiece is held properly, damage can still occur.

The entire milling machine table will be pulled ahead if any there is any play, that is known as backlash, exists. The next cutter tooth will probably have too much material to remove, and the cutter will be chipped or broken. Since down milling is better since it can be done safely, some milling machines are designed for it. All backlash must be eliminated.

CUTTING SPEED, FEED, AND DEPTH OF CUT

SPEEDS AND FEEDS

Cutting speed as applied to milling can be defined as circumferential speed of the milling cutter expressed in surface feet per minute (sfpm). It is the distance that the periphery of a milling cutter tooth travels in one minute.

The revolutions (rpm) refers to the number of revolutions that the cutter makes in one minute. A small milling cutter must rotate at a higher rpm to cut at the given cutting speed of a larger cutter. A small cutter is more efficient because it travels a shorter distance.

To calculate the proper rpm for the spindle, a formula similar to that is used for lathe and drilling operations is used.

RPM of the milling cutter = 12 X cutting speed, ft/min / Pie X diameter of the cutter, in.

Feed is the rate at that a workpiece is moved toward a rotating milling cutter, that removes material from its surface. Feed is limited by the depth of the material that can be removed by each tooth of the milling cutter per revolution. This depth is called the feed per tooth, and its units are inches per tooth. For calculating the feed in inches per minute for a milling machine, the following formula is employed:

Feed, in./min = feed per tooth in. tooth) x no. of teeth on milling / Cutter x rpm of milling cutter

Feed for milling machines is given in inches per minute, because various milling cutters with different number of teeth may be employed on a milling machine. The desired feed in inches per minute is set by quick-change gears, and the power feeds are engaged by control levers at the front of the machine. Power feeds are usually available for moving the table, saddle, and knee.

Trip dogs are set to disengage power feeds at the correct positions. They are especially useful when more than one similar workpiece is to be machined. Ordinarily it is best to use the largest feed per tooth that can be employed safely. By doing this it will reduce the time required for a milling operation and increase the life of a milling cutter between resharpenings.

Milling cutter life is increased, because the number of cutter-tooth contacts with the workpiece surfaces are reduced. A light feed may have to be used for a fragile workpiece or when it is difficult to hold a workpiece securely. Depth of cut is the normal distance between the workpiece surfaces before and after milling.

ADJUSTING THE LOCATION OF THE WORKPIECE TO THE CUTTER

A surface of the workpiece should be located by adjustment so that it just contacts the milling cutter when the latter is rotating. I use a thin piece of paper, of a known thickness. It is held on the workpiece surface, while the workpiece is carefully moved toward the rotating cutter. Once this contact is made, the workpiece is moved accurately a desired distance from this reference to remove the desired depth of material. An edge finder can be used to locate the surface with better accuracy.

The depth of material removed can be held to within 0·001 inch. Tolerance of from 0·002 to 0·005 inch is more practicable for milling. To align a machined surface of a workpiece, vise, or milling fixture with the movement of the table, saddle, or knee, a dial indicator is attached to the spindle or arbor. This test indicator is positioned to contact the machined surface.

The table, saddle, or knee then is moved, and slight changes of the machined-surface location are made, usually with light hammer blows. This is done until the reading of the test indicator either does not change or remains within the desired tolerance.

ATTACHMENTS FOR MILLING MACHINES

The principle function of attachments is to increase the variety of work that can be accomplished on milling machines. These attachments position and hold the workpiece to the table. Two other important milling-machine attachments are the rotary table and the dividing head. These will be discussed in following paragraphs.

ANGLE PLATE

If your milling machine head doesn't rotate, the adjustable angle plate can be used. It is bolted to the table of the milling machine, and any angle can be set on it. A vise can then be bolted to it to hold small work.

ROTARY TABLE

A rotary table is mounted on the table of a milling machine as an auxiliary table to superimpose a rotary motion upon the other movements for the workpiece. This rotary movement may be used for feeding or for adjustment in locating a cut. This rotary movement is about a vertical axis, and since the rotary table is mounted on the regular milling machine table.

A worm gear directly fastened to the rotary-table vertical shaft is rotated by a worm on a horizontal shaft. The horizontal worm shaft may be turned by hand. If power is used suitable shafting may bring the power, when desired, from the milling machine feed-power mechanism. The workpiece may be accurately rotated by turning the worm shaft by hand with the aid of a micrometer dial, or an index plate.

Fractions of a complete worm-shaft turn are obtained with a micrometer dial or with an index plate on the worm shaft. An index plate is a disk and has in its surface several holes, arranged in circles. The holes within each circle are equally spaced with accuracy. The circles of holes are concentric with the worm shaft. A spring-loaded pin in the crank handle is adjustable to fit into holes of a particular circle to be used.

The index plate with the crank and spring loaded pin provide an accurate means for obtaining desired fractions of a complete worm-shaft turn. One space between two adjacent holes in a particular circle may represent several degrees or minutes of angular movement of the rotary table. A vertical milling machine provided with a rotary table may machine a complete or partial cylindrical surface having any desired radius. Internal cylindrical surfaces with radius that are too small for available end mills may not be cut. Partial cylindrical surfaces may be joined to flat surfaces by using rotary-table feeds with the regular milling machine feeds.

UNIVERSAL DIVIDING HEAD

Like a rotary table, a universal dividing head may be used for rotating a workpiece through precise angles. A dividing head has a spindle that supports and rotates the workpiece. A center may be mounted in the tapered hole at the front end of the dividing-head spindle for holding a workpiece between centers.

Other workpiece holding devices, such as a chuck, also may be mounted on the front end of the dividing-head spindle. The workpiece is mounted so that it will rotate accurately concentric with the axis of rotation of the dividing- head spindle. The workpiece is rotated by turning the index crank. Since the gear ratio is 40 to 1, it takes, 40 turns of the crank to rotate the spindle and workpiece through one complete revolution. In comparison with a rotary table, that rotates only about a vertical axis. A dividing head may rotate a workpiece about an axis at any angle, and its indexing ability is much greater. The dividing head spindle axis can be positioned at various angles from below the horizontal to slightly beyond the vertical position.

CHAPTER 7 MAKING CUTTING TOOLS

All the processes that are described in my books involve making special tools. It is important that you become proficient in the art of tool making.

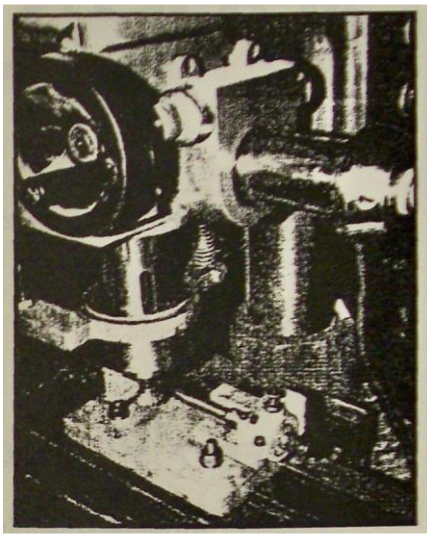
It is very important and almost a requirement that you also have a Tool Post Grinder if you make tools. In my books I describe how to make the Tool Post Grinder, special jigs and fixtures as well as Heat Treat Furnaces.

Tool Making is not difficult, it just takes time to learn how to do it. Once you understand the processes involved, all tools, etc. that you need can be made quickly. No more turning away a job because of not knowing where to find the tooling.

There are many machining operations that are done in the shop, and many are done with reamers or other cutting tools. Most of the time you will be able to purchase standard “Jobber” reamers and cutters. When you can't, then you must make what you need. What I describe here is for making reamers, but all cutting tools are made the same.

TYPES OF REAMERS

Reamers are made with both straight and helical flutes. The latter provided a shearing cut and is especially useful in reaming holes having key ways or grooves. These are bridged over by the helical flutes, thus preventing binding or chattering. Hand reamers are made in both solid and expansion forms.



MILLING A CUTTER

SHAPE OF FLUTES

Style and shape of the flute determine its ability to carry away chips and also the relative strength of the tooth. For manufacturing a straight shank may be used. With fluting reamers, the cutter is so set about to the center of the reamer blank so the tooth gets a slight negative rake. The amount is so selected that a tangent to the circumference of the reamer at the cutting point makes an angle of about 95 degrees with the front face of the cutting edge.

When fluting reamers, it is necessary to break up the flutes that are to space the cutting edges uneven around the reamer. The difference in spacing should be very slight and need not to exceed about ·004 either way.

The manner that you use to break up the flutes, is if the reamers are made on a mill is to set the cutter anywhere from ·002 to ·004 in front of the center of the reamer blank, and changing it a ·001 or so on each flute.

The relief of the cutting edges should be comparatively slight. They can be ground close on the tool post grinder and stoned flat with an Arkansas stone. The flat relief is what I have used all along, because the reamer has a keener cutting edge.

CHATTER

This can be the one thing in machining that will cause you the most trouble. Even if the reamer and relief are just about perfect, you can still get chatter. The first thing you need to do when using a new reamer is to spray the reamer with layout fluid. If the reamer is chattering, you will when you remove the reamer be able to see if you have enough clearance, etc.

Sometimes a too sharp a reamer will chatter, or it may have too much clearance. If the shank on the reamer is not a close enough a fit, that can cause chatter. Sometimes reducing or increasing the speed or feed will help. I cut all of my reamers on “0” rake and have very little trouble. If you are using commercial reamers, there can be too much positive or negative rake, and that can cause chatter.

REAMING SPEEDS

In most of my reaming I find that a good starting point is about two-thirds the drilling speed for a given material. If too high a feed or speed premature dulling, chatter, and usually a rough a rough finish is the result.

REAMING FEEDS

In reaming, feeds are usually much higher than those employed for drilling, often being 200 to 300% greater. Too low a feed may result in excessive reamer wear. Always the feed must be high enough so that the tool cuts, rather than rubs. I find that too high a feed will cause the hole to be oversize, and rough. I also have that a good starting point is somewhere between ·0015 to ·004 feed per flute per revolution.

STOCK ALLOWANCES

Usually only leave enough material to clean up your work.

LUBRICANTS

This is a very an important as the type of lubricant that you use for reaming will determine the quality of the reamed bore. The lubricant is used to cool the reamer, remove the chips, and to improve the finish of the work. Normally for most steel a sulfurize oil, or a high E.P. value mineral oil. Contact a business that supplies cutting oil and they will be able to help you. You will have times when you are unable to get a smooth reamed hole.

CHATTERING

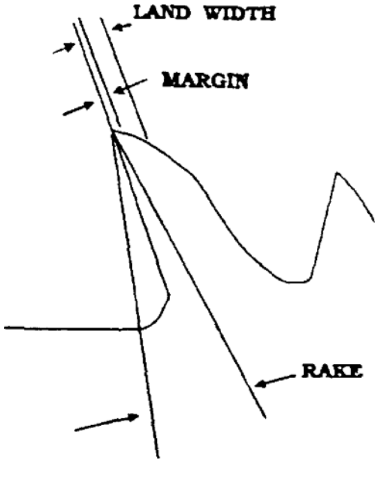
When the flutes are not evenly stoned, or flutes cut back of center line, stoned with too great a clearance. Chips clinging to the flutes - caused by too high a revolving velocity.

REAMER PLUGGING

Reamer plugging up by not having enough oil pressure and flow, flutes not being stoned out, and allowing saw teeth to form on the cutting edge. Enlarged holes caused by the reamer being oversized.

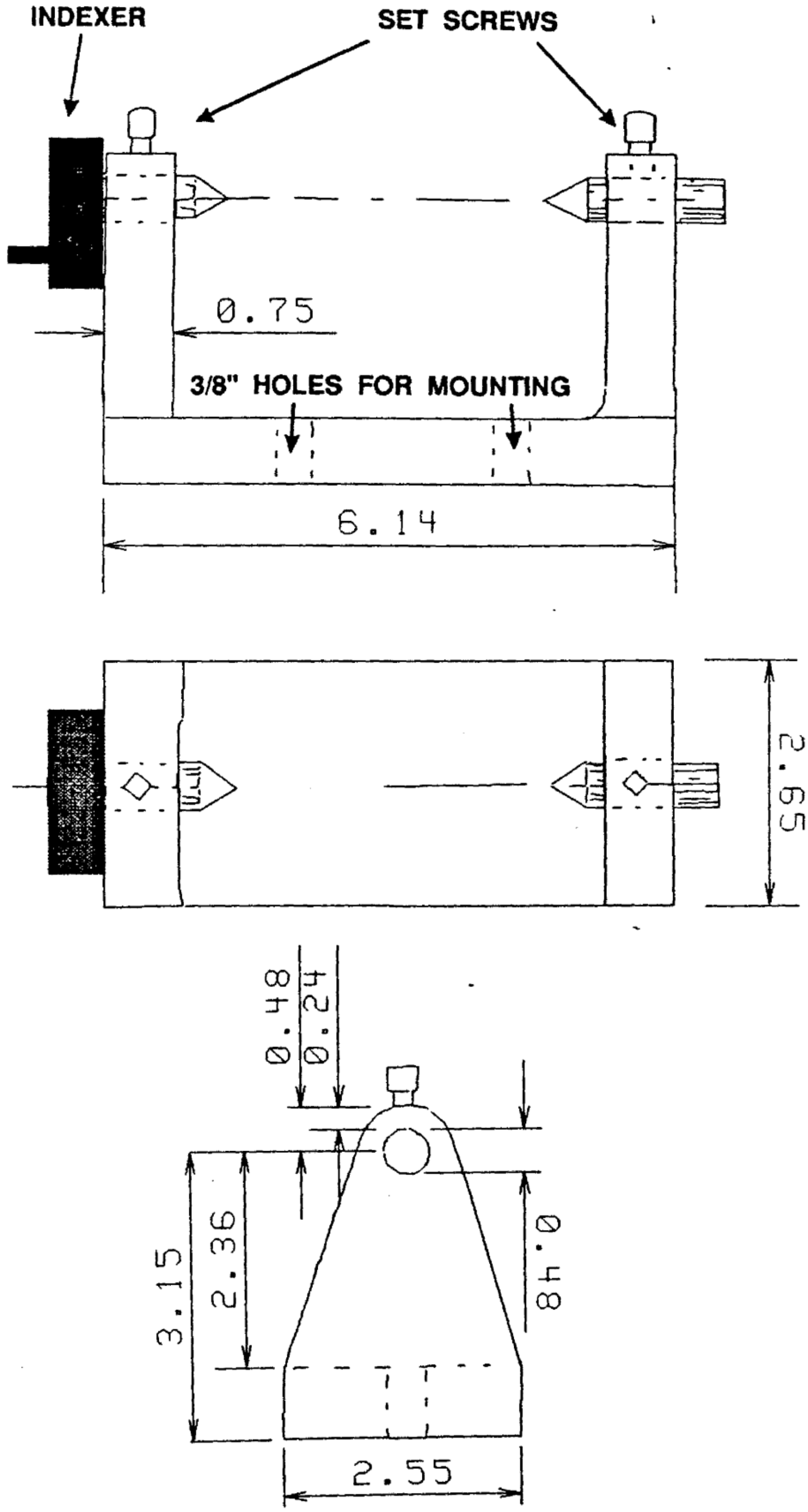
ARKANSAS STONE

The cutting edge of the reamer is kept sharp by honing with an Arkansas stone over the cutting edges. The reamer should be touched up about every two barrels.



REAMER FLUTE SHAPE & DESIGN

Shape of Reamer Flutes



INDEXING FIXTURE

SURFACE SPEED

The surface speed for reaming should be rather slow. Instead of using an open belt and higher speed, the lathe, drill, or milling machine is set in back gear, and run at the slowest speed.

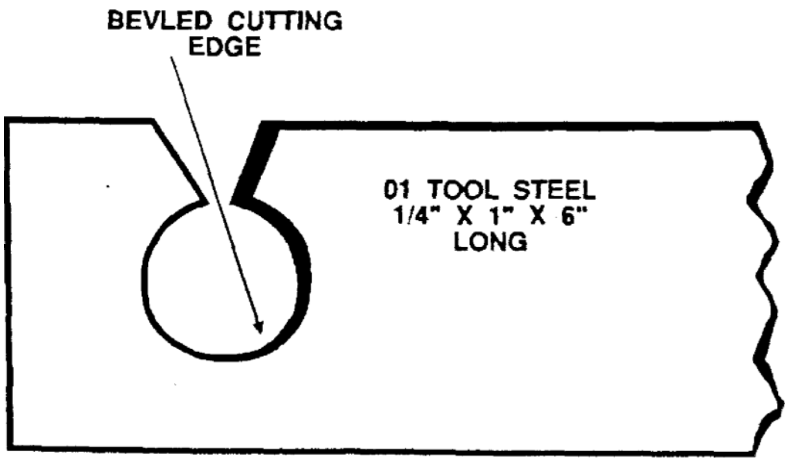
MAKING THE REAMER OR CUTTING TOOL BALL CUTTER

BALL CUTTER

To any one not experience with the process of making a sphere in metal it may seem a very difficult operation. It only requires a little knowledge and experience to make a ball cutter to fit any bore of gun. A rotating piece of pre-turned tool steel is passed through a properly-shaped circular aperture in a flat steel die. It is held with its upper or cutting in the same plane as the axial line of the body rotating.

We have a rifle of a certain bore to which is to be fitted a round ball. First, take a piece of steel about one-quarter of an inch thick, one inch wide and six or eight inches long. A piece of 01 tool steel will do the job for this cutter.

Drill a hole near one end, but a little less in size than the bullet to be made. With a taper half-round reamer cut out the hole until that side of it that is to be the upper or cutting edge is exactly the size of the bullet desired.



TOOL FOR MAKING ROUND BALL CUTTER

The advantage of using this taper half-round reamer is the hole is made perfectly round. At the same time the taper of the reamer gives a bevel to the hole that forms a good strong and effective cutting edge. With a file cut out a portion of the tool of a V shape, beveling the edges the same as the hole.

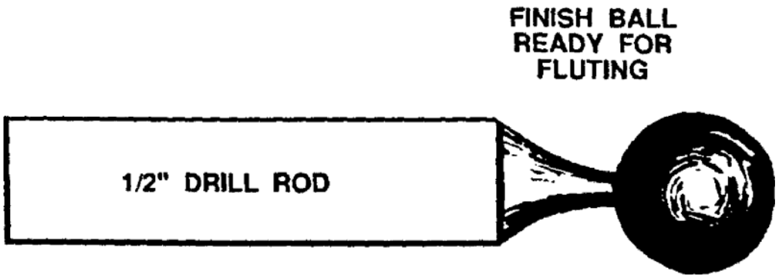
The V may be either cut on the end, a side of the tool, but in use it is best to cut on the side. When done, harden and temper for use. For the cutter, turn a piece of steel in the lathe to fit a chuck. Half-inch round stock is perhaps the best size of steel from which to make cutters of less than half-inch, and the length about six inches.

The end on which the cutter is to be made is roughly turned into a ball, leaving the end where the center supports it to be removed by the lathe tool or by filing.

MAKING THE BALL CUTTER

When fitted so that the rough blank will be held firmly in the chuck, run the lathe at moderate speed. Set the T rest so that the steel die can be held on it about level with the underside of the rotating blank. On the rest lay the die, and press the opening to receive the rough turned sphere, applying oil and not pressing too hard.

Let it gradually, scrape its way through the circular aperture, the V- shaped opening in the side receiving the stem to which the cutter is attached. If it be preferred the die may be held in the hands and not supported on the T rest. Take care to supply plenty of oil to the work, as this will prevent it from scratching or tearing the cherry while being formed. Make two of these holes, one at each end of the piece of steel, roughing the blank with the first. The second which is a little larger than the one used to finish the cherry of the exact size.



BALL CUTTER READY FOR FLUTING

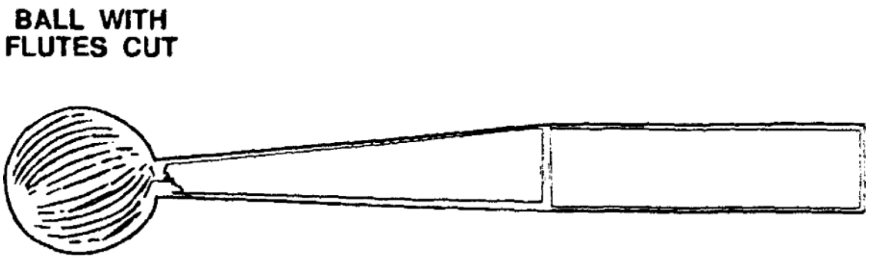
When this tool gets dull grind on the upper or cutting edge, but not too much, as it will enlarge the hole, and the result will be to make a larger cherry.

To form these blanks into cutting tools, the flutes will need to be cut mostly by hand. Keep in mind to leave the grooves deep enough to receive the metal cut from the blank mold when in operation.

The cutters do not have their cutting edges terminate in that portion of the mold, but usually on one side. This is done to insure a perfectly spherical form by having a cutting side operating at the bottom of the hole while it is being formed. It is difficult to make this form of cutting edge, and patience and care must be exercised or the blank may be spoiled.

In using a three-square file, to get a fine-cutting edge that will make a sharp V cut, grind away the teeth of one side, or get a good set of die maker's files. This will remove the slightly rounded or blunt edge as usually made on this form of file. By grinding two acute cutting angles can be had from one file. If they get a little dull on the sharp cutting comer, a little grinding will restore the edge and make it sharp again.

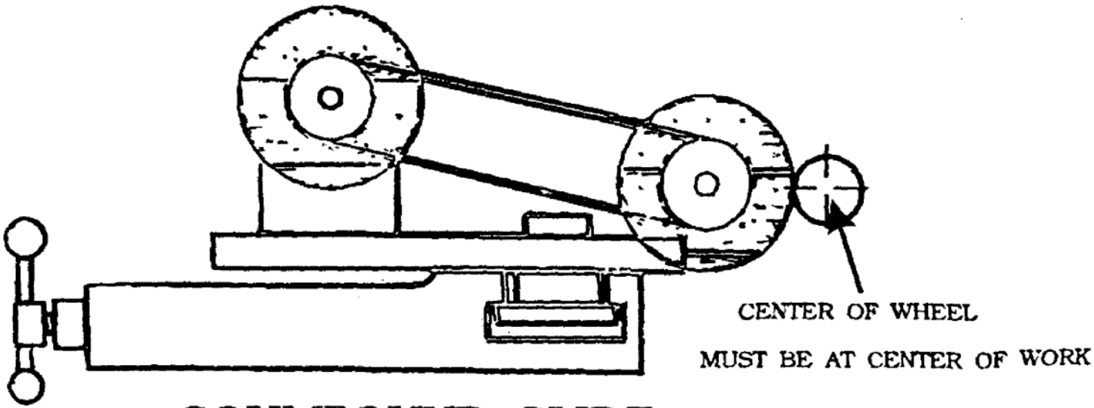
The ball cutter can be made to the exact size ready for use after heat treating. To do this, coat the complete tool with a descaling compound and then heat treat. After quenching, the compound can be scrubbed off, and it is ready for use.



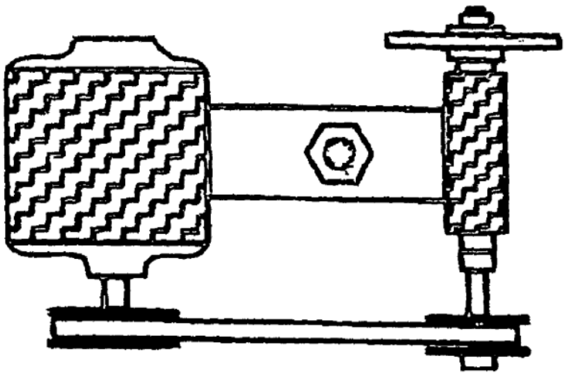
BALL CUTTER READY FOR USE

MAKING THE SIZING REAMER

The sizing reamer is no different in grinding than the standard reamer, except there are more angles, etc. First, look at the following drawings to see how the to turn the tool steel blank to shape. The blank does not have to have the angles turned on as shown in the drawings, this will be done during the , grinding process.



COUMPOUND SLIDE



TOOL POST GRINDER

Shop Made Tool Post Grinder

TAPER TURNING

Two methods are used to turn a taper in a lathe. You can cut the small angle taper, using the offset tail stock, and it is the quickest. Knowing the length of the taper (usually 1″) of the work between centers, the amount of the tail-stock offset can be computed to give any taper per foot, or inch within the capacity of the offset. To cut these tapers you will need two one inch dial indicators. One is set up to read the longitudinal feed for one inch, the other is to read how much taper is cut on the blank on one inch of carriage travel.

On the cartridge dimensions chart the difference between "J" minus "I" will give the length of the taper area. This will have to be reduced to a one inch length so you can find the correct angle.

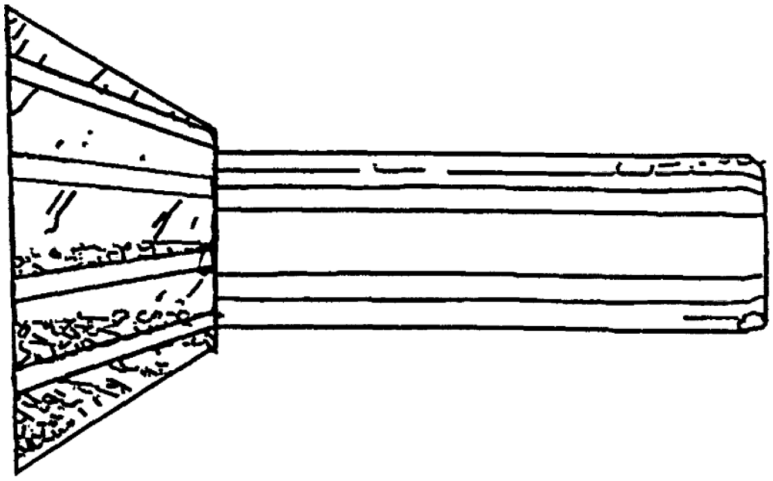
Once the correct taper is achieved, the taper can then be cut on the reamer blank. The correct taper can be calculated from the taper chart.

Formula is—

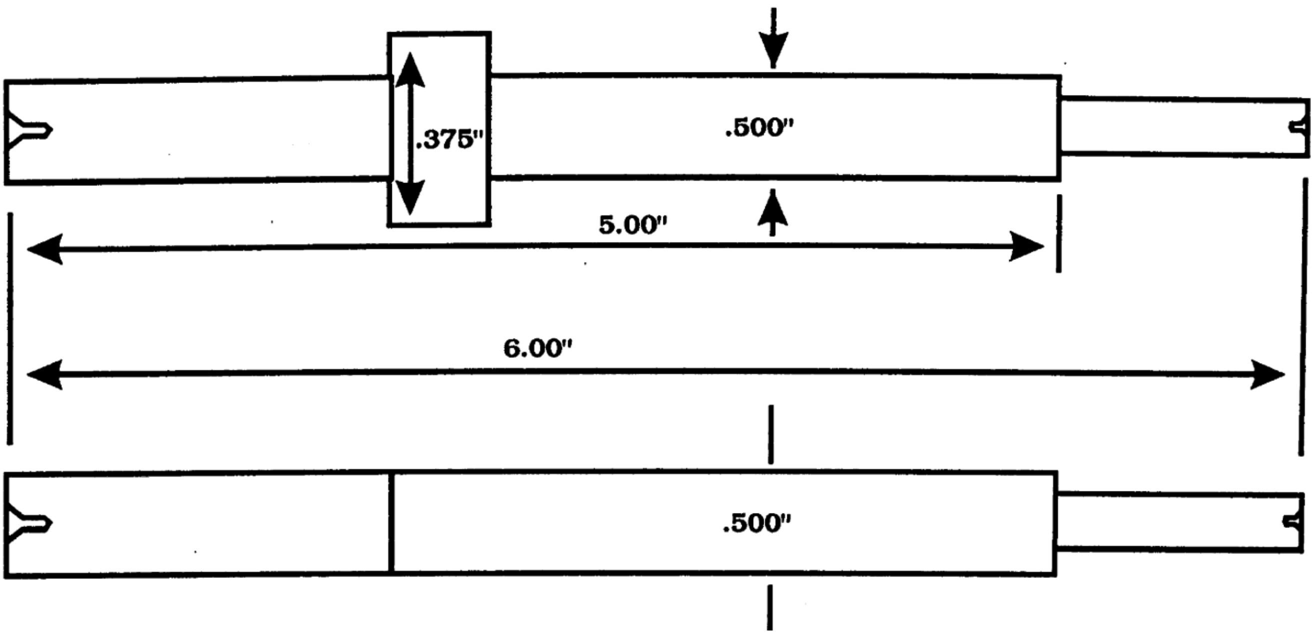
Difference between large and small end = x
Length between the above divided into x

The angle on the shoulders can be found on the same chart. To check to see if the correct angle is right, a large blank can be turned and the angle can be measured with the dial indicators.

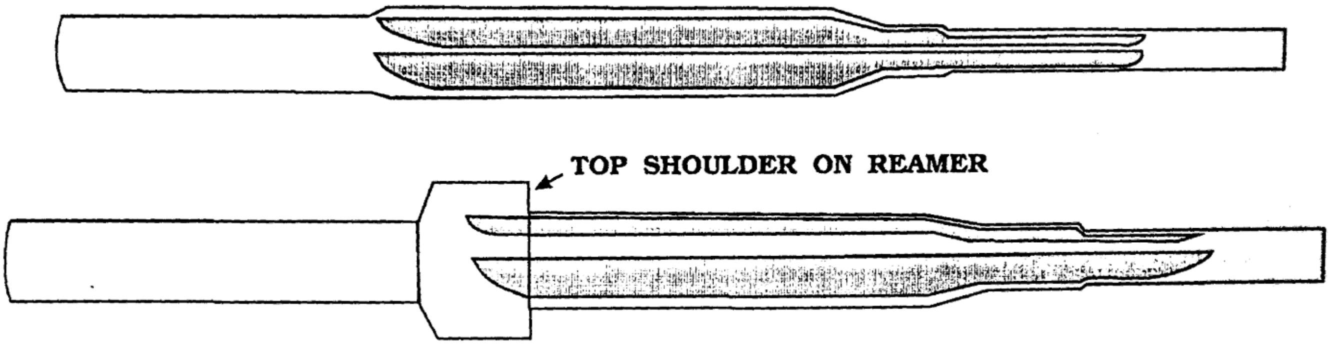
The centers are working out of line for this work and so are making a line contact, which may cause the tail center to heat. Watch it and correct the pressure and oil as necessary. As the lathe dog driving the work is operating at an angle to the head stock, the work will not turn uniformly* At one position the dog will be close to the driving plate, and as the driving leg of the dog is parallel to the center line of the work. The driving leg will be nearer the lathe center than when it is diametrically opposite.



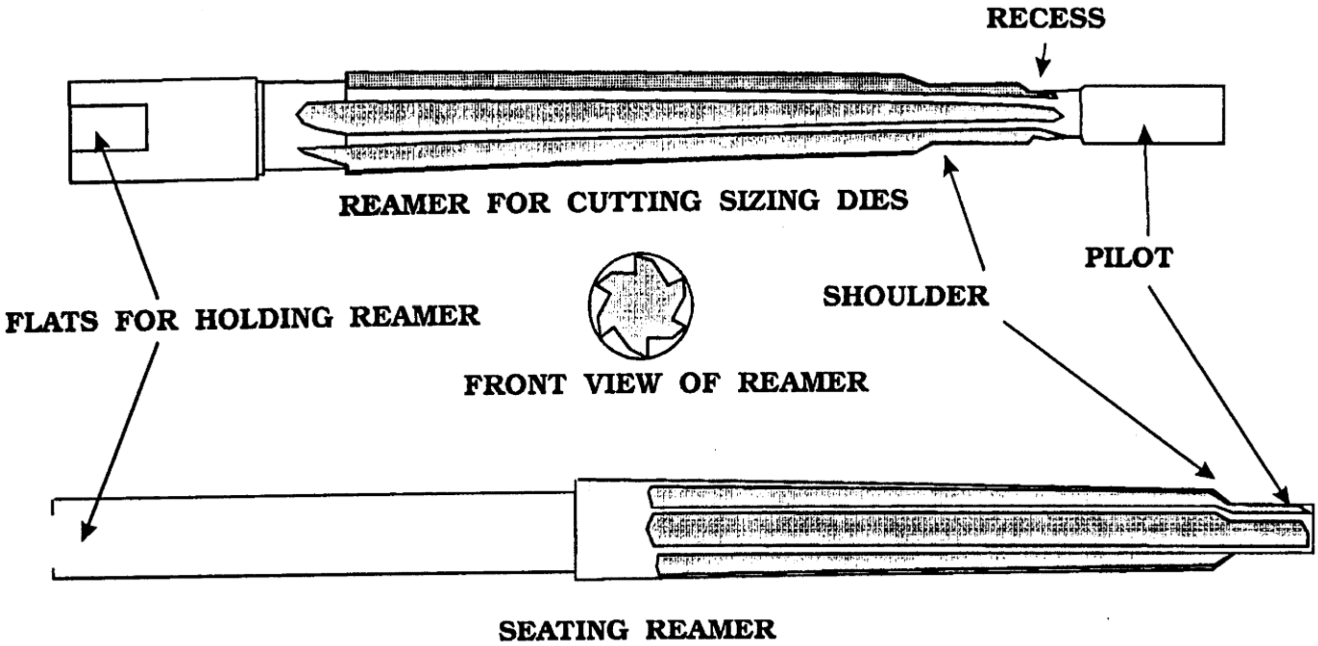
Angle Cutter



REAMER BLANKS AFTER TURNING READY FOR FLUTING



DIFFERENT TYPES OF REAMERS



SIZING AND SEATING REAMERS

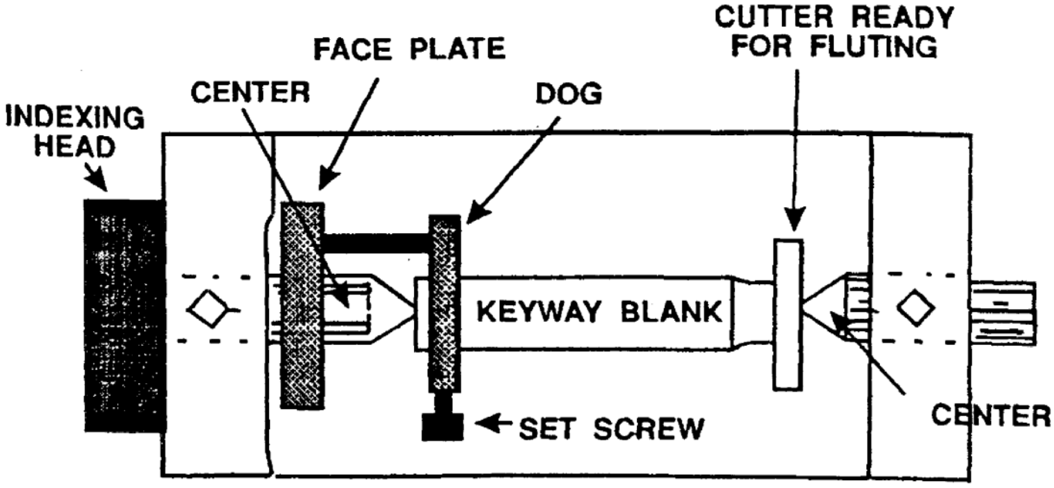
TAPER PER INCH FOR VARIOUS INCLUDED ANGLES						
The Tabulated Quantities = Twice the Tangent of Half the Angle						
Deg	0'	10'	20'	30'	40'	50'
0	0·00000	0·00290	0·00582	0·00872	0·01164	0·01454
1	0·001746	0·02036	0·02326	0·02618	0·02910	0·03200
2	0·03492	0·03782	0·04072	0·04364	0·04656	0·04946
3	0·05238	0·05528	0·05820	0·06110	0·06402	0·06692
4	0·06984	0·07276	0·07566	0·07858	0·08150	0·08440
5	0·08732	0·09024	0·09316	0·09606	0·09898	0·10190
6	0·10482	0·10774	0·11066	0·11356	0·11648	0·11940
7	0·12232	0·12524	0·12816	0·13108	0·13400	0·13694
8	0·13986	0·14278	0·14570	0·14862	0·15156	0·15448
9	0·15740	0·16034	0·16326	0·16618	0·16912	0·17204
10	0·17498	0·17790	0·18084	0·18378	0·18670	0·18964
11	0·19258	0·19552	0·19846	0·20138	0·20432	0·20726
12	0·21020	0·21314	0·21610	0·21904	0·22198	0·22492
13	0·22788	0·23082	0·23376	0·23672	0·23966	0·24262
14	0·24556	0 ·24852	0·25148	0·25444	0·25738	0·26034
15	0·26330	0·26626	0·26922	0·27218	O·27516	0·27812
16	0·28108	0·28404	0·28702	0·28998	0·29296	0·29502
1 7	0·29890	0·30188	0·30486	0·30782	0·31080	0·31378
18	0·31676	0·31976	0·32274	0·32572	0·32870	0·33170
19	0·33468	0·33768	0·34066	0·34366	0·34666	0·34966
20	0·35266	0·35566	0·35866	0·36166	0·36466	0·36768
21	0·37068	0·37368	0·37670	0·37972	0·38272	0·38574
22	0·38876	0·39178	0·39480	0·39782	0·40084	0·40388
23	0·40690	0·40994	0·41296	0·41600	0·41904	0·42208
24	0·42512	0·42816	0·43120	0·43424	0·43728	0·44034
25	0·44338	0·44644	0·44950	0·45256	0·45562	0·45868
26	0·46174	0·46480	0·46786	0·47094	0·47400	0·47708
27	0·48016	0·48324	0·48632	0·48940	0·49248	0·49556
28	0·49866	0·50174	0·50484	0·50794	0·51004	0·51414
29	0·51724	0·52034	0·52344	0·52656	0·52966	0·53278
30	0·53590	0·53902	0·54214	0·54526	0·54838	0·55152
31	0·55464	0·55778	0·56092	0·56406	0·56720	0·57034
32	0·57350	0·57664	0·57980	0·58294	0·58610	0·58926
33	0·59242	0·59560	0·59876	0·60194	0·60510	0·60828
34	0·61146	0·61464	0·61782	0·62102	0·62420	0·62740
35	0·63060	0·63380	0·63700	0·64020	0·64342	0·64662
36	0·64984	0·65306	0·65628	0·65950	0·66272	0·66596
37	0·66920	0·67242	0·67566	0·67890	0·68216	0·68540
38	0·68866	0·69192	0·69516	0·69844	0·70170	0·70496
39	0·70824	0·71152	0·71480	0·71808	0·72136	0·72464
40	0·72794	0·73124	0·73454	0·73784	0·74114	0·74446
41	0·74776	0·75108	0·75440	0·75774	0·76106	0·76440
42	0·76772	0·77106	0·77442	0·77776	0·78110	0·78446
43	0·78782	0·79118	0·79454	0·79792	0·80130	0·80468
44	0·80806	0·81144	0·81482	0·81822	0·82162	0·82502
45	0·82842	0·83184	0·83526	0·83866	0·84210	0·84552

TAPER PER INCH FOR VARIOUS INCLUDED ANGLES						
Deg	0'	10'	20'	30'	40'	50'
46	0·84894	0·85238	0·85582	0·85926	0·86272	0·86616
47	0·86962	0·87308	0·87656	0·88002	0·88350	0·88698
48	0·89046	0·89394	0·89744	0·90094	0·90444	0·90794
49	0·91146	0·91496	0·91848	0·92202	0·92554	0·92908
50	0·93262	0·93616	0·93970	0·94326	0·94682	0·95038
51	0·95396	0·95752	0·96110	0·96468	0·96828	0·97186
52	0·97546	0·97906	1·98268	0·98630	0·98930	0·99354
53	0·99716	1·00080	1·00444	1·00808	1·01174	1·01538
54	1·01906	1·02272	1·02638	1·03006	1·03376	1·03744
55	1·04114	1·04484	1·04854	1·05226	1·05596	1·05970
56	1·06342	1·06716	1·07090	1·07464	1·07840	1·08214
57	1·08592	1·08968	1·09346	1·09724	1·10102	1·10482
58	1·10862	1·11242	1·11624	1·12006	1·12388	1·12770
59	1·13154	1·13538	1·13924	1·14310	1·14696	1·15082
60	1·15470	1·15858	1·16248	1·16636	1·17026	1·17418
61	1·17810	1·18202	1·18594	1·18988	1·193S3	1·19776
62	1·20172	1·20568	1·20966	1·21362	1·21762	1·22160
63	1·22560	1·22960	1·23362	1·23764	1·24166	1·24570
64	1·24974	1·25378	1·25784	1·26190	1·26598	1·27006
65	1·27414	1·27824	1·28234,	1·28644	1·29056	1·29468
66	1·29882	1·30296	1·30710	1·31126	1·31542	1·31960
67	1·32378	1·32796	1·33216	1·33636	1·34056	1·34478
68	1·34902	1·35326	1·35750	1·36176	1·36602	1·37028
69	1·37456	1·37984	1·38314	1·38744	1·39176	1·39608
70	1·40042	1·40476	1·40910	1·41346	1·41782	1·42220
71	1·42658	1·43098	1 43538	1·43980	1·44422	1·44864
72	1·45308	1·45754	1·46200	1·46646	1·47094	1·47542
73	1·47992	1·48442	1·48894	1·49348	1·49800	1·50256
74	1·50710	1·51168	1·51624	1·52084	1·52544	1·53004
75	1·53466	1·53928	1·54392	1·54856	1·55322	1·55790
76	1·56258	1·56726	1·57196	1·57668	1·58140	1·58612
77	1·59088	1·59562	1·60040	1·60516	1·60996	1·61476
78	1·61956	1·62440	1·62922	1·63406	1·63892	1·64380
79	1·64868	1·65356	1·65846	1·66338	1·66830	1·67324
80	1·67820	1·68316	1·68814	1·69512	1·6981 2 [	1·70314
81	1·70816	1·71320	1·71824	1·72332	1·72836	1·73348
82	1·73858	1·74368	1·74882	1·75396	1·75910	1·76428
83	1·76946	1·77464	1·77984	1·78506	1·79030	1·79554
84	1·80080	1·80608	1·81138	1·81668	1·82198	1·82732
85	1·83266	1·83802	1·84340	1·84878	1·85418	1·85960
86	1·86504	1·87048	1·87594	1·88142	1·88690	1·89240
87	1·89792	1·90346	1·90902	1·91458	1·92016	1·92576
88	1·93138	1·93700	1·94266	1·94832	1·95400	1·95968
89	1·96540	1·97112	1·97686	1·98262	1·98840	1·99420

The lathe dog will travel farther when in the outer position than when nearer the center, and this will result is an irregular turning of the work. The lead screw is turning uniformly with the head stock and advances the cutting tool at a uniform rate. The turning of the work is regular, making the amount of metal removed by the tool more on one side of the work than on the opposite side. The finished cuts must be very light to prevent the springing of the work and to assure a true circular cut.

For short tapers with medium to large angle, the compound slide can be set to give the required taper. The feed will be by hand, as there is no power feed on the compound slide. The compound is graduated in degrees, so that the taper per foot must be converted to degrees to be set directly by the graduations. Tapers longer than the feed of the compound cannot be cut at one setting, although it is possible by careful work to move the carriage to a new position and cut an extension of the taper. Never move the work until the taper is completed.

Indexing Fixture Set For Cutting Woodruff Cutter

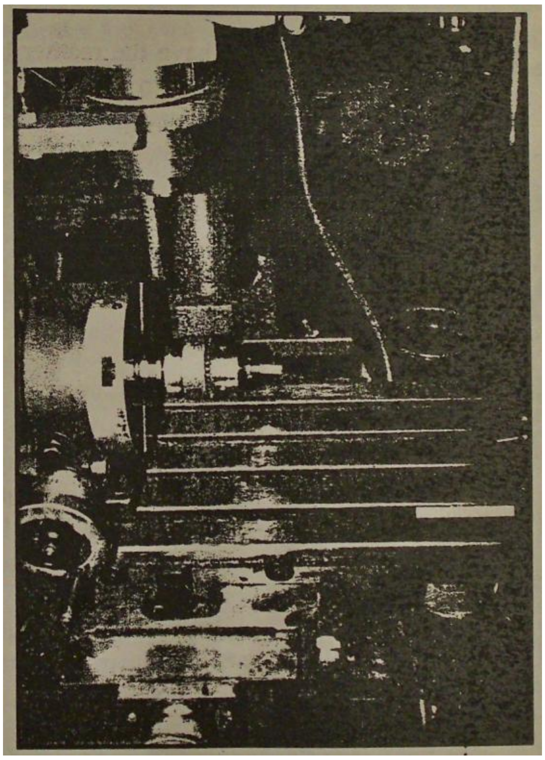


INDEXING FIXTURE

The Taper Charts gives the half-angles for changing a given taper in inches per foot to degrees.

Leave all diameters about ·020 larger than shown. The tool steel blank should be cut 6½ to 7 inches long. Use the smallest center drills for center drilling the blank. When turning the shoulder can be left square, but there should be a recess cut in front of the shoulder, end of neck, etc. for clearance when grinding.

I will go through all the steps of making reamers, from the start of reamer blanks from a piece of tool steel, to the finish reamer, ready to be used. To start you will need a vertical milling machine, and an indexing fixture. These two items are the two main pieces of equipment necessary to make cutting tools.



Indexer

KEYWAY CUTTER

Use the reamer blank that you turned in the previous chapter to make the keyway cutter. When you finish you will have a good cutter when completed. Once you have made this cutter, you will be able to make all the different types of cutters and tools that you will need in the shop.

Once you have the reamer blank completed, go to the milling machine and set up the indexing head. This should have centers also with some way to attach a small lathe dog to hold the reamer blank solid. If the indexing head supports a collet, the stock can be held with a collet.

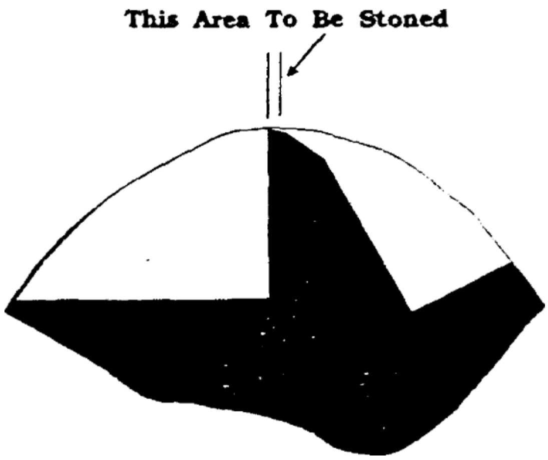
ANGULAR CUTTER

Next chuck up a 60 degree angular cutter in the mill. The diameter of the cutter should be at least 1½ inch. Slow down the mill to about 100 to 150 rpm as the tool steel tends to get hot.

Measure the blank on the flute end, which in this case has been turned so it is ·760. Bring down the cutter on the milling head while it is running and just touch the drill rod and stop.

Half of that size is ·380, we do not want to cut the flutes half way as the reamer would probably chatter. The first flute should be cut ·002 in front of center, the next should be cut ·004 ahead of center, and the third should be cut ·006 ahead of center for the total of 12 flutes.

After the third flute is cut, the fourth should start back at ·002, then ·004 and the finish cut is ·006 ahead of center.



STONING THE REAMER EDGE
Stoning Finish Cutter

THE FLUTE THICKNESS

The flute’s wall should have a thickness of about ·060 to ·080. This would of course be thinner on the smaller reamers and thicker on the larger reamers. You do not want to make flute walls to thin as they tend to break if the reamer gets to dull, and when this happens the barrel will more than likely be ruined.

DEPTH OF FLUTES

You want to use an ample supply of coolant when cutting the flutes. Depending on the reamer size you will go in about 3/4 of the depth on the first cut and then finish to the correct depth the second pass. Watch for bowing as you cut the flutes. If it is bowing you are either taking too big a cut, or the cutter is dull, or you may be cutting to fast.

Whatever the reason you do not want this to happen as it is putting much internal stress in the reamer blank. When heat treating the blank will probably warp bad. If you fill there is stress in the reamer blank, I would suggest that when you get ready to heat treat the reamer, that you put the reamer in the furnace when you turn it on.

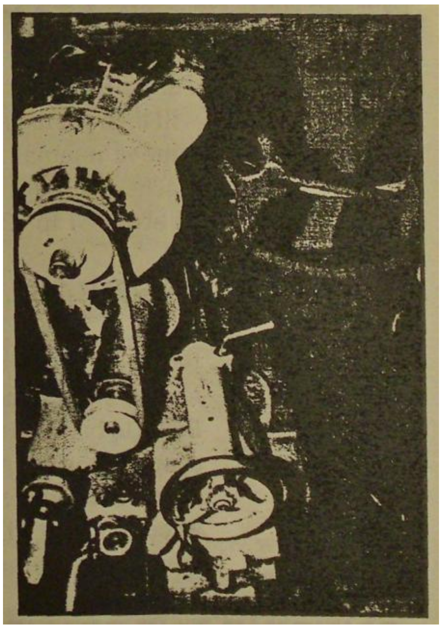
Bring the temperature up to 1000 degrees, and let it set for 30 to 40 minutes. Remove and bury it in lime until cool, or turn off the furnace and let it cool over night.

HEAT TREATING THE REAMER

Bring the temperature of the furnace up to the temperature that is recommended by the maker of the tool steel. Coat the reamer blank with some decarbonizing powder, put the reamer blanks in the furnace and let set for 10 minutes. Remove and quench in the oil tank, or what other quenching medium the manufacture recommends.

When you quench the blank make sure that you go straight in the quenching tank, if you quench the reamer at an angle, you will warp it. If you do warp a reamer you must bring the reamer up to 1500 degrees and let it cool in the oven, then straighten it when cool.

When you have the reamer quenched, lay them down on something where they won't roll off. They are very hard and brittle, and if they fell on the cement floor, they would probably break.



GRINDING A TOOL

TEMPERING THE REAMERS

Turn off the furnace, close it up and let it cool down to 350 degrees. We will then put the hardened reamers into the furnace to draw the hardness, and remove internal stress. Leave the blanks in the furnace until the furnace reaches 100 degrees, or better yet leave them in overnight. The reamers will be about 61 to 62 Rc in hardness.

TEMPERING THE BALL CUTTER

In tempering the cherry do not leave it too hard, and be careful not to temper too hard above the spherical portion. Leave the shank softer, as it will be less liable to break. It is not necessary to shape the shank to exactly conform to the shape of the V opening in the forming tool. The edges of this opening have been beveled the same as the opening that shapes the cherry. They will act as cutters, and will produce the shank to conform to the V. It is well to mark the cutters in some place on the shank, with their size in hundredths of an inch.

GRINDING THE REAMER

The toolpost grinder is what we will use to grind the tool. It has an electric motor of about 1/4 H. P. and a ballbearing spindle on which grinding wheels can be mounted. Mount the grinder on the compound slide of the tool post. If you make the grinder, or buy one the spindle and motor should be equipped to take pulleys of various sizes. The speed of the grinding wheel can then be adjusted according to its size.

The spindle should turn in a counterclockwise direction, as you face the end on which the grinding wheel is mounted. This will cause the sparks to be directed downward, so that the grindings can be caught on a paper on the lathe ways. The motor is mounted so the belt tension can be adjusted for each set of pulleys used. The motor should run without any vibration, and the bearings of the spindle should be a sealed, and permanently greased type that will run smoothly for a long time before needing adjustment.

Each time a wheel is mounted on the spindles, it must be trued up while running at full speed. To do this, hold a diamond tool rigidly against the wheel, taking light cuts with the diamond until the wheel has an even cutting face over all.

The face of the wheel will show when it is true, since the color will be darker where the diamond has not cut away any low spot. I prefer to hold the diamond in the tail stock chuck or the headstock three-jaw chuck and then feed the wheel across its point. If you use the tailstock chuck, the tailstock ram must be locked in position, to stop any looseness that will let the diamond vibrate.

If it is loose it will not cut a smooth surface on the wheel. The point of the diamond must be at the exact center of the grinder and should be turned frequently to equalize the wear. When it is gripped in the chuck, the point will be slanting downward at a 5° angle and sideways at a 30° angle. This will prevent vibration of the diamond and will make the wear on the diamond tend to keep it sharp.

The rate of feed of the diamond across the face of the wheel will govern the wheel's cutting characteristics. A quick feed will leave a more open grain structure to the wheel, so that it will cut, more quickly for roughing but not smoothly enough for finishing. For finishing, a very slow feed with small infeed is used. This will leave all the grains cut to a very smooth surface and it will not be as sharp but will cut very smoothly. The smooth surface will become glazed very quickly if heavy cuts are taken with it.

The reamer blanks are now ready to be ground to size. We will grind the flutes first. Set up the tool post grinder on the lathe. Get everything lined up and put the small lathe dog on the reamer blank, on the shank end and grind the flutes end first.

Clean out the centers on the blank, and set between the centers. Cover the bed of the lathe up to keep the grinding dust off the ways. The sparks and particles removed from the work will be directed down onto the bed of the machine. This material is very abrasive and, if it gets under the slides of the saddle or cross feed, it will wear them out fast. The slides should be covered to protect them.

Paper towels or old newspapers fitted under the work and weighted down so that they will remain in place will do a fine job of keeping the area clean. Since the movement of the slides will disturb the paper covering, watch it and adjust it to any movement as you work. Also put a small dish under the grinding wheel to catch the sparks. Fill this dish with oil or water, and the sparks will collect in it and not bounce around.

Set the lathe in back gear drive, turn on the lathe so it will run in reverse. Then turn on the tool post grinder, and starting at the tail stock end, touch the grinder to the blank.

The work should revolve slowly in the opposite direction so that of the grinding wheel, and the feed should be fairly fast, with light cuts. The grinding will generate much heat, which will make the work expand lengthwise, putting increased pressure on the tail center. The center must be kept well lubricated and should be adjusted frequently to prevent damage to the work or center, so work slowly and let it cool often.

Move the grinder past the blank, set in about ·005 and engage the feed. It will not clean up completely, but it will start to clean up any part that is warped. Make one pass and if it is cleaned up enough so you can get a measurement, check the size of both ends. If you are making bore reamers there should be about ·002 taper from the shank end to the end of the blank. On this cutter there should be no taper.

The tail stock end should be centered on the cutter end. We will allow the ·0005 to hone into size. Turn the lathe on in reverse, and the grinder and start feeding the wheel by hand on the cutting edge of the reamer. Take light passes not much over ·005 so you will get a good finish.

SIZING OR SEATING REAMER

When you first start grinding, grind the pilot first. The pilot should be ground about ·001 smaller than the bore diameter. All of these sizes are shown on sizing or seating die dimensions. After grinding the pilot to size, grind the angle shown for the end of the neck of the cartridge, going slightly deeper to give a recess.

The angle can be set using the compound on the carriage. From there grind the neck of the case. The neck can be ground slightly shorter than shown. The reason for this is that when you grind the shoulder, the neck then can be set to the proper length by taking the shoulder back a little at a time.

The shoulder angle is next, and it IS VERY IMPORTANT TO GET RIGHT as this is where SET THE DEPTH ON on rimless cases. This can be set from the taper and angle charts. If you fill that you can set it properly from the angle setting set on the compound of the lathe do so.

Grind now the taper on the body of the case. This is also set from the angle and taper charts. This is important to have this right so cartridges will fit in other guns of the same caliber.

GRINDING THE SHANK

When you have completed the cutter end of the cutter, take the blank from the lathe. Turn it around, put the dog on the other end and put it back in the lathe. Before doing this, make sure you bring the tail stock back to center.

Square the grinding wheel up, it is time to grind the shank to the correct size. Do not take over ·003 per side as it will heat the metal too much. Go slow and easy. The shank on the cutter will be ground the same way, and the size of the shank will be ·500.

GRINDING THE CLEARANCE

Now comes the time to relieve or grind the clearance on the back of the flutes for clearance. If you do not have indexing on your lathe, it must be ground by hand. If you have indexing on the lathe, the tool post grinder will grind this clearance. To grind the relief by hand you will need a small hand grinder. Coat the reamer flutes with a lay out fluid. This will darken the metal so you can see how close you are getting to the cutting edge.

Grind the relief just back of the cutting edge, and up to within ·005 to ·010 of the cutting edge. It won't take much to give the necessary relief, as all you need is clearance so the reamer won't rub.

GRINDING THE CUTTING EDGES

All the edges of the reamer has to be sharpened so the reamer will cut freely. After making one or two reamers, you will understand fully how to do this.

STONING THE CUTTING EDGE

Once the face has been honed, it will be necessary to hone the flutes. Hone right up to the edge watching the lay out fluid coating. This will tell when you have gotten it honed tight.

Start from the area where you ground the relief, and slowly go up to the cutting edge. It should feel sharp when completed.

A new reamer or cutter has more tendency to chatter than one that has been . used for some time. Chatter may often be reduced by reducing the speed, also sometimes increasing the feed will eliminate chatter. If for some reason there should be to little clearance the reamer will not cut freely, as the lands or margin will rub instead of cut against the walls.

Sizing and seating dies are made the same and the charts will give you all the dimensions for the different cartridges. Bullet mold cutters are made and ground the same way as the above reamers.

CHAPTER 8 CARTRIDGE DIMENSIONS

It is not difficult to make the reamers that are used for cutting the reamers. You will have to make a few to get proficient at making them.

If you have cartridges that you need loading dies for, but are not listed in the following pages, don't despair. It is a simple matter to get the dimensions that you need, then follow the instructions given to make them.

To make dies not listed, you will need a new unfired case to get the dimensions. First, measure the case ·050 in front of the extractor, and the shoulder to get the different in sizes. The taper charts will give you the correct angle (Doubled). Keep this dimension for later use.

With a caliper, measure the length from the shoulder to the point in front of the shoulder where you took the first measurement. Also, get a measurement from the front of the shoulder next to the neck.

You can also set this angle with the compound slide and a dial indicator. To do this, put a center in the headstock spindle and tailstock. Check to see if the tailstock is setting ·000 before measuring this angle.

FINDING THE ANGLE

Set up a case between centers, with the neck of the case in the tailstock center. Set the dial indicator against the case body, give the compound slide a slight angle. With the crank on the compound slide and with the carriage locked in place, run the dial indicator along the case body. Reset the angle and repeat the operation until the indicator remains at 0″.

Once this angle is obtained, the body for the reamer can be cut. Do the same thing for the shoulder to get the correct angle.

All dimensions for the sizing die should be about ·005- ·008″ smaller for the sizing die, and about ·010″ larger for the seating die.

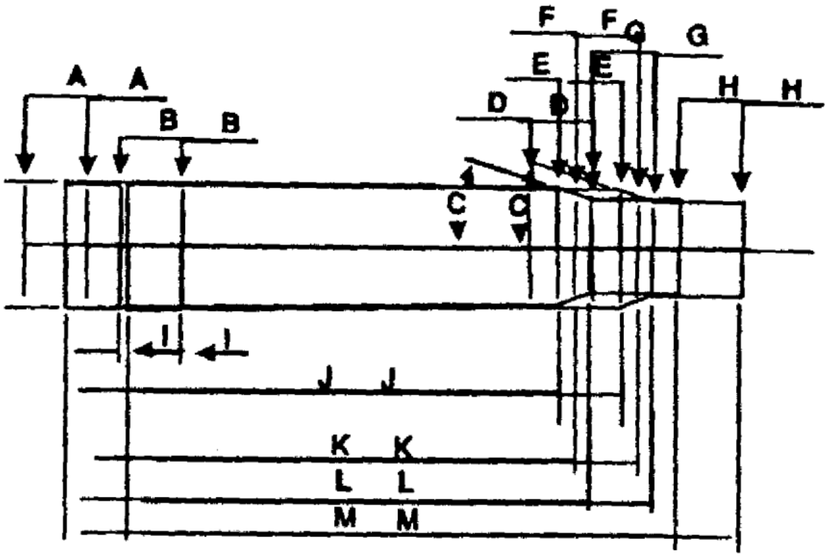
In making the reamers, the dimension "E" is not really important, as it is used primarily for chambering. To find the "J" dimension, subtract "I" from "J" for the shoulder length, as this is the proper point for measuring.

LOADING DIES FOR A PRESS

If you are making loading dies for a press, you can cut a stop shoulder ·130 from the back of the case. Using the ·200″, subtract ·130″ from ·200 to get ·070″ for the shoulder. This will guarantee that you do not cut the die too deep.

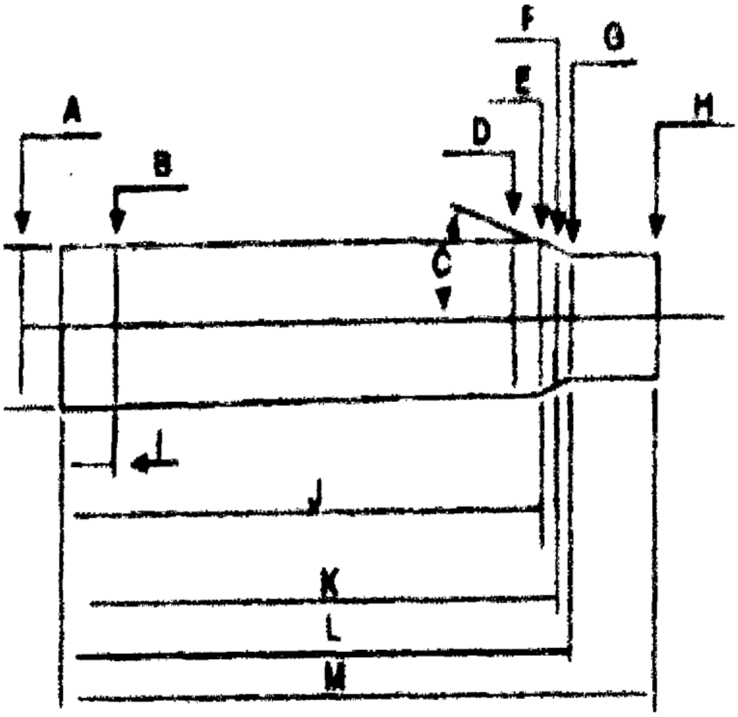
If you are making a sizing or seating die that you will run the case in fall length, cut a shoulder at the end of "J" for a stop.

As you see it will not be difficult to make these reamers, just a little time. Once you have the reamers, they will make hundreds of dies before having to be replaced.



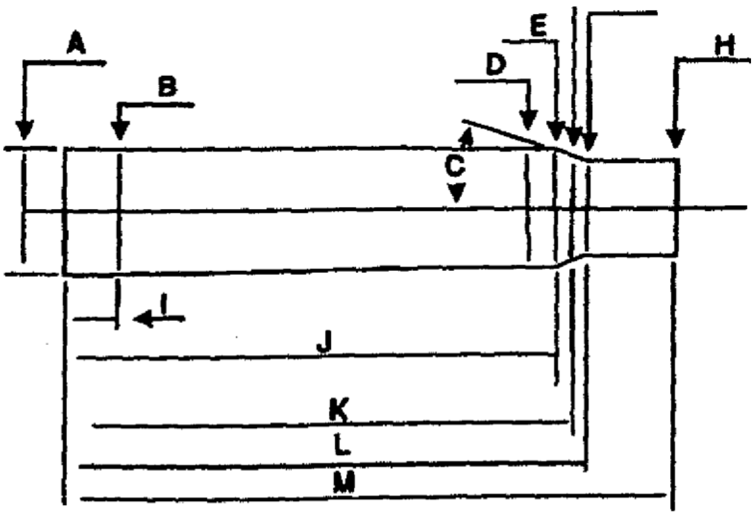
DIMENSIONS FOR RIFLE SEATING DIE

DIMENSIONS FOR RIFLE SEATING DIES													
	A	B	C	D	E	F	G	H	I	J	K	L	M
17 Rem.	·380	·377	23°	·359	·3568	·330	·201	·200	·200	1·346	1·378	1·530	1·808
218 Bee	·418	·347	15°	·347	·333	·287	·244	·243	·200	·932	1·017	1·097	1·355
22 Hornet	·360	·296	5° 29'	·279	·278	·262	·245	·243	·200	·844	·930	1·017	1·408
22-250 Rem.	·476	·468	28°	·418	·415	·347	·257	·255	·200	1·510	1·574	1·659	1·924
220 Swift	·483	·445	21°	·406	·403	·335	·262	·261	·200	1·510	1·574	1·659	1·924
222 Rem.	·380	·377	23°	·359	·358	·330	·253	·253	·200	1·264	1·303	1·387	1·700
222 Rem. Mag.	·379	·376	23°	·3589	·358	·330	·255	·254	·200	1·459	1·502	1·580	1·862
223 Rem.	·380	·3769	23°	·3594	·355	·330	·255	·254	·200	1·433	1·473	1·552	1·772



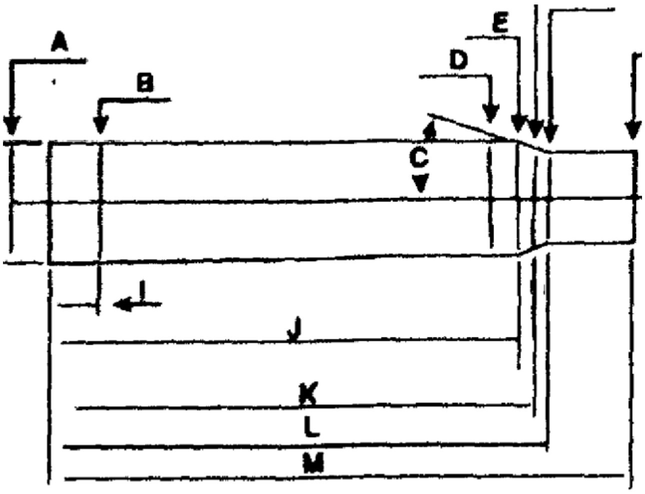
DIMENSIONS FOR RIMLESS SEATING DIE

DIMENSIONS FOR RIFLE SEATING DIES													
	A	B	C	D	E	F	G	H	I	J	K	L	M
225 Win.	·483	·423	25°	·408	·407	·350	·262	·261	·200	1·524	1·596	1·680	1·940
6mm Rem.	·478	·472	26°	·433	·4317	·375	·278	·277	·200	1·718	1·786	1·876	2·253
243 Win.	·474	·471	20°	·456	·455	·400	·278	·277	·200	1·554	1·640	1·797	2·055
25-06 Rem.	·474	·4708	17° 30'	·440	·442	·375	·292	·291	·200	1·940	2·058	2·182	2·502
25-35 Win.	·516	·4168	11° 34'	·370	·364	·325	·289	·282	·200	1·3S5	1·482	1·569	2·048
250 Savage	·478	·469	26° 30'	·420	·415	·347	·289	·286	·200	1·510	1·589	1·636	1·922
257 Roberts	·477	·472	20° 39'	·432	·431	·375	·292	·291	·200	1·718	1·803	1·903	2·253
7mn-08 Rem.	·473	·471	20°	·456	·455	·400	·317	·316	·200	1·554	1·640	1·744	2·045



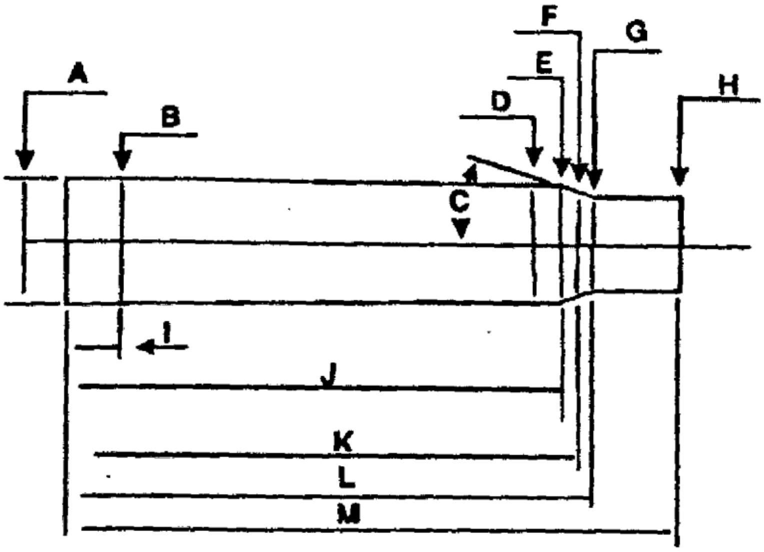
DIMENSIONS FOR RIMLESS SEATING DIE

DIMENSIONS FOR RIFLE SEATING DIES													
	A	B	C	D	E	F	G	H	I	J	K	L	M
7 mm Express	·474	·471	17° 15'	·4427	·442	·375	·315	·315	·200	1·992	2·110	2·924	2·560
7mm Mauser	·477	·472	20° 39'	·433	·431	·375	·325	·3217	·200	1·720	1·804	2·255	2·312
280 Rem.	·474	·471	17° 15'	·4427	·442	·375	·317	·315	·200	1·992	2·110	2·192	2·560
284 Win.	·504	·501	35°	·478	·476	·420	·323	·322	·200	1·770	1·820	1·879	2·180
30 Carbine	·370	·357				·346		·338	·200				1·300
30Rem.	·425	·422	23°	·403	·4025	·375	·333	·332	·200	1·496	1·538	1·577	2·055
30-06	·474	·470	17° 15'	·440	·442	·375	·339	·339	·200	1·939	2·058	2·101	2·502
30-30 Win.	·516	·421	15° 39'	·404	·403	·375	·333	·331	·200	1·454	1·504	1·578	2·083



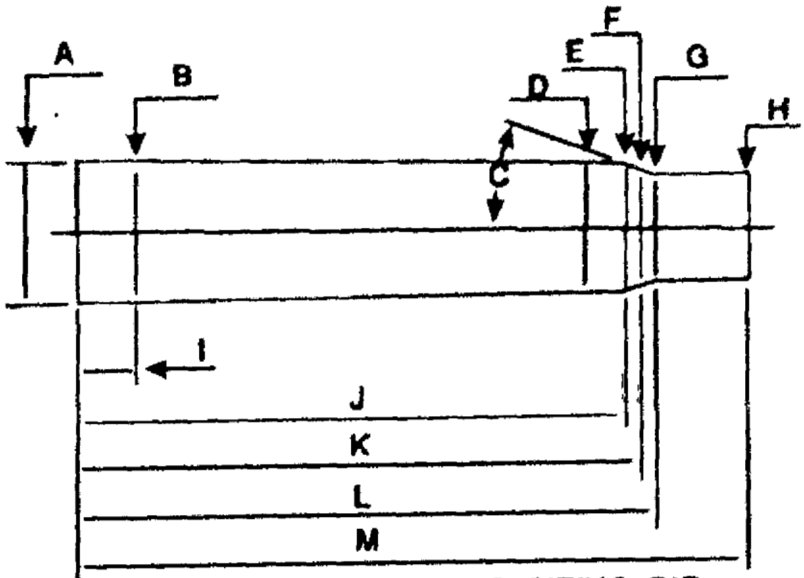
DIMENSIONS FOR RIMLESS SEATING DIE

DIMENSIONS FOR RIFLE SEATING DIES													
	A	B	C	D	E	F	G	H	I	J	K	L	M
30-40 Krag	·555	·459	21° 6'	·423	·420	·375	·338	·338	·200	1·728	1·787	1·832	2·322
300 Savage	·475	·471	30°	·447	·4473	·396	·342	·340	·200	1·558	1·611	1·648	1·899
·308 Win.	·474	·471	20°	·456	·455	·400	·346	·344	·200	1·554	1·640	1·703	2·025
303 British	·462	·457	20° 16'	·416	·403	·375	·345	·341	·200	1·812	1·851	1·892	2·222
303 Savage	·515	·443	16°	·421	·419	·375	·344	·336	·200	1·351	1·428	1·482	2·073
32 Rem.	·425	·422	23°	·403	·402	·375	·347	·344	·200	1·496	1·538	1·561	2·055
32 Win. Special	·516	·421	14° 44'	·405	·403	·375	·346	·343	·200	1·449	1·502	1·556	2·083
8mm Mauser	·475	471	19°	·436	·433	·392	·358	·356	·200	1·814	1·884	1·923	2·247



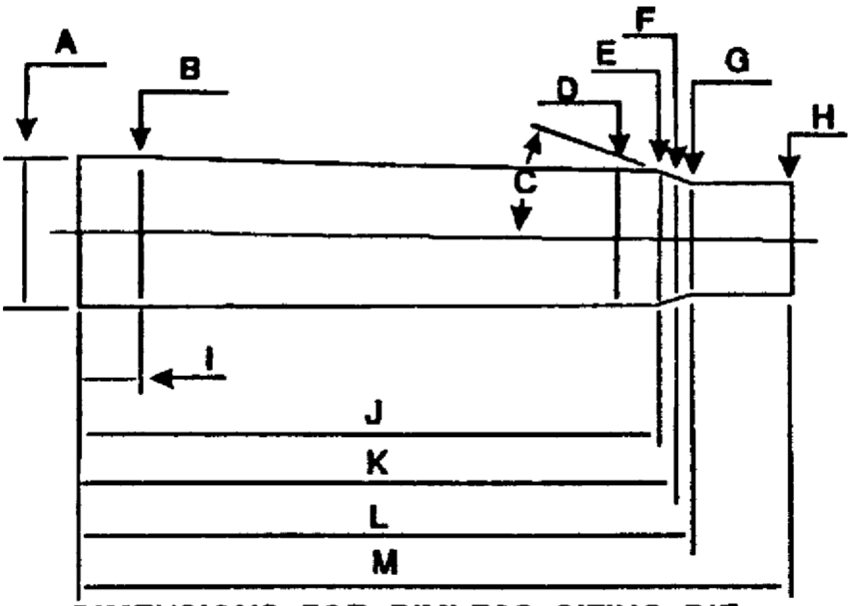
DIMENSIONS FOR RIMLESS SEATING DIE

DIMENSIONS FOR RIFLE SEATING DIES													
	A	B	C	D	E	F	G	H	I	J	K	L	M
348 Win.	·620	·548	19° 10'	·488	·486	·434	·379	·376	·200	1·660	1·735	1·814	2·265
35 Rem.	·463	·459	23° 25'	·429	·427	·405	·387	·383	·200	1·532	1·568	1·578	1·924
351 Win.	·420	·387		·383				·382	·200				1·404
358 Win.	·474	·471	20°	·456	·455	·420	·391	·389	·200	1·554	1·612	1·642	2·025
375 Win.	·516	·421		·401				·401					2·080
38-55 Win.	·516	·420		·395				·393					2·118
444 Marlin	·524	·470		·456				·454					2·240



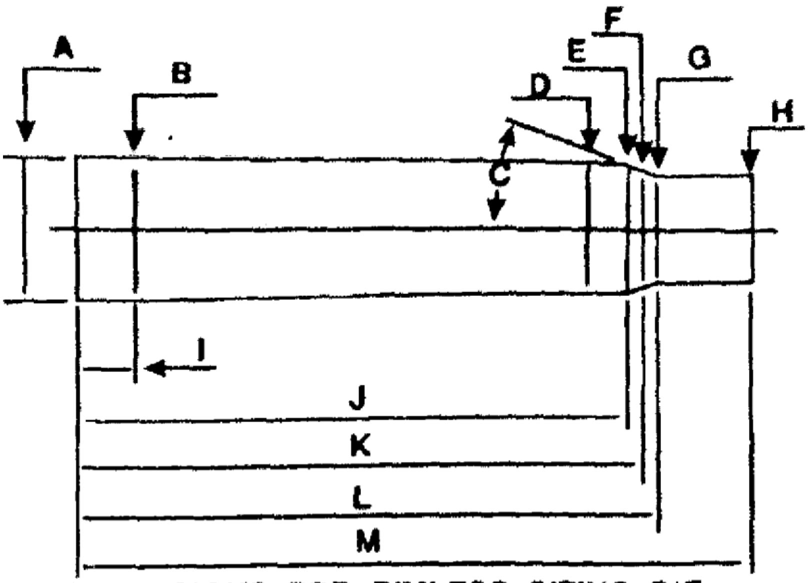
DIMENSIONS FOR RIMLESS SIZING DIE

DIMENSIONS FOR RIFLE SIZING DIES													
	A	B	C	D	E	F	G	H	I	J	K	L	M
17 Rem.	·378	·3709	23°	·353	·3508	·325	·194	·194	·200	1·351	1·381	1·535	1·796
218 Bee	·408	·3425	15°	·329	·328	·283	·239	·238	·200	·9233	1·007	1·089	1·345
22 Hornet	·350	·291	5° 38'	·274	·273	·258	·2408	·238	·200	·853	·930	1·017	1·403
22-250	·473	·4628	28°	·413	·410	·343	·252	·250	·200	1·515	1·578	1·663	1·912
220 Swift	·473	·4409	21°	·4015	·398	·331	·257	·256	·200	1·722	1·810	1·905	2·205
222 Rem	·378	·371	23°	·354	·353	·325	·249	·249	·200	1·264	1·296	1·387	1·700
222 Rem Mag.	·378	·371	23°	·354	·353	·225	·249	·249	·200	1·463	1·495	1·578	1·850
223 Rem.	·378	·3719	23'	·354	·350	·225	·249	·249	·200	1·438	1·466	1·557	1·760



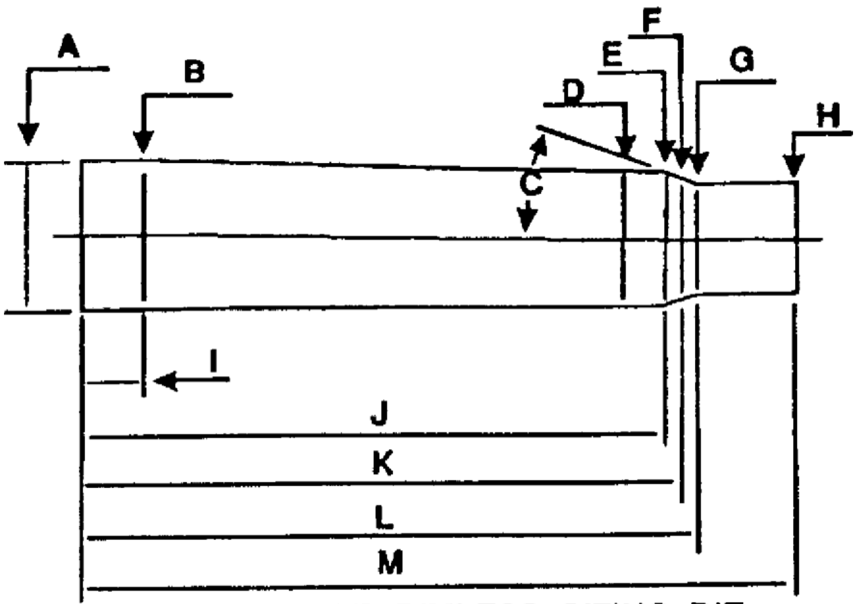
DIMENSIONS FOR RIMLESS SIZING DIE

DIMENSIONS FOR RIFLE SIZING DIES													
	A	B	C	D	E	F	G	H	I	J	K	L	M
225 Win.	·473	·418	25°	·4028	·402	·346	·255	·255	·200	1·530	1·590	1·686	1·930
6 mm Rem.	·473	·467	26°	·4258	·425	·371	·272	·272	·200	1·724	1·780	1·882	2·233
243 Win.	·473	·466	20°	·451	·450	·396	·271	·271	·200	1·599	1·634	1·804	2·045
25-35 Win.	·506	·412	11 34'	·364	·360	·320	·283	·280	·200	1·380	1·475	1·565	2·043
250 Savage	·473	·4628	26° 30'	·4147	·410	·343	·284	·281	·200	1·512	1·579	1·637	1·912
257 Roberts	·473	·467	20° 45'	·427	·425	·371	·285	·285	·200	1·727	1·799	1·911	2·233
25-06 Rem.	·473	·464	17° 30'	·450	·499	·396	·271	·271	·200	1·559	1·634	1·804	2·045
270 Win.	·473	·465	17° 30'	·437	·436	·371	·305	·303	·200	1·948	2·052	2·155	2·540



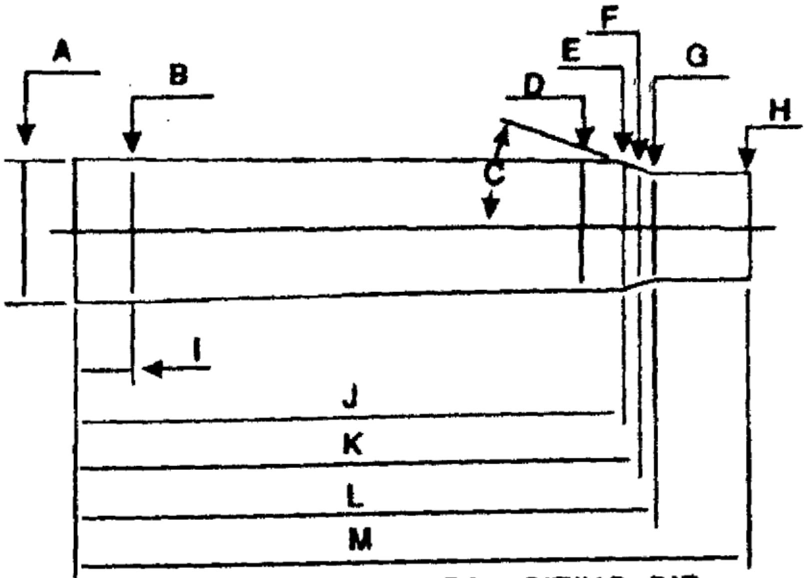
DIMENSIONS FOR RIMLESS SIZING DIE

DIMENSIONS FOR RIFLE SIZING DIES													
	A	B	C	D	E	F	G	H	I	J	K	L	M
7mm Mauser	·473	·466	20° 45'	·426	·425	·370	·319	·315	·200	1·794	1·901	2·255	2·312
7mm-08 Rem.	·473	·465	20°	·450	·449	·395	·310	·310	·200	1·559	1·634	1·750	2·035
7mm Exp. Rem.	·473	·465	17° 30'	·437	·436	·370	·310	·310	·200	1·999	2·104	2·199	2·540
280 Rem.	·473	·465	17° 30'	·437	·436	·370	·310	·310	·200	1·999	2·104	2·199	2·540
284 Win.	·473	·495	35°	·472	·469	·415	·315	·315	·200	1·775	1·814	1·885	2·170
30 Carbine	·360	·350				·3347	·332	·332	·200			·982	1·290
30 Rem.	·422	·4157	23°	·3969	·396	, 370	·328	·327	·200	1·498	1·528	1·578	2·050
30-30 Win	·506	·414	15° 39'	·397	·396	·370	·328	·328	·200	1·440	1·487	1·562	2·039



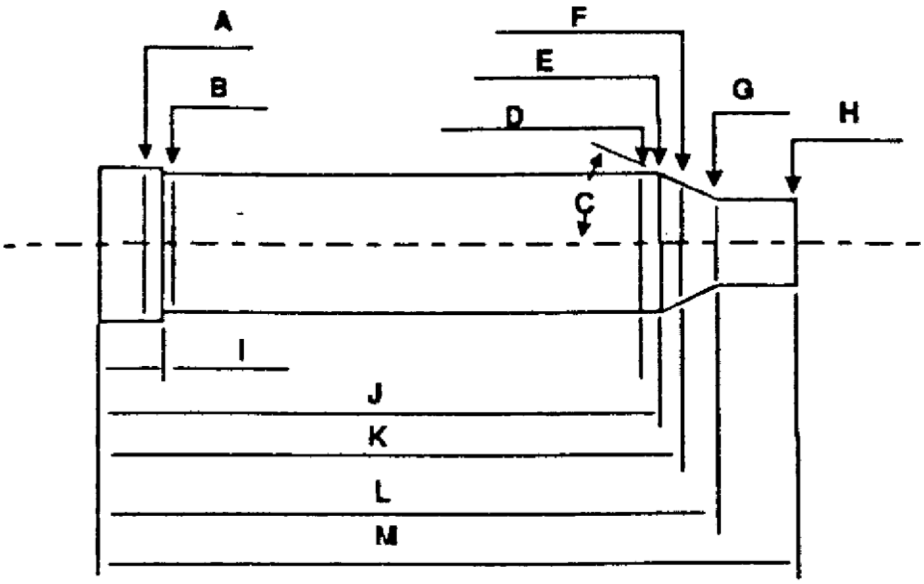
DIMENSIONS FOR RIMLESS SIZING DIE

DIMENSIONS FOR RIFLE SIZING DIES													
	A	B	C	D	E	F	G	H	I	J	K	L	M
30-40 Krag	·545	·452	21° 6'	·417	·414	·370	·3338	·333	·200	1·728	1·787	1·832	2·322
300 Sav.	·473	·4656	30°	·443	·4416	·3918	·335	·334	·200	1·558	1·601	1·650	1·871
303 British	·460	·4504	16° 58'	·4076	·406	·370	·335	·333	·200	·1790	1·832	1·890	2·222
303 Savage	·505	.·434	16°	·411	·4085	·370	·346	·327	·200	1·350	1·418	1·477	2·015
308 Win.	·473	·4653	20°	·4503	·449	·395	·338	·338	·200	1·559	1·634	1·711	2·015
32 Rem.	·422	·4157	23°	·3969	·396	·370	·341	·338	·200	1·498	1·528	1·561	2·050
32 Win. Special	·425	·4149	14° 31'	·398	·396	·370	·340	·338	·200	1·446	1·497	1·553	2·040
348 W in.	·610	·5424	19° 4'	·482	·480	·433	·373	·370	·200	1·649	1·723	1·804	2·255



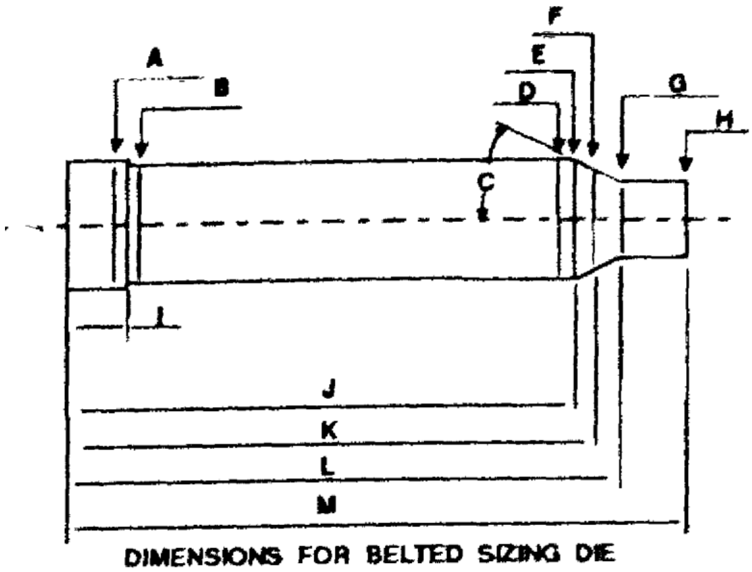
DIMENSIONS FOR RIMLESS SIZING DIE

DIMENSIONS FOR RIFLE SIZING DIES													
	A	B	C	D	E	F	G	H	I	J	K	L	M
35 Rem.	·460	·452	23° 25'	·423	·4209	·400	·382	·379	·200	1·539	1·563	1·584	1·920
351 Win.	·410	·3754				·372		·372	·200				1·380
358 Win.	·473	, 465	20°	·4503	·449	·415	·383	·383	·200	1·559	1·606	1·650	2·015
375 Win.	·506	·414				·395		·395	·200				2·020
38-55 Win.	·506	·414				·388		·387					2·129
444 Marlin	·514	·464				·449		·448					2·225

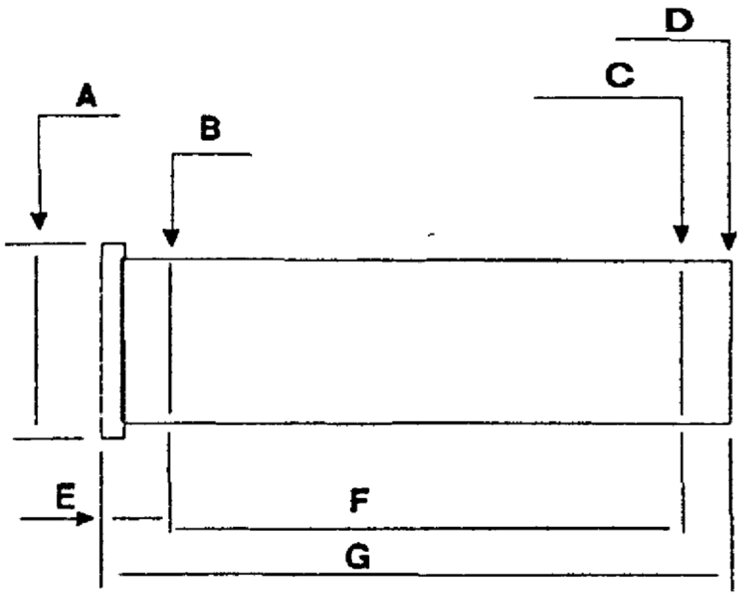


DIMENSIONS FOR BELTED SEATING DIE

DIMENSIONS FOR BELTED SEATING DIE													
	A	B	C	D	E	F	G	H	I	J	K	L	M
6·5 Rem Mag.	·535	·513	25°	·497	·496	·440	·302	·300	·220	1·708	1·768	1·915	2·194
7mm Rem Mag.	·535	·513	25°	·492	·491	·420	·317	·316	·220	2·048	2·125	2·235	2·524
8 mm Rem Mag.	·535	·513	25s		·487	·420	·357	·356	·220	2·397	2·496	2·537	2·872
264 Win Mag.	·535	·513	25°	·493	·492	·420	·301	·300	·220	2·048	2·125	2·252	2·524
300 H&H Mag.	·535	·513	9°		·449	·375	·340	·339	·220	2·125	2·360	2·468	2·870
300 Win Mag.	·535	·513	25°		·490	·420	·342	·340	·220	2·204	2·279	2·362	2·644
338 Win Mag.	·535	·513	25°	·493	·492	·420	·371	·370	·220	2·048	2·125	2·177	2·524
375 H&H Mag.	·535	·513	15°		·448	·420	·405	·404	·220	2·416	2·470	2·497	2·870
458 Win Mag.	·535	·514						·484	·220				2·520

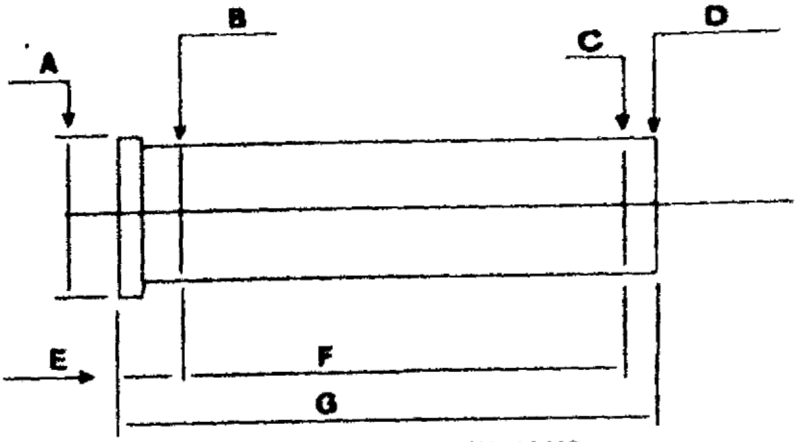


DIMENSIONS FOR BELTED SIZING DIE													
	A	B	C	D	E	F	G	H	I	J	K	L	M
6·5 Rem Mag	·527	·507	25°	·491	·490	·440	·295	·293	·220	1·380	1·759	1·908	2·170
7mm Rem Mag	·527	·507	25°	·486	·485	·415	·310	·310	·220	2·040	2·116	2·228	2·500
8mm Rem Mag	·527	·507	25°		·481	·415	·350	·349	·220	2·380	2·460	2·530	2·850
264 W Mag.	·527	·507	25°	·487	·486	·294	·260	·293	·220	2·040	2·116	2·245	2·500
300 H&H Mag.	·527	·507	8° 30'		·444	·375	·333	·333	·220	2·104	2·355	2·478	2·850
300 Win Mag.	·527	·507	25°		·484	·415	·334	·334	·220	2·195	2·270	2·356	2·620
338 Win Mag.	·527	·507	25°	, 487	·486	·415	·365	·364	·220	2·040	2·116	2·169	2·500
375 H&H Mag.	·527	·507	15°		·442	·415	·397	·397	·220	2·412	2·464	2·497	2·850
458 Win Mag.	·527	·507			·481				·220				2·500



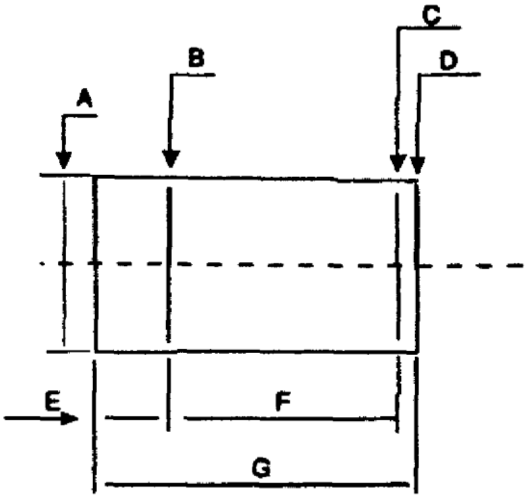
RIMMED SEATING DIMENSIONS

DIMENSIONS FOR RIMMED PISTOL SEATING DIES													
	A	B	C	D	E	F	G	H	I	J	K	L	M
22 REM JET	·444	·376	6° 40' 30"	·368,	·366	·300	·253	·252	·200	·600	·883	1·084	1·298
221 Rem Fireball	·380	·377	23°	·363	·361	·330	·255	·254	·200	1·066	1·103	1·192	1·432
256 WIN MAG.	·450	·380	25°	·370	·369	·324	·287	·286	·200	·988	1·037	1·076	1·292
25-20 Win.	·418	·347	16° 34'	·336	·334	·312	·281	·278	·200	·858	·896	·947	1·365
30 Luger	·397	·392	18°	·382	·380	·355	·3375	·332	·200	·6225	·6618	·6887	·8599
32-20 Win.	·418	·352	5° 45'	·342	·343	·338	·331	·327	·200	·881	·908	·941	1·315
38-40 Win.	·535	·468	6° 51'	·458	·456	·435	·419	·417	·200	·912	1·002	1·067	1·317
44-40 Win.	·535	·469	4° 30’	·459	·458	·452	·445	·443	·200	·918	·960	1·002	1·313



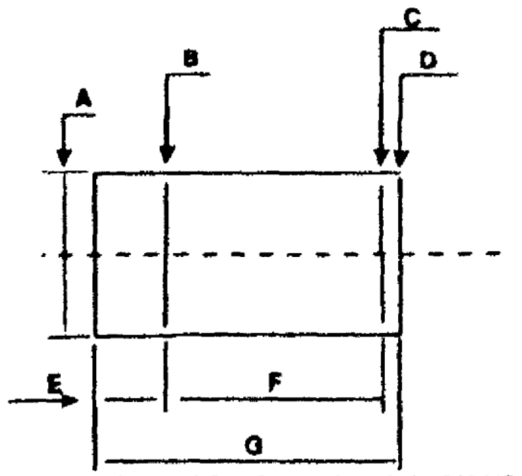
RIMMED SIZING DIMENSIONS

DIMENSIONS FOR RIMMED PISTOL SIZING DIES													
	A	B	C	D	E	F	G	H	I	J	K	L	M
22 REM. JET	·440	·370	6° 40' 30"	·363	·360	·295	·246	·246	·200	·600	·883	1·084	1·298
221 Rem Fireball	.·378	·371	23°	·357	·355	·325	·248	·248	·200	1·070	1·106	1·197	1·400
25-20 Win.	·408	·341	16° 34'	·330	·328	·307	·271	·268	·200	·858	·893	·952	1·330
32-20 Win.	·408	·351	5° 42'	·337	·336	·327	·325	·321	·200	·881	·933	·938	1·315
256 Win Mag.	·440	·374	25°	·364	·363	·319	·280	·280	·200	·983	1·031	1·072	1·281
30 Luger	·394	·385	18"	·377	·374	·351	·228	·228	·200	·375	·6225	·6887	·8599
38-40 Win.	·525	·462	6° 48'	·451	·430	·413	·413	·411	·200	·922	1·003	1·074	1·305
44-40 Win.	·525	·463	4°	·453	·451	·441	·440	·438	·200	·918	·960	1·002	1·313

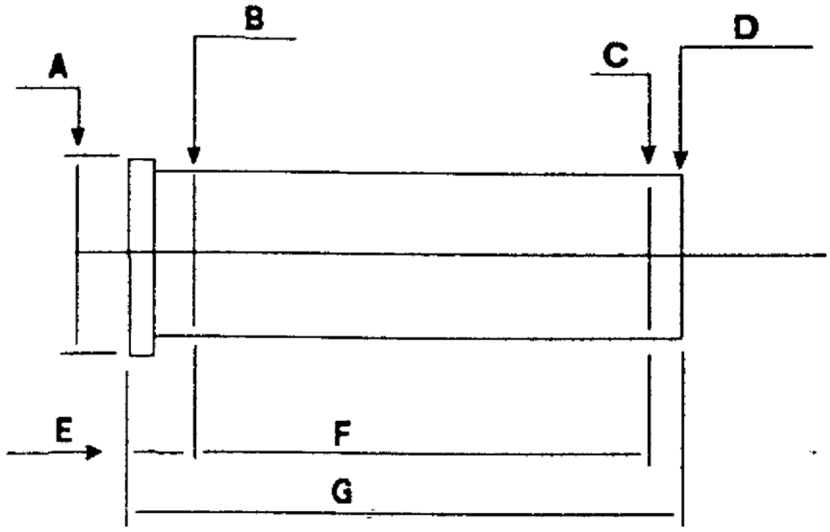


PISTOL RIMLESS SIZING DIMENSIONS

DIMENSIONS RIMLESS PISTOL SIZING DIES							
	A	B	C	D	E	F	G
25 Auto	·302	·274	·274	·274	·200		·615
9 mm Luger	·394	·386	·376	·375	·200	·500	·754
9 mm Win Mag.	·394	·386	·375	·374	·200	·880	1·160
32 Auto	·358	·332	·331	·331	·200	·400	·680
38 Auto.	·345	·380		·380	·200		·900
380 Auto	·374	·369	·3685	·368	·200	·440	·680
45 Auto	·480	·471	·4684	·468	·200	·650	·898
45 Auto Match	·480	·471	·4686	·468	·200	·650	·898
45 Win Mag,	·480	·471	·4683	·468	·200	·965	1·198

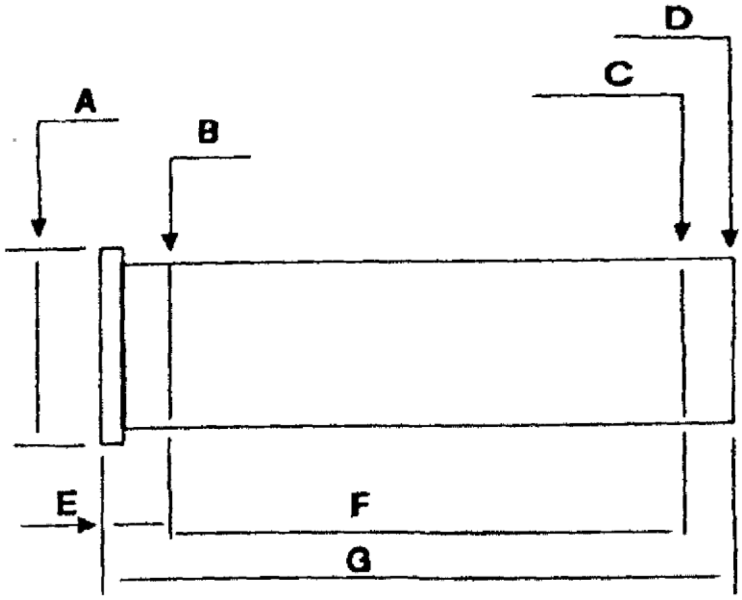


DIMENSIONS RIMLESS PISTOL SEATING DIES							
	A	B	C	D	E	F	G
25 Auto	·312	·282	·2798	·279	·200	·365	·638
32 Auto	·362	·3428	·3397	·3390	·200	·400	·693
9mm Luger	·395	·391	·382	·381	·200	·500	·754
9 mm Win Mag.	·396	·3934	·382	·381	·200	·880	1·160
38 Auto	·408	·3887	·3872	·387	·200	·650	·9179
380 Auto	·382	·3809	·3773	·3770	·200	·440	·681
45 Auto	·481	·4796	·4744	·474	·200	·650	·898
45 Auto Match	·481	·4796	·4744	·474	·200	·650	·898
45 Win Mag.	·482	·4807	·4742	·474	·200	·965	1·198



RIMMED SIZING DIMENSIONS

DIMENSIONS RIMMED STRAIGHT RIFLE SIZE DIES							
	A	B	C	D	E	F	G
375 WIN.	·506	·414		·395	·200		2·020
38-55 WIN.	·506	·414		·387	·200		2·129
·444 MARLIN	·514	·464		·450	·200		2·225
45-70	·608	·498		·476	·200		2·105



RIMMED SEATING DIMENSIONS

DIMENSIONS RIMMED STRAIGHT RIFLE SEAT DIES							
	A	B	C	D	E	F	G
375 WIN.	·506	·421		·401	·200		2·080
38-55 WIN.	·506	·420		·393	·200		2·18
·444 MARLIN	·514	·470		·4555	·200		2·240
45-70	·608	·507		·481	·200		2·109

CHAPTER 9 MAKING THE DIES

In making the loading or sizing you will need to get some one inch steel for the body. For the first die you can get some scrap steel such as an old axle from a car or truck. You will be able to machine the steel, but I recommend that you anneal it first.

Annealing is necessary to ensure grain refinement and to place the structure of the steel in the proper state for machining and heat treating. The advantages of this treatment may be summarized as follows:

1. Softens the steel for ease in machining
2. Changes the structure, thereby altering the machining characteristics
3. Relieves stresses induced by mechanical working
4. Produces a structure that will respond uniformly to hardening.

To minimize scaling and decarburization, the material to be annealed should be packed in an inert substance such as dry sand and heated to the annealing range. Packing ensures a slow cooling rate from the annealing temperature, but packing materials such as charcoal or cast iron chips should be avoided since they may cause slight decarburization. Time at temperature during the anneal should be controlled according to section thickness of the steel; up to a 1 inch section size. Then 15 minutes at temperature is adequate, while an 8 inch section would require about 2½ hour.

Cooling from the annealing temperature should be slow (maximum rate of 50° F per hour) down to 1000° F, at which point more rapid cooling is permissible.

Care should be taken not to heat the steel much above the decalescence or hardening point. When steel is heated above this temperature, the grain assumes a definite size for that particular temperature, the coarseness increasing with an increase of temperature.

If you are using a test piece of steel, which I recommend to check the reamer size, cold roll steel will be suitable.

Cut off a piece of steel about five to six inches long for the die body, two if you plan to make a sizing and seating die. Clamp one end of the steel in the three or four jaw chuck for turning. Make a light, fine clean up pass to remove any imperfections in the steel.

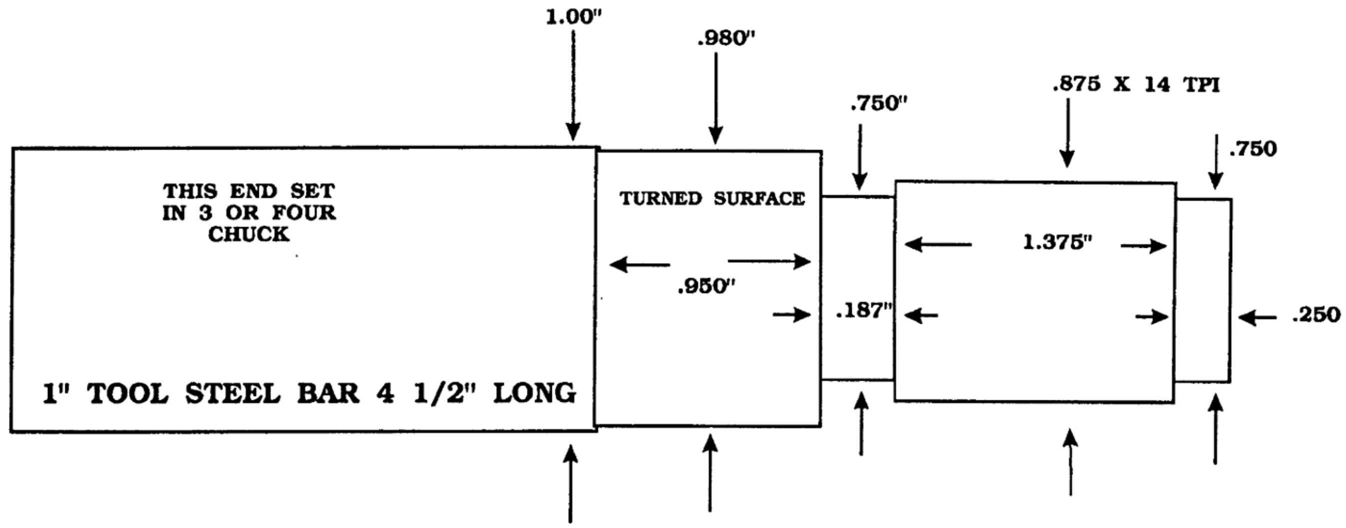
Once it is cleaned up, remove it from the lathe, reverse it and clamp on the cleaned up end. Clamp it tight so there can be no slippage or movement, as most of the work will be done without removing it from the lathe.

FACING THE STOCK.

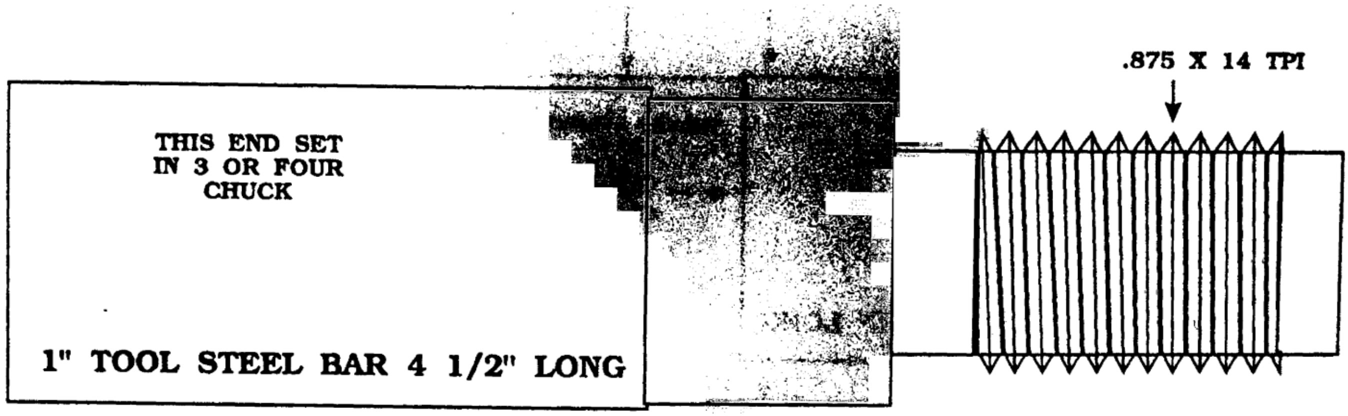
It is important to true up the end of the blank before you cut a center in it. If it is cut slightly off, the center drill may start off center and end up causing you to scrap the work. Use fine cuts here also to clean it up, then center drill the end to a depth of 1/16″ on the shoulder of the center drill.

TURNING THE BLANK.

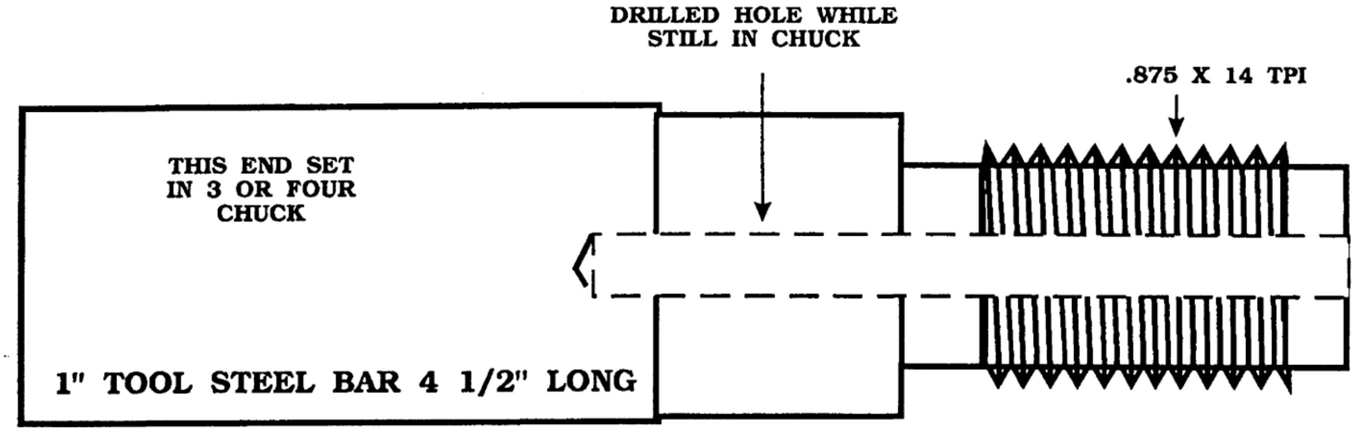
Make a pass or two to clean up the steel to get a diameter of one inch and a length of 2¾″ for a pistol cartridge. If you are making a rifle die, use a length of 3¾″ to 4″. All the dimensions giving in the drawings are for pistol cartridges. The only differences will be the decapper rod, which will of course will be longer. This rod will vary on the length of the various rifle cartridges. At this point you can knurl the outside of the die if you want.



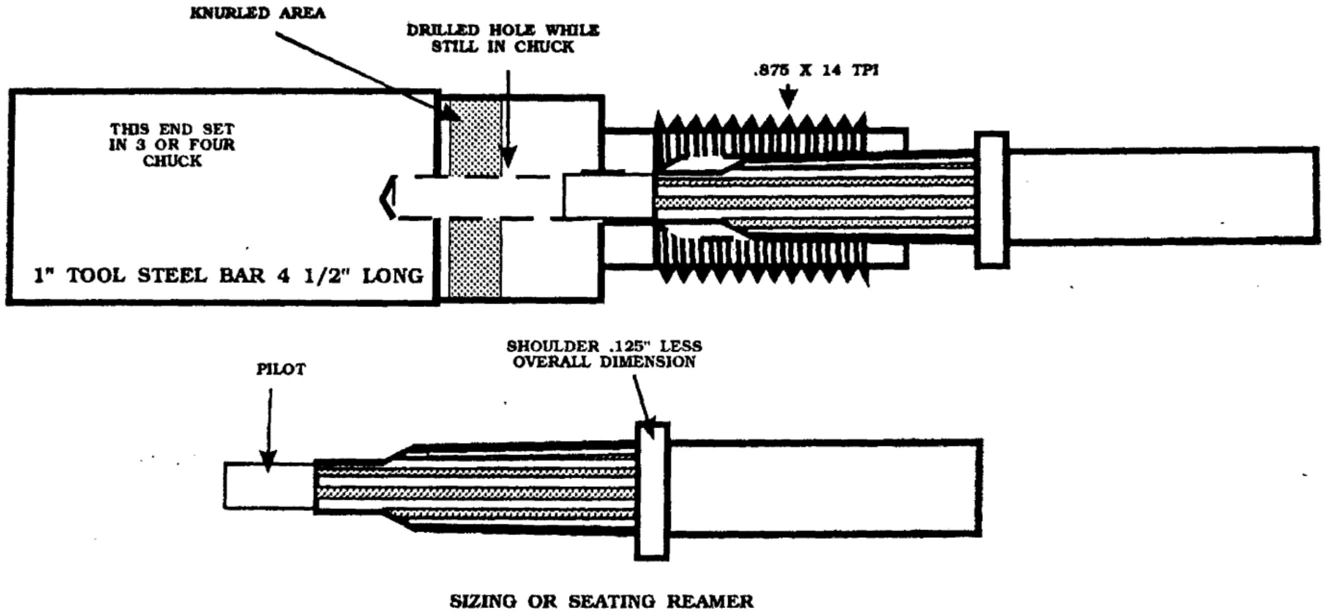
STEP NO. ONE-TURNING THE BLANK



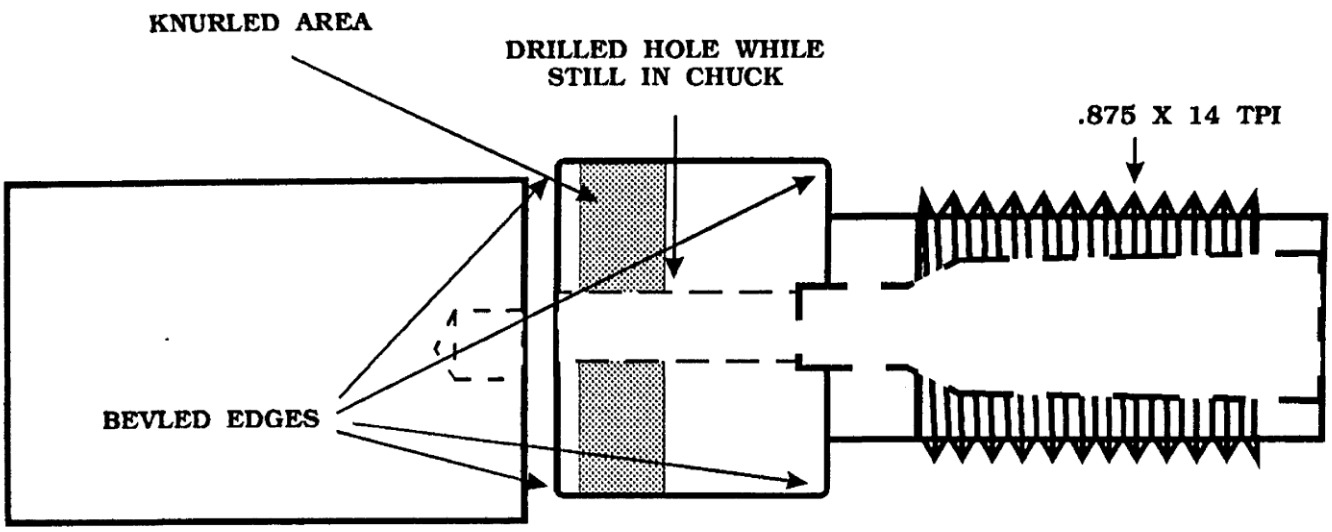
STEP NO. TWO-CUTTING THE THREADS ON THE BLANK



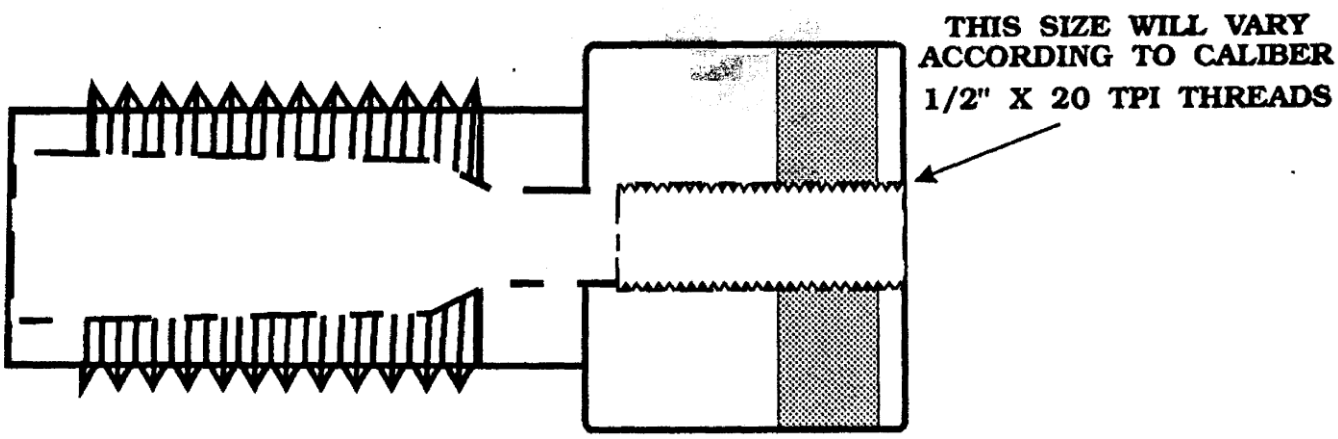
STEP NO. THREE-DRILLING PILOT HOLE IN THE BLANK



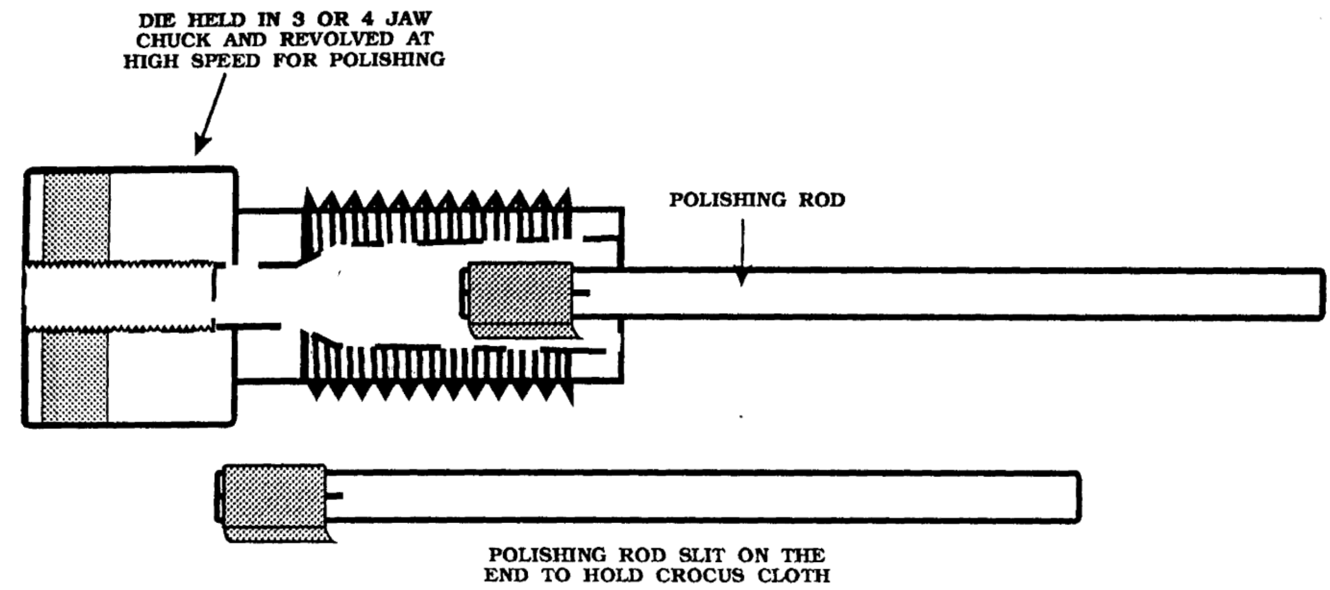
STEP NO. FOUR-CUTTING THE SIZING OR SEATING CHAMBER IN THE BLANK



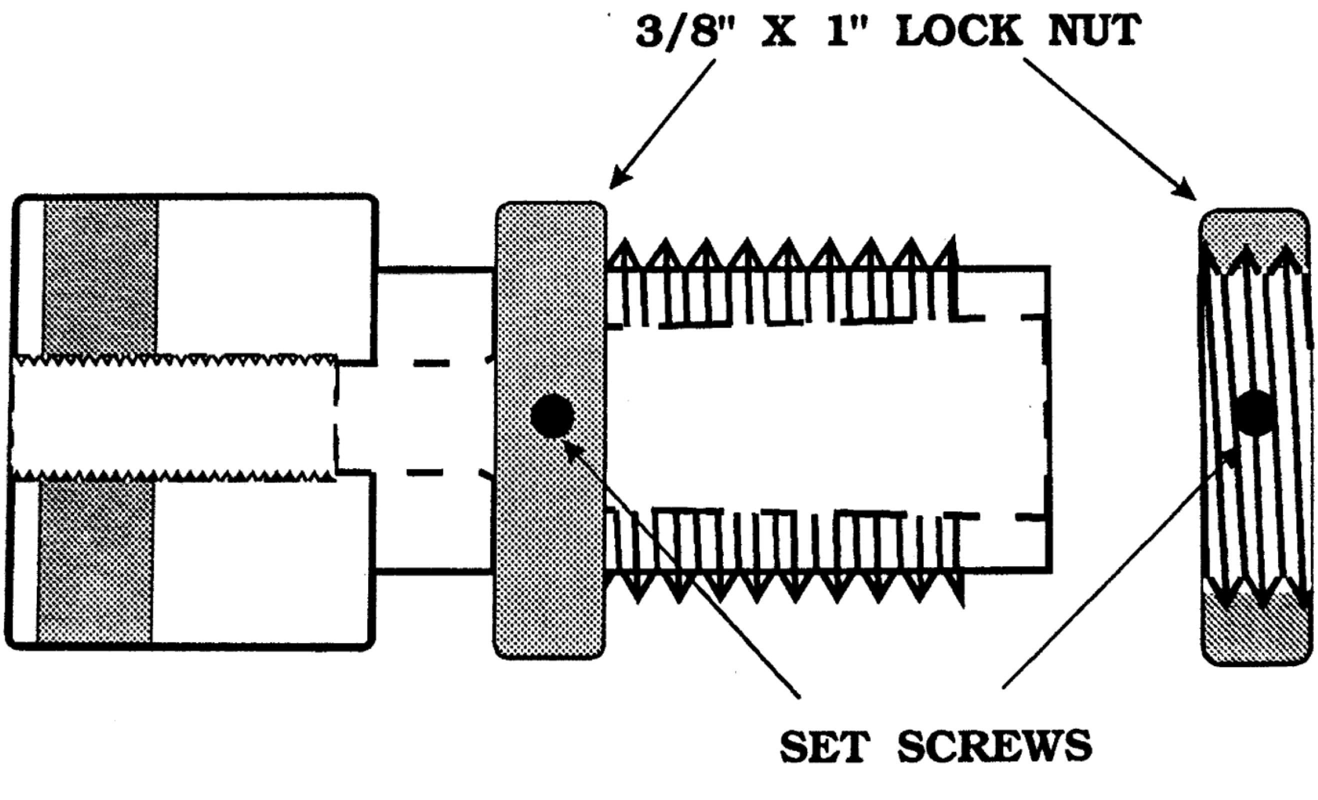
STEP NO. FIVE- CUTTING OFF THE BLANK



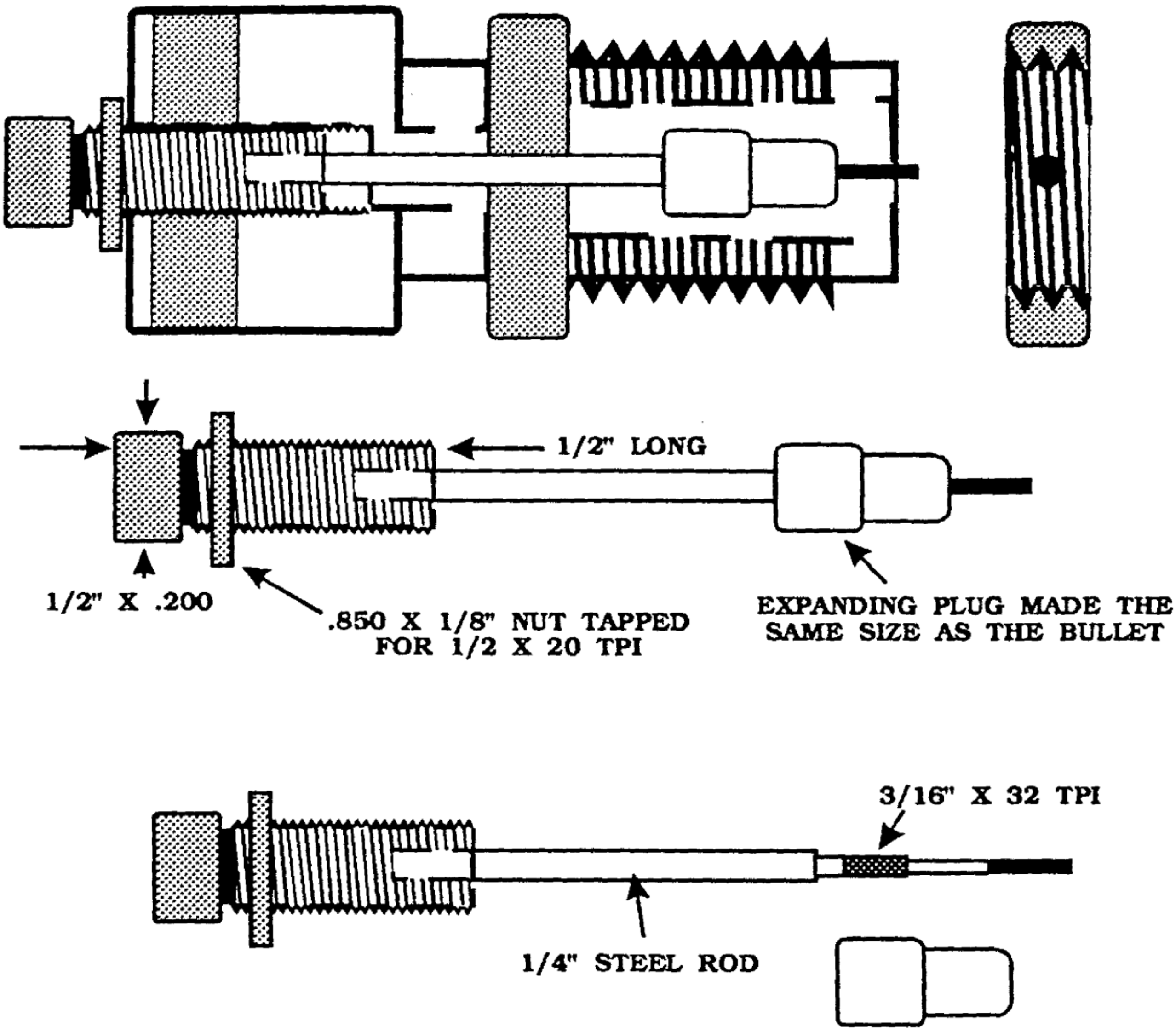
STEP NO. SIX-DRILLING AND TAPPING DIE FOR EXTRACTOR/SIZING PIN



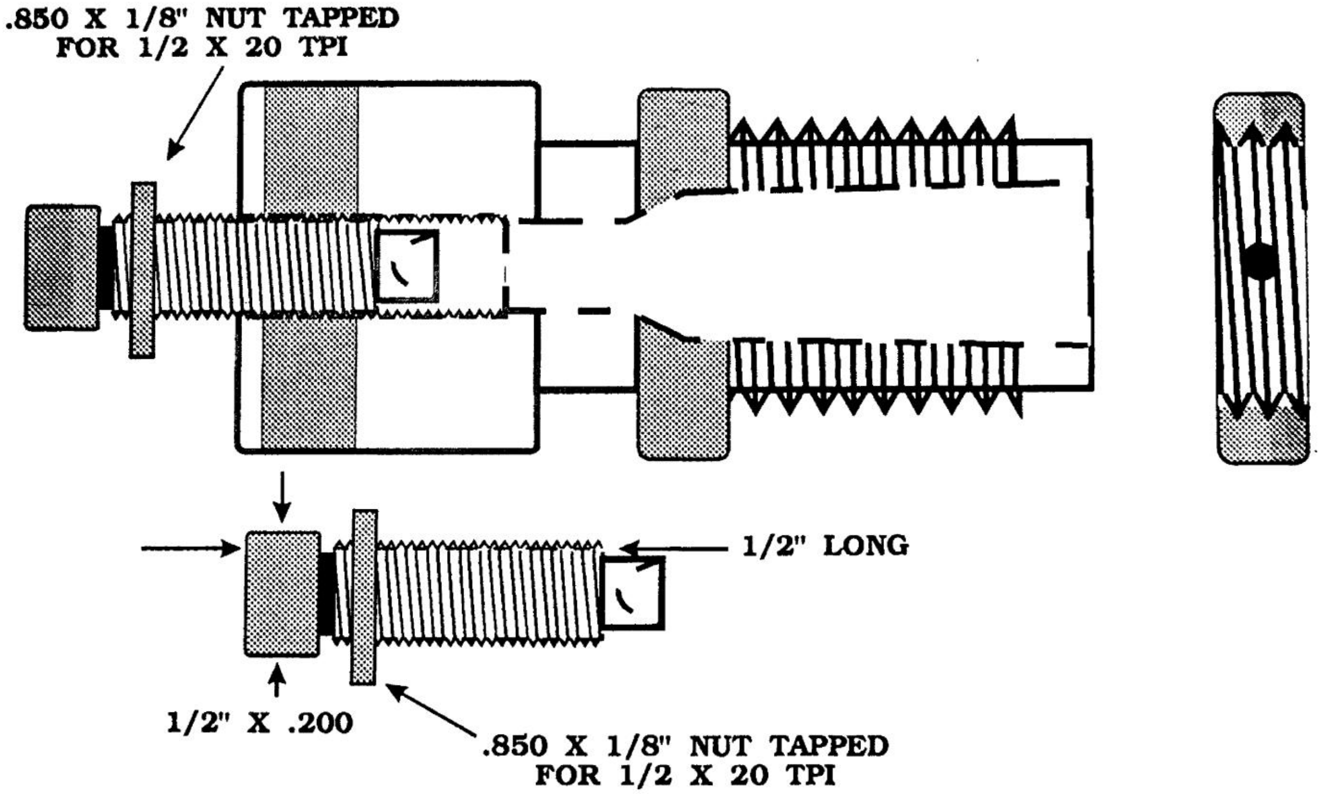
STEP NO. SEVEN-POLISHING THE SIZING OR SEATING DIE



STEP NO. EIGHT-MAKING THE LOCK NUT



STEP NO. NINE--THE DECAPPER AND EXPANDING PLUG



STEP NO. TEN- THE SEATING DIE WITH SEATING PLUG

When this length has been turned, cut a recess groove in front of the shoulder to a depth ·750 and a width of ·200″. This is the clearance so you won't run into the shoulder when threading. Now turn in the tool ·020 and make another pass on the blank, stopping when you have turned a section 1·875 inch long. Measure the stock and set the tool in and make another pass until you reach a depth of ·875″. This will be the final size that you use before threading.

On the very front of the blank cut another recess ·750 by ·200″ long. Use a file to slightly round all the shoulders. You are ready to thread the die that will be cut at 14 TPI.

THREAD CUTTING IN A LATHE

Set the lathe for a 14 TPI feed, put it in back gear drive and you are ready to cut the threads. The tool is set so its center line is at a right angle to the axis of the workpiece. This setting can be obtained by the use of the center gage as shown.

When the tool point fits uniformly into the v notch of the gage, the tool is at a 90° angle. The cutting tool is ground to the shape required for the form of screw thread being cut. For cutting 60° V threads, a center gage is used for checking the angle when grinding the tool to shape.

In cutting a right-hand exterior thread, the compound is turned in the direction of the headstock and set at an angle of 29°.

NOTE: The point of the tool should be at the same elevation as the center line of the workpiece.

The compound slide is set to an angle of 60°, and the tool is set square with the work, using the "V" notch of the thread gauge to set the tool. The point of the tool must be at the same height as the lathe centers. The tool is run up to the work with the cross feed, and the cross-feed stop is set to always bring the cross feed back to the same position after backing out the tool to return for another cut.

The compound slide is used to feed the tool into the work. By feeding the tool on the 60° angle to which the compound slide is set, the tool cuts on one side only, and it can be given a side rake to make the chip clear the thread groove. If the tool is fed in square with the work, it must cut on both sides. No side rake can be used, and the two opposing chips will interfere and jam in the cut.

The compound is adjusted so the micrometer dial on its collar is at zero. The tool is then brought into contact with the workpiece by adjusting the crossslide and setting its micrometer dials to zero. All adjustments for depth of cut can be made from these settings.

It is a practice to use both the cross-slide, and the compound. The tool is backed off the workpiece and the carriage is moved to where the tool is, at a point beyond the end of the workpiece. The cross-slide is then advanced until the micrometer dial reads the same as where the tool was touching the workpiece.

Next, the compound is advanced ·002 to ·003″ and a trial cut is taken. At the end of the cut, the cross-slide is backed off and the tool returned to its starting point. The cross-slide is then adjusted to its zero reading and the compound advanced a distance equal to the next cut. The operation is repeated until the proper depth of thread is obtained.

The carriage is attached to the feed screw by closing the half-nuts. There is a safety interlock between the friction feed for turning and the half-nuts for thread cutting, so the two cannot be engaged simultaneously, which would wreck something.

At the end of each cut, the half-nuts are opened, and the tool is withdrawn from the cut, so the carriage can be returned to the start for another cut. To be successful you must work quickly with both hands, back the tool out with one hand while you open the half-nuts with the other.

When you return the tool for another cut, advance the compound slide by the amount of the chip. Never change the setting of the cross-feed stop after you have started to cut a thread or you will throw the tool out of alignment with previous cuts. If the tool is not withdrawn from the cut, the backlash of the feed gears would leave the tool out of line with the thread and if the lathe was reversed, the tool would damage the thread.

If your lathe is not equipped with a thread cutting dial, you must reverse the lathe to return the tool to the start for another cut. Without the thread dial, the half-nuts cannot be opened until the thread is completed,

The thread-cutting dial indicator is a dial geared to the lead screw. When the carriage is stationary, the dial revolves, but when the carriage is cutting a thread, the dial is still. There are several graduations on the dial, each numbered. As the dial revolves, the half-nuts are closed when the correct number comes up to the index mark. For most even numbered threads, there are several places on the dial that can be used to close the half nuts. For odd- numbered threads there is only one position, and the half-nuts must always be closed on the same number used to start the first cut.

Start the first cut, close the half-nuts on the number 1 line of the dial and feed the tool with the compound until the tool just scratches a fine line, indicating the thread. Shut down the lathe and test this line with the thread-pitch gauge to see that the lathe is cutting correctly. The cross feed of the carriage must always be up tight to the cross-feed stop before moving the tool with the compound feed.

When you are getting close to the final size, use a precut nut (which you can get from a factory loading die) to check the size. If the nut will not screw on make another light pass and try again. When the nut will just screw on, make two or three additional passes at the same setting to finish cleaning up the threads.

A lock nut has to be made for each die that you make. To make this you will need to bore and thread a piece of steel. The stock for the lock nut is 1·250″ and will be cut off to a thickness of 1/8″. You can drill and then bore the bar to the correct I. D. for the ·875 by 14 TPI thread.

You can either thread it on the lathe or buy a tap the correct size. When you drill and bore the bar, go deep enough so after tapping you will be able to get several nuts from it. If you use a tap, use the tail stock center to hold it straight in the bar.

Lock the spindle and with plenty of oil on the tap work it in with a small wrench, backing it off about every full turn. A cut off tool is used to cut off these nuts so they will be cut straight.

CHAMBERING THE DIE

Now drill a hole down through the center of the die to a length just past the where the die will be cut off. Use a smaller drill than the finish size as you will clean up the hole with the correct size drill.

When you make the reamer you will need to make the pilot to the size of the drill that you use to clean up the bored hole. The correct size of the drill will be one that is slightly smaller than the bullet that will be used. Make a reamer the size of the bullet and ream the drilled hole to that size. The pilot should just slip in the hole that the drill has made with a close fit.

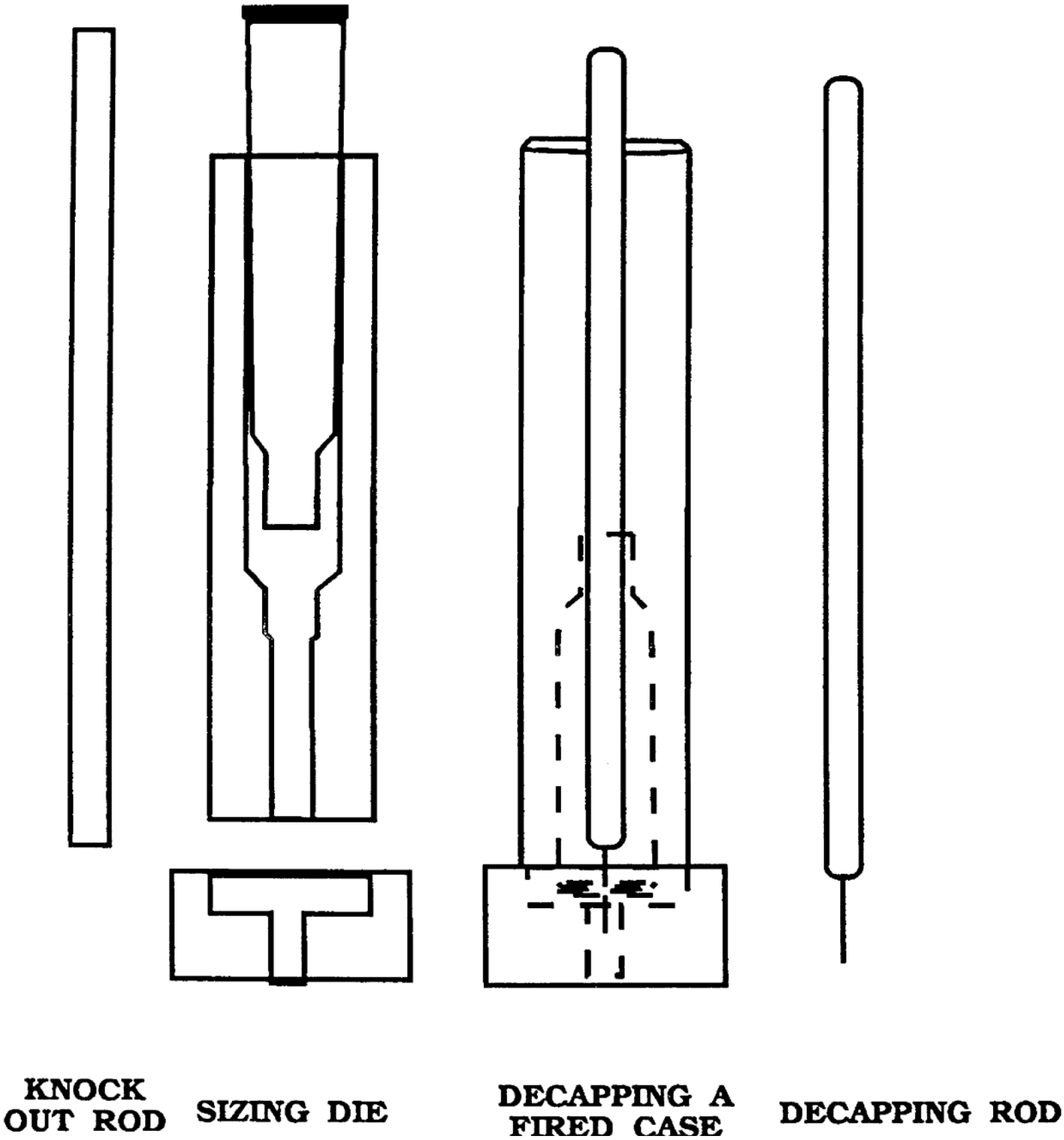
Drill the hole out with the correct size drill, using a good lubricant and a good speed on the lathe as you want to ream it as well as drill it to size.

Chambering the dies is the next operation that will be done. Put a small lathe dog on the end of the reamer. Start the pilot into the drilled hole, and put the tail stock center in the center in the reamer. Move the compound slide in position so that the lathe dog rests on it.

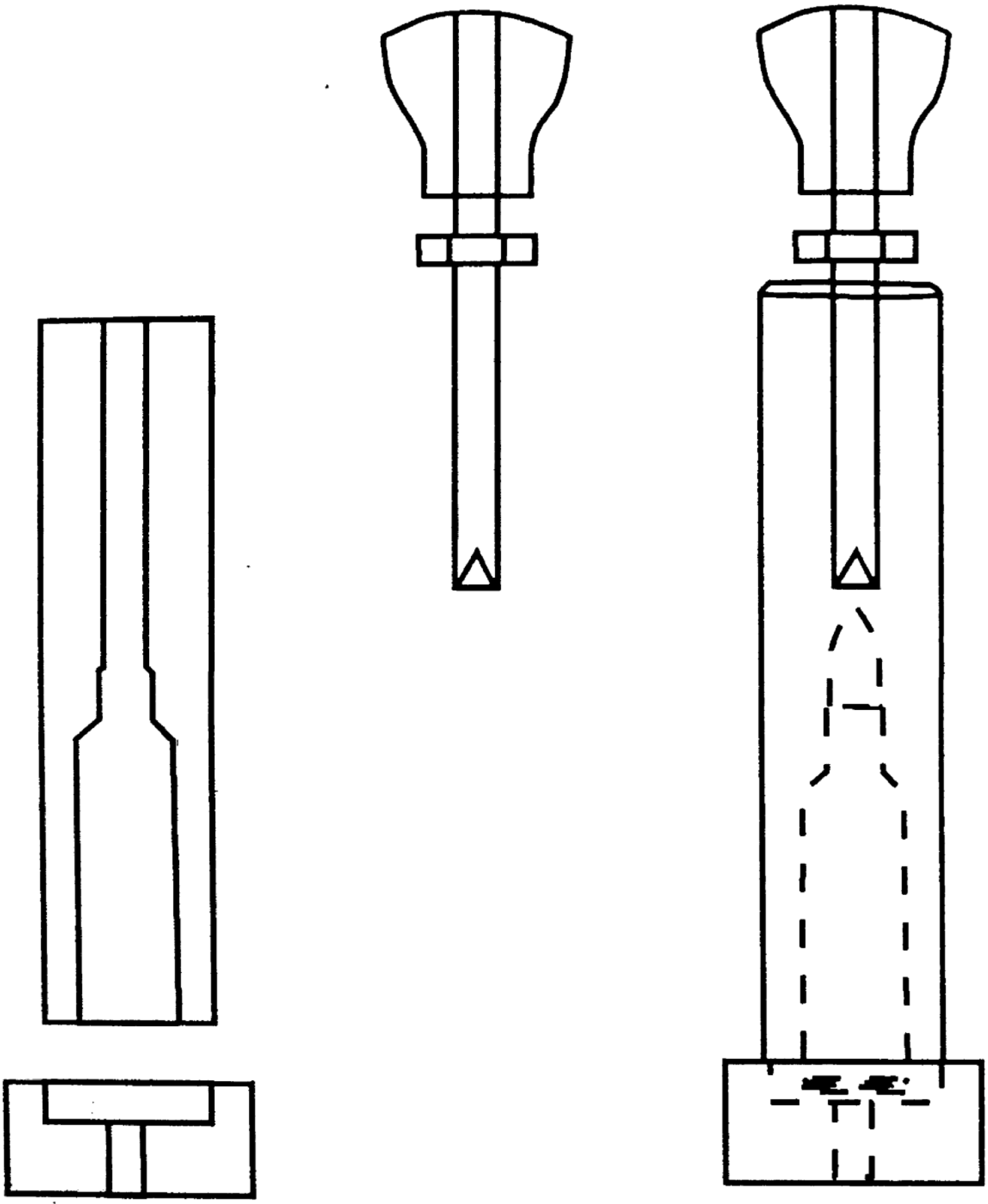
Set the lathe in back gear, and holding the reamer back against the tail stock center, turn the lathe on. Slowly feed the reamer into the die. If you are using a large case die, first drill it out to slightly smaller than the shoulder of the reamer using a standard drill bit and up to 1/8″ of where shoulder will be.

It would be better if you use a reamer with a stop shoulder on it so you do not have to worry about the depth. If you are making loading dies for a press, you can cut a stop shoulder ·130 from the back of the case. Using the ·200″, subtract ·130″ from ·200 to get ·070″ for the shoulder. This will guarantee that you do not cut the die too deep.

If you are making a sizing or seating die that you will run the case in full length, cut a shoulder at the end of “J” for a stop.



TWO PIECE DIES



BULLET SEATING DIES WITH SEATER

Run the reamer in using the tail stock crank to about a 1/4″ to 1/2″. This depends on how much stock you are removing from the die. Back out the reamer ALWAYS HOLDING IT AGAINST THE CENTER and blow out the hole and all the chips from the reamer. If you do not hold it against the center it may grab and wreck the die and break the reamer.

Set up the lathe for a cut off tool and cut off the die from the rest of the blank. Put the die back in the lathe with the cut offend sticking out so it can be faced off. Face off the end with fine feeds and polish and radius the edges.

When you set the die back in the lathe, use a dial indicator to get it trued up perfect. If you don't, you will size the neck or seat the bullet crooked, so be extra careful here.

Now drill and tap the hole for the decapper or seater stem just as you did in making the nuts. The seater neck should just clear the walls of the reamed hole.

Turn the die back around and clamp it back in the lathe for the final finishing.

When you have reamed to the stop, back out the reamer, turn off the lathe and blow out all the chips from the hole. It is now ready for polishing and finishing.

If you are going to heat-treat the die now is the time to do it. Be sure to coat the entire die with a anti-scaling compound including inside the reamed hole.

POLISHING THE DIE

Get a 1/8″ rod about six or seven inches long and using a hack saw cut a slot one inch in the end of the rod. This slot will hold the various grit sizes that you will use to polish the inside of the die.

Set the lathe to the highest speed and turn it on. Put a piece of medium emery cloth in the slot and stick it in the reamed hole. Polish in long strokes from the shoulder to the end of the reamer. DO NOT OVER POLISH THE HOLE AND MAKE THE DIE OVERSIZE.

Stop the lathe and blow out the hole and wipe it clean so you can inspect it. If it looks smooth without reamer marks, change the grit to a fine grit and repeat the process. Finish with crocus cloth to a mirror finish. Repeat the above process on the shoulder and the neck.

The other parts for the die can be machined in the same fashion as the die. When you make the decapping rod it has to be centered and the hole drilled in the end for the decapping pin. Use a small center drill to start it exactly center, then drill the hole. The expanding plug is made of tool steel, turned to ·002 smaller than the bullets. It is then polished, tapped, and coated with a anti-scaling compound and heat treated.

You will probably need to make a couple to get the process down pat. All the tools such as reamers can be used on a hundred or more dies before being discarded.

CHAPTER 10 MAKING BULLET MOLDS

There is nothing difficult in making bullet molds once you have mastered tool making. You will need two main cutting tools:

ROUGH DRILL

1. A drill to drill out the mold before you cut the mold with the bullet cutter or cherry. I have found that it is easier to make this drill from a suitable size number drill. Get the drill the right size for the bullet that you want to make, minus ·040″. By drilling out the mold first with this rough drill it will be easier and take less time to finish cutting the mold.

To make this drill, chuck the drill in the lathe, and with the Tool Post Grinder grind the drill to the basic shape of the bullet that you are making. See the section on making tools.

After you finish grinding the drill to shape, sharpen the ground area so it will cut again. This drill will work for several different bullet sizes. Be very careful here as if you are working on small calibers it may grab and ruin the cherry.

BULLET CHERRY OR CUTTER

2. There are two ways to make a cherry, both will do just as good a job. The first is to make the bullet cherry from a piece of 01 tool steel and finish it to size. Once it is to size, cut the grease grooves in the cherry with cutter bits ground to the correct shape.

All that is left then is to set up the Mill and indexer and flute the cherry using six or eight flutes. Here again be very careful and do not take heavy cuts, as you might bend the cherry. Use plenty of coolant or oil to keep the heat down to minimize chances of warpage.

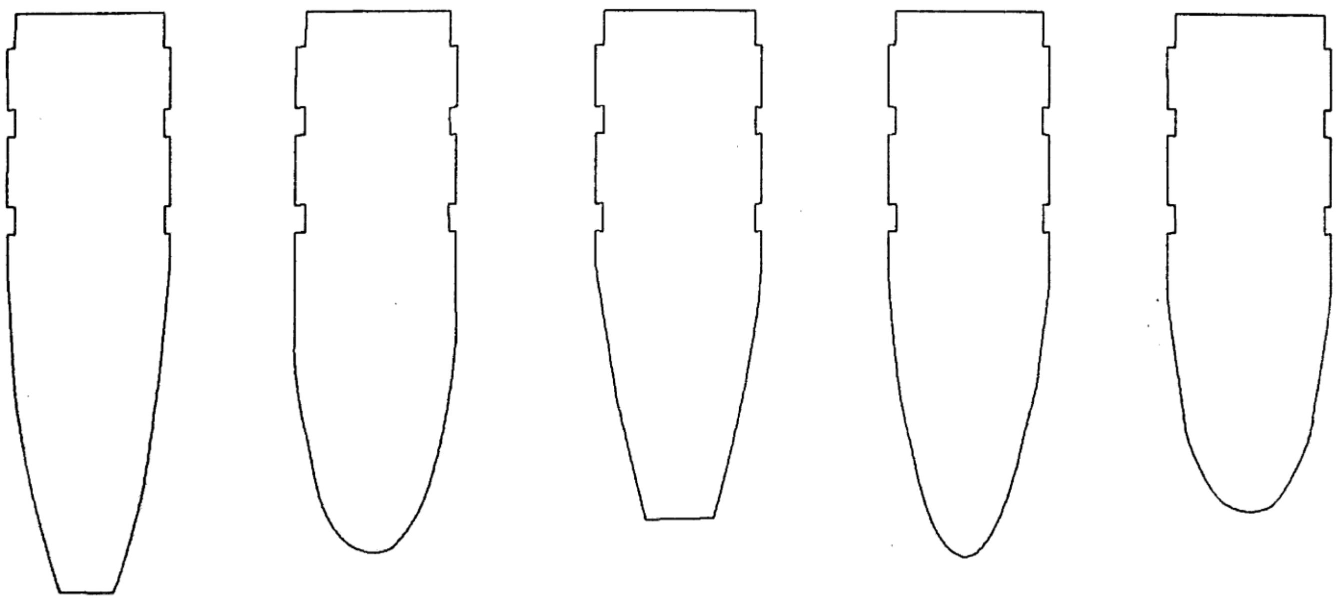
After finishing the fluting, put the cherry back in the lathe. Now with Machinist Needle Files carefully file in the back clearance from the cutting edge. It is easier to do it now than to try to grind it in after heat treating. File within about ·010 of the cutting edge and stop.

The clearance needed there can be honed in with an Arkansas stone after heat treating. The cherry can be finished to size, and then coated with a anti-scaling compound that can be purchased from Brownells. See Appendix.

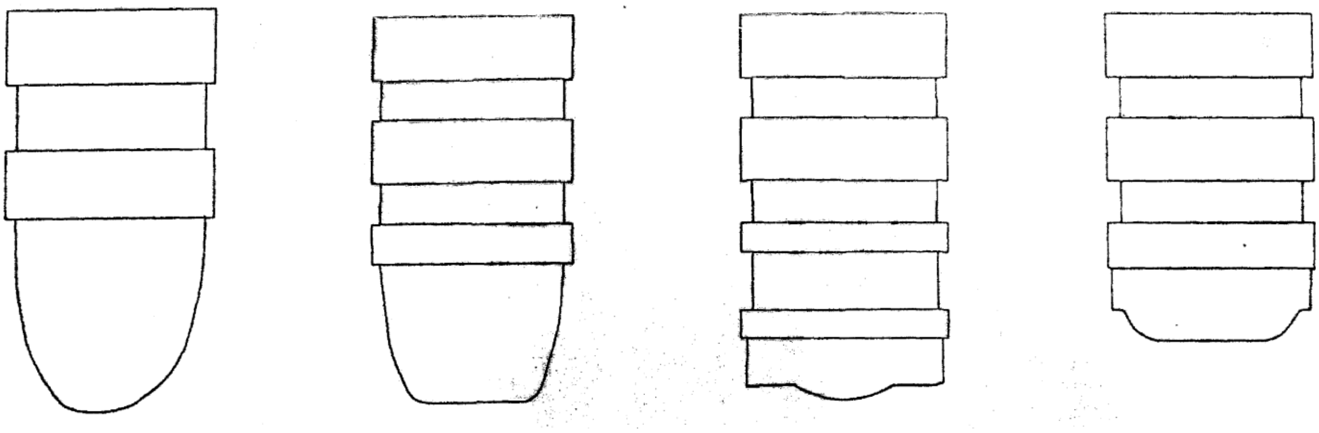
It can now be heat-treated but be careful that when you quench the cherry go absolutely straight into the oil. If you are using a water hardening tool steel it can be quenched in a warm brine solution.

DUTCHMAN REAMER

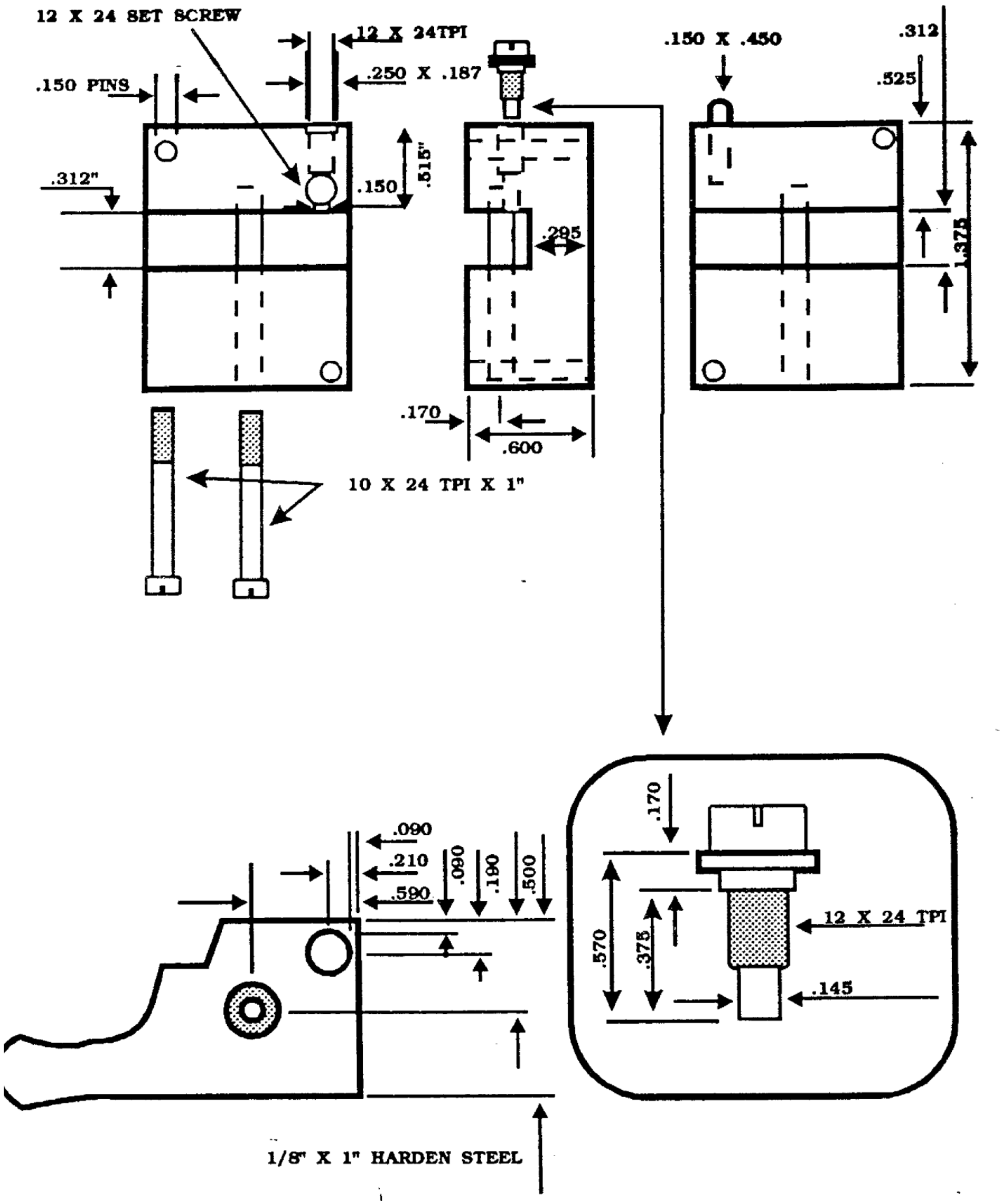
You can also make the cherry the same as above but as a Dutchman reamer. This type of reamer can be used as a chambering reamer or handle just about any job a fluted reamer can do. You make the cherry the same way as above, finish it and heat-treat it. When completed you carefully grind the reamer to exactly 1/2 of its diameter. See Drawing. Then you grind the clearance on the back side as shown and stone the cutting edge. This type of reamer will cut a very smooth hole that may not need to be lapped.



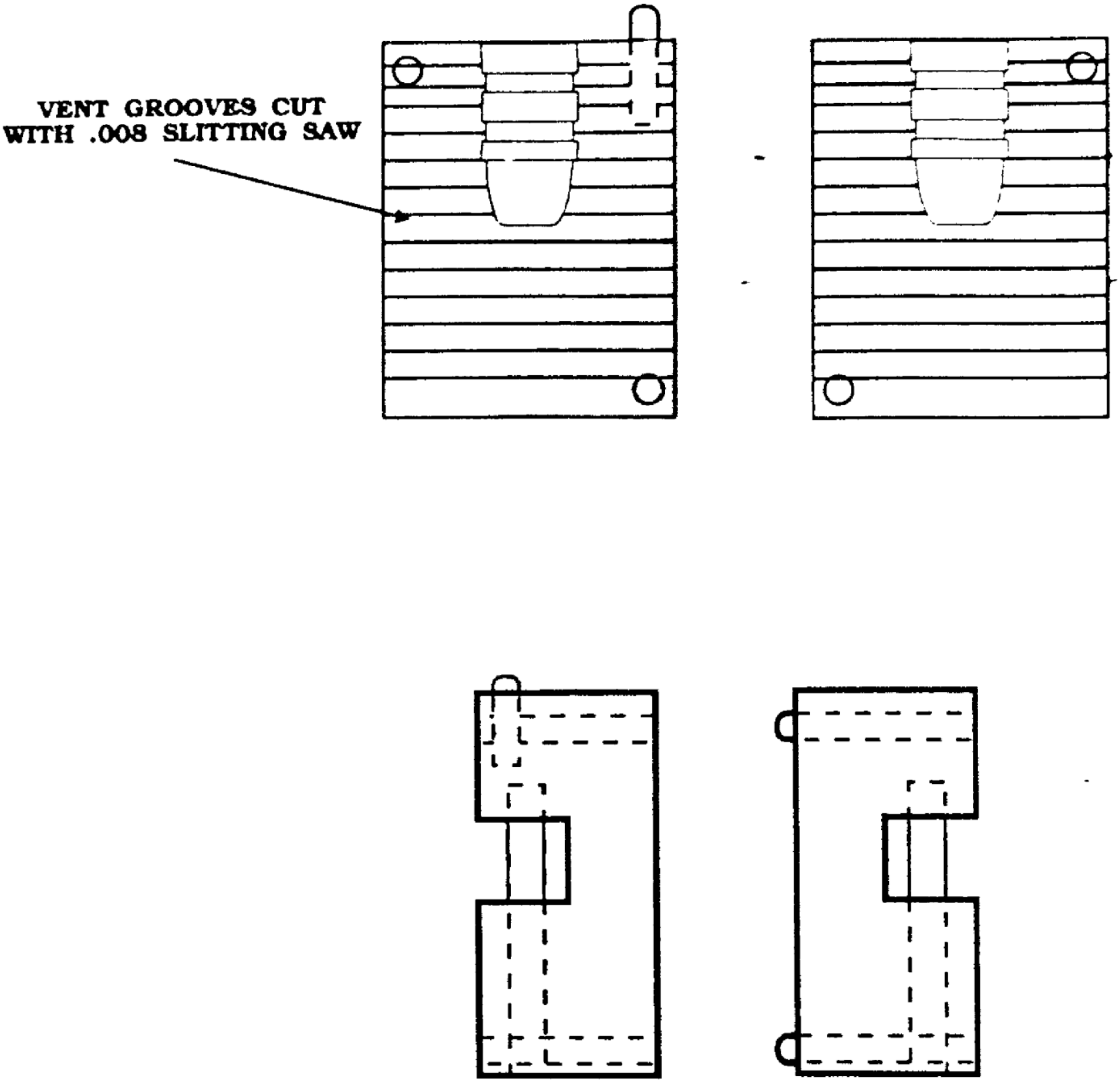
DIFFERENT STYLES OF RIFLE BULLETS



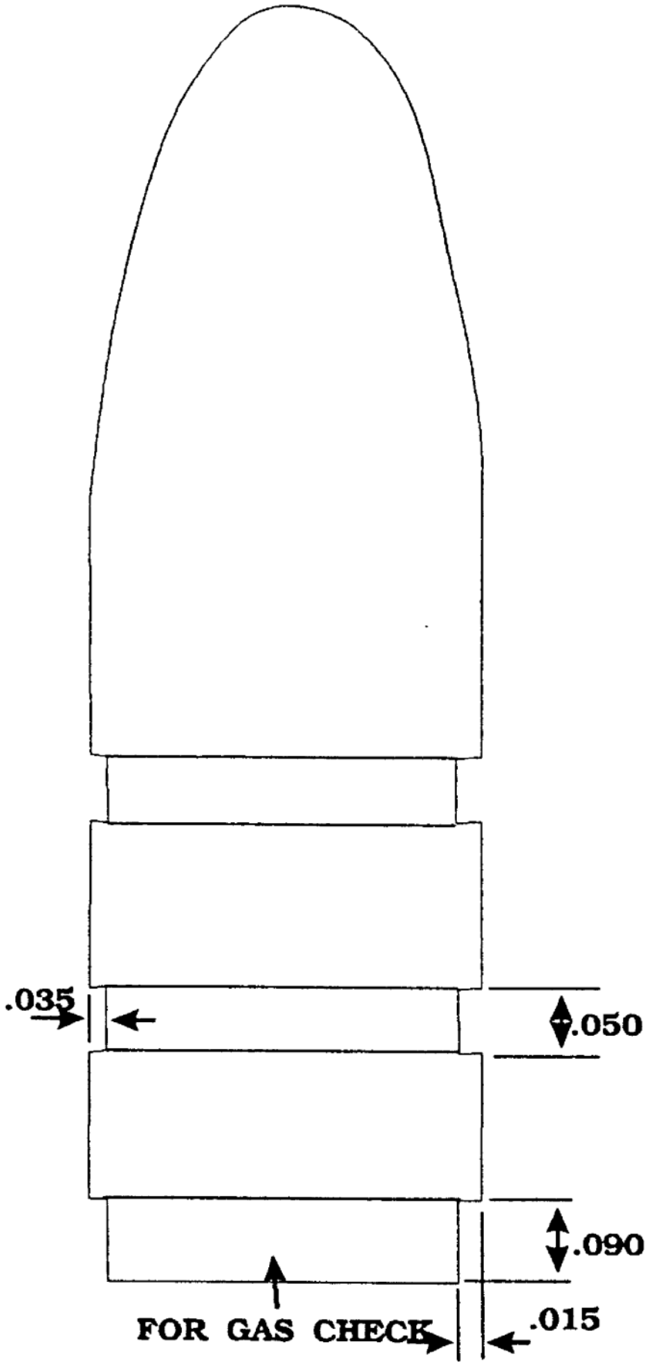
DIFFERENT STYLES OF PISTOL BULLETS



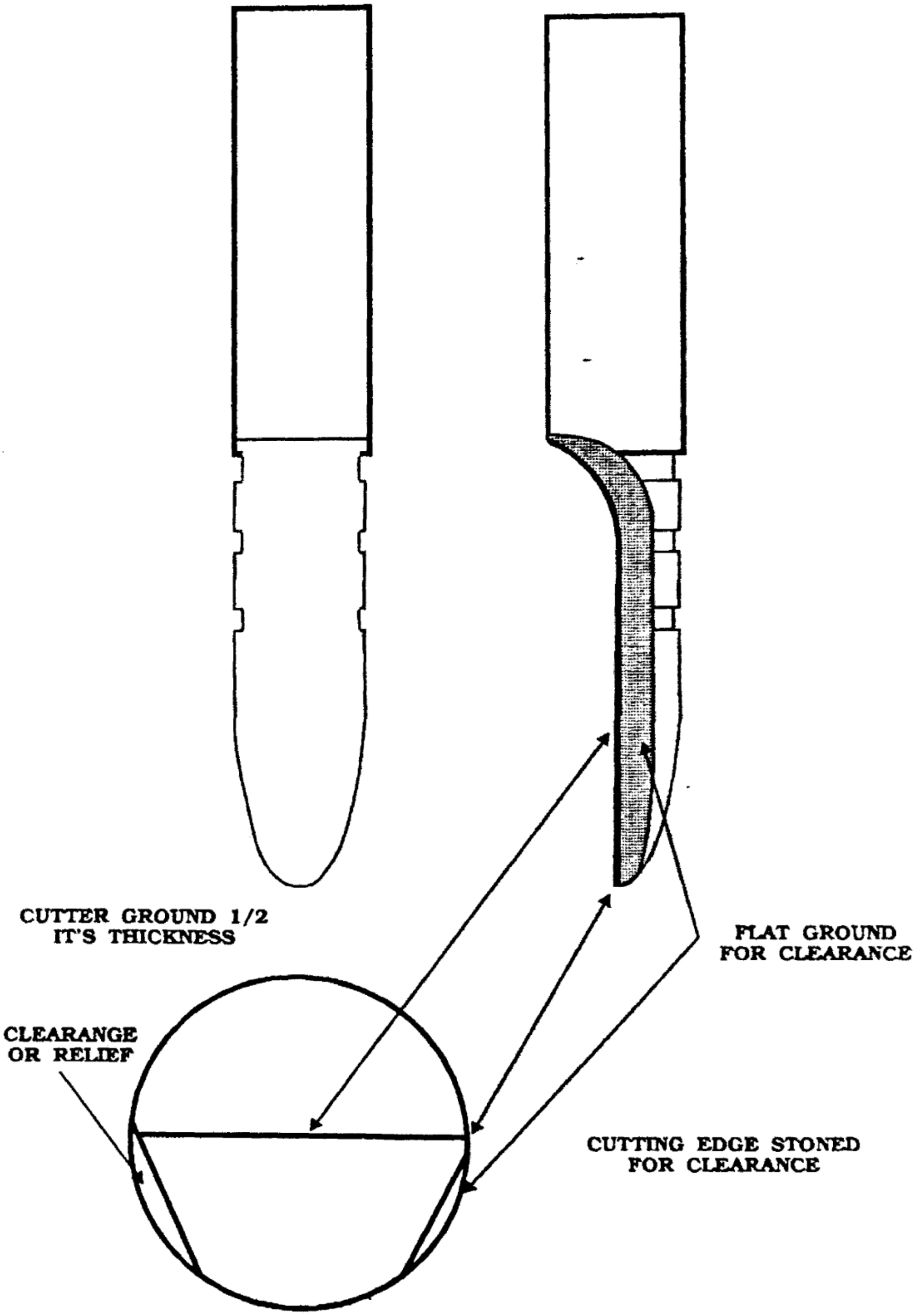
MOLD BLOCK PARTS



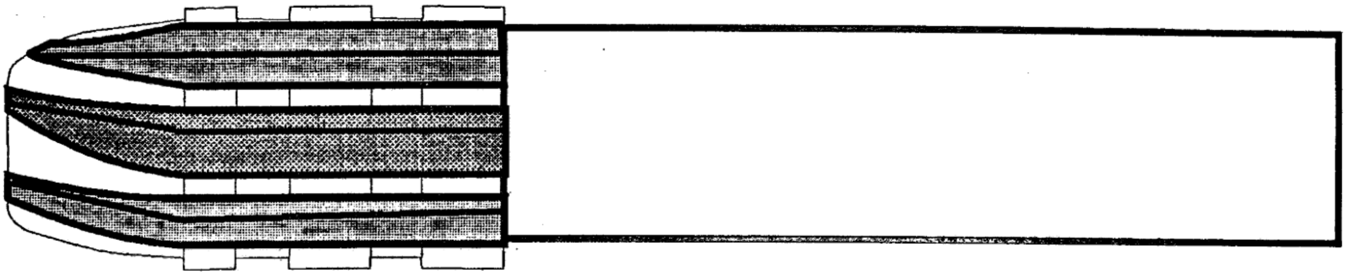
BULLET MOLD CONSTRUCTION



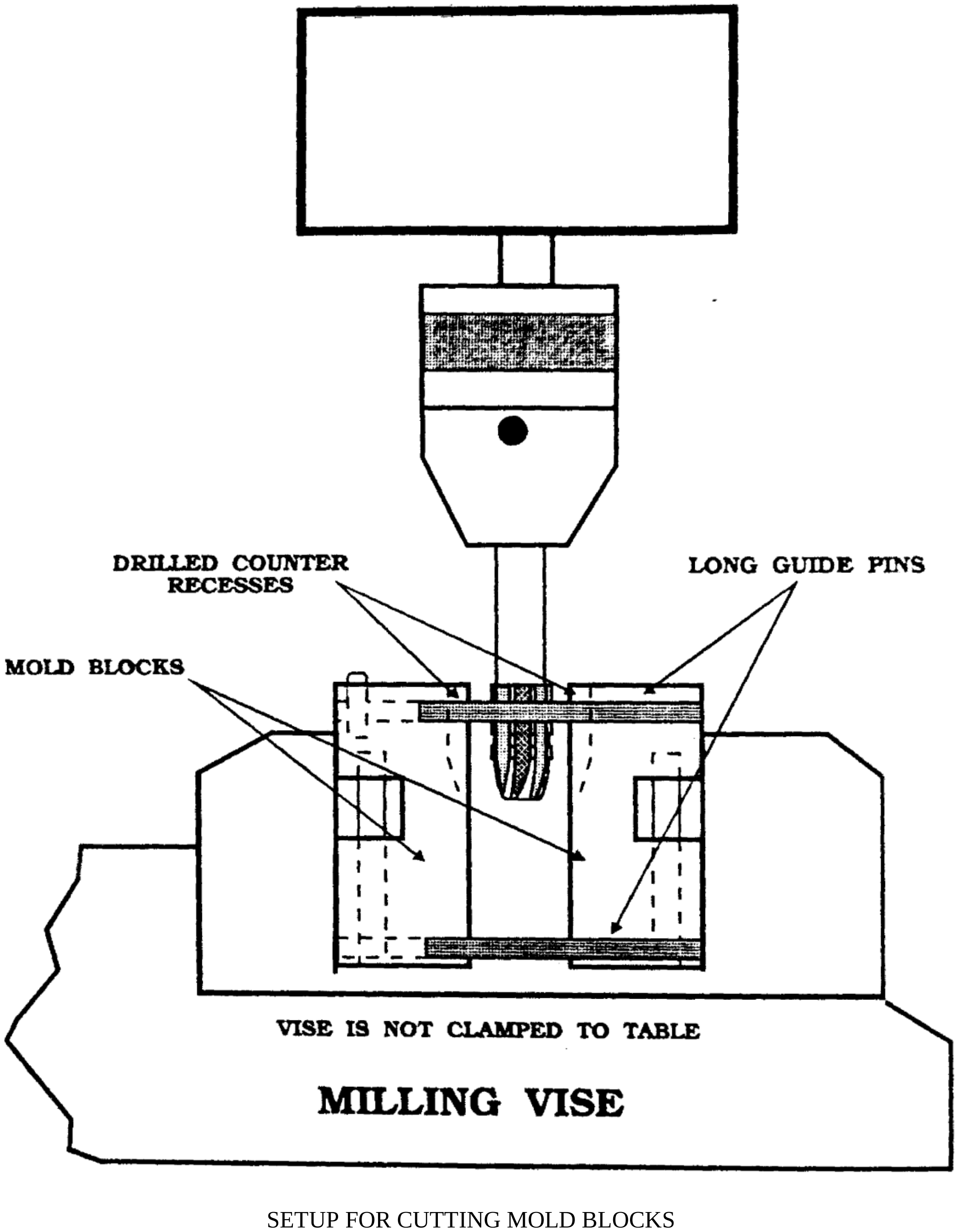
AS CHECK AND GREASE GROOVES ON BULLET



MAKING THE DUTCHMAN BULLET REAMER



FINISH CUTTER FOR BULLET MOLD



SETUP FOR CUTTING MOLD BLOCKS

MAKING THE MOLD BLOCKS

Making the blocks are a simple machining operation on the Milling Machine. The blocks can be made from Steel or Aluminum, and the Aluminum will be the easiest to make and will last a long time with a little care.

The block material can be cut from suitable material with a band saw or cutoff saw. Once it is cut to a rough size, clamp it in the Milling Vise and fly cut it to the correct size. A standard end mill will work but the finish will not look as good as if you used a fly cutter.

After machining clamp each block in the vise and true it up with a dial indicator. You will need a ·008 slitting saw which you can purchase from a machine supply house or Wholesale Tools. With the slitting saw cut air vents in the mold blocks as in the drawing. You can buy an arbor from Wholesale Tools so the slitting saw will work in a Vertical Mill.

After drilling all the holes insert a line up pin into the holes on both blocks. The blocks should line up perfectly. When you drill the holes for the line up holes, drill one side of the block, clamp them together. Use the holes in the first block as a guide and drill the other block.

Tap all the holes that need tapped and make sure everything fits as it should.

With the pins in the blocks clamp the blocks upright in the Vise, and true it up. Put a drill chuck in the mill and a small center drill for marking the blocks for rough drilling. Carefully center the center drill exactly over the seam of mold halves. The exact center can be found with an edge finder that is sold through Wholesale Tools.

Center drill the blocks deep enough so the roughing drill won’t wander when you start it. With the roughing drill you only want to go deep enough but not so deep that the cutter or cherry will not be able to clean up the mold.

At this time unclamp the vise from the table as when you cut the mold it wants to be free to keep the cutter in the center. Once the roughing hole is drilled, remove the drill and replace it with the cherry. Unclamp the mold blocks from the vise and spread them apart far enough that the cherry will just go down inside the roughing hole.

Adjust the cherry to the right depth and lock the spindle then turn on the Milling Machine. Slowly tighten up the vise and the cherry will start to cut the blocks to shape. Keep closing the vise until it is fully closed, being careful that there are no chips between them. You should not have to lap the as the finish if you were careful at the final operation of the last four to five thousands.

The mold is done except for the aging process of using.

BULLET SIZES

SIZES OF COUNTER BORES AND CUTTERS

Here is a list of finish sizes of the counter bore drill, and the bullet cutter.

CALIBER	BORE DRILL	CUTTER
22 short	·185	·2255
22 LR	·185	·2255
22WRF	·185	·2285
22WCF	·185	·226
22 Hornet	·185	·226
218 Bee	·185	·226
221 Fireball	·185	·226
222 Rem	·185	·2265
223 Rem	·185	·2265
222 Mag	·185	·2265
219 Wasp	·185	·2265
219 Zipper	·185	·2265
224 Wea.	·185	·2265
225 Win	·185	·2265
22-250	·185	·2265

CALIBER	BORE DRILL	CUTTER
220 Swift	·185	·2265
22 Savage	·185	·2285
243 Win	·206	·246
244 Rem	·206	·246
6mm	·206	·246
6mm 2 84	·206	·246
25-20	·220	·260
25-35	·220	·260
256 Win	·220	·260
250-3000	·220	·260
257 Roberts	·220	·260
257 Roberts Imp	·220	·260
25-06	·220	·260
257 Wea. Mag.	·220	·260
6·5 Jap	·220	·266
6·5mm M. S.	·220	·266
6·5x55	·220	·266
6·5 Rem Mag.	·220	·266
264 Win Mag.	·220	·266
270 Win	·240	·2805
270 Wea Mag.	·240	·2805
7 mm Mauser	·240	·287
284 Win	·240	·287
280 Rem	·240	·287
7x61 S&H	·240	·287
7 mm Rem Mag	·240	·287
7mm Wea Mag	·240	·287
7·5 Swiss	·271	·311
30 Carbine	·271	·311
30-30 Win	·271	·311
300 Sav	·271	·311
308 Win	·271	·311
30-40 Krag	·271	·311
30-06	·271	·311
30-06 Imp	·271	·311
300 H&H Mag.	·271	·311

CALIBER	BORE DRILL	CUTTER
308 Norma Mag	·271	·311
300 Win Mag.	·271	·311
300 Wea Mag.	·271	·311
7·65mm Mauser	·271	·315
303 British	·271	·315
7·7 Jap	·271	·315
32 Win Spc.	·271	·323
8mm Mauser	·271	·327
8 mm-06	·271	·327
338 Win Mag.	·301	·341
340 Wea Mag.	·301	·341
348 Win	·311	·351
35 Rem.	·311	·361
350 Rem Mag.	·311	·361
35 Whelen	·311	·361
358 Norma Mag	·311	·361
375 H&H Mag	·338	·3785
378 Wea Mag.	·338	·3785
44 Rem Mag	·390	·430
444 Marlin	·390	·430
45-70	·420	·4605
458 Win Mag.	·420	·4605
460 Wea Mag.	·420	·4605
PISTOLS AND REVOLVERS		
CALIBER	BORE DRILL	CUTTER
22 LR	·185	·2255
25 ACP	·220	·2525
30 Luger	·271	·313
32 ACP	·271	·314
32-20	·271	·314
32 SW	·271	·315
357 Mag.	·311	·360
38 Spec.	·311	·360

CALIBER	BORE DRILL	CUTTER
38 SW	·311	·364
38 ACP	·311	·359
380	·311	·359
9mm	·311	·357
38-40	·364	·404
41	·364	·404
44-40	·390	·430
44 Russian	·390	·430
44 Spec.	·390	·430
45 ACP	·420	·454
45 Colt	·420	·455
455 Web.	·420	·458

CHAPTER 11 THREAD CUTTING

TEST BAR

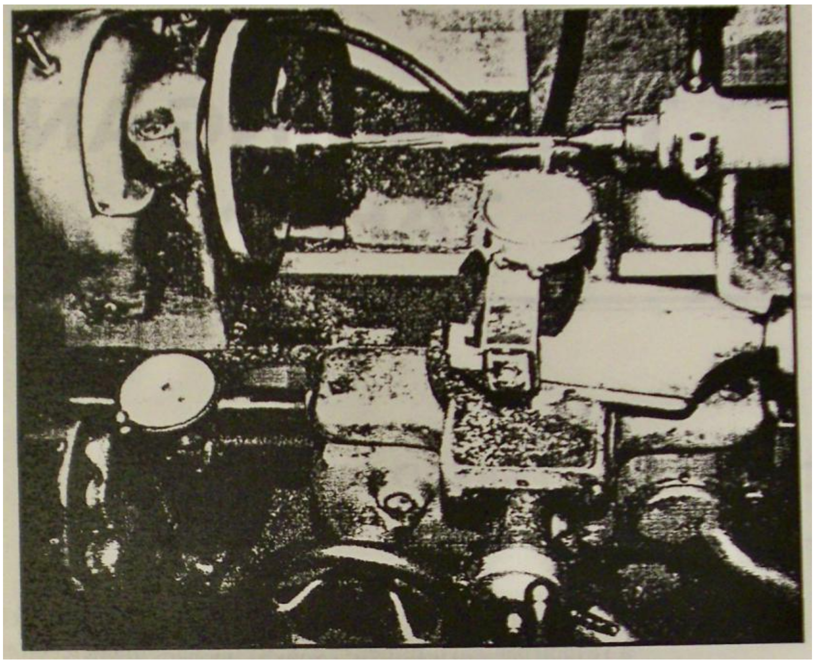
Before you start the threading operation, set the tail stock back to 000 using a 18 inch bar that is turned to exactly the same diameter on each end. To make this bar, get a 1 inch bar 18 inches long, center it and set it up between centers.

Make a light pass and check both ends to see if they measure the same. If not, adjust the tailstock and make another pass. Repeat the above operations until the bar measures the same on both ends.

This bar, you save as you will be using it again each time you true up the tailstock. Once you have the bar completed, all that is necessary is to put it between centers. Clamp a dial indicator to the carriage on the lathe.

With the plunger of the indicator on the bar, start from the headstock end (without the lathe being turned on) and move the carriage to the tail stock end. If there is any difference in size, adjust the tail stock and repeat until the dial indicator reads the same on each end.

In order to produce an American Standard thread on the engine lathe, the ratio of tool feed to workpiece speed must be adjusted. The width of the crest will correspond with the width of the tool's nose when the thread is cut to its full depth. Thus, the root and crest are the same width.



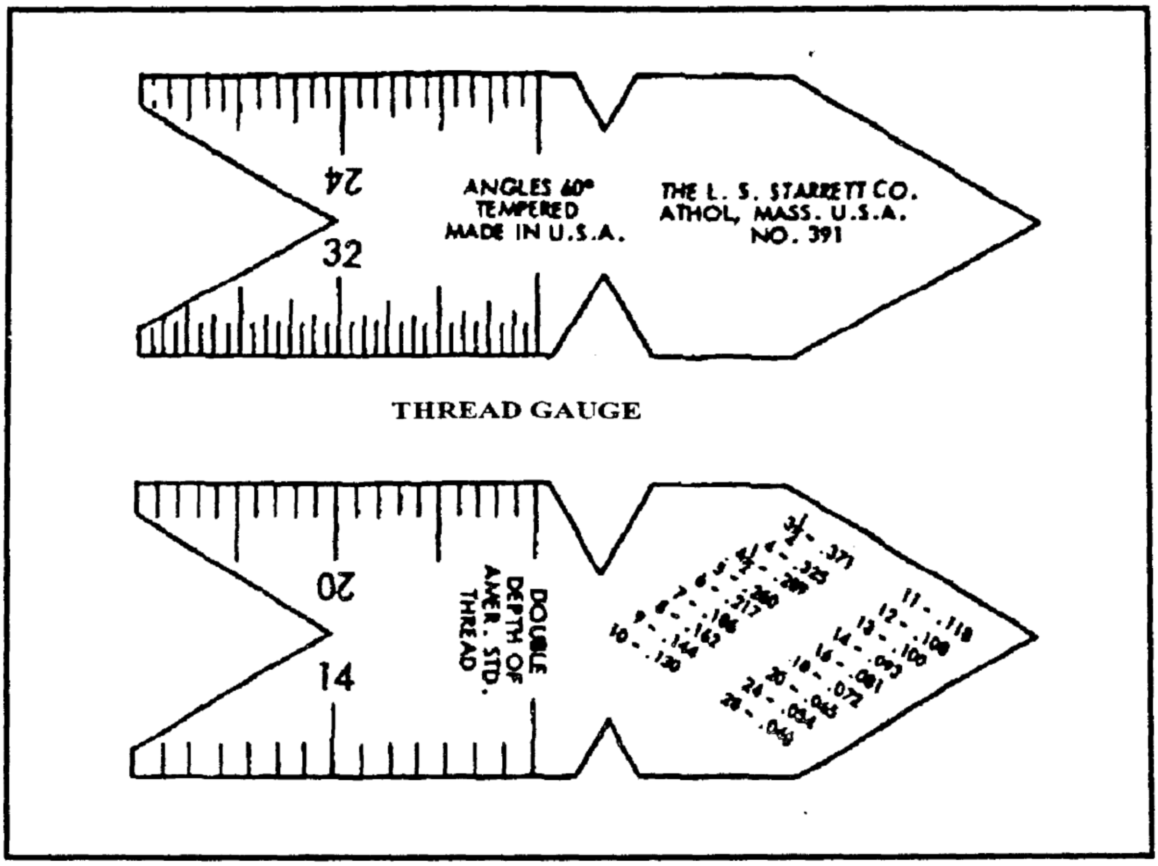
SETTING THE TAIL STOCK

THREAD-CUTTING TOOLS AND THREAD FORMS

For each of the types of thread a tool bit to form that particular thread must be ground. Since the form of thread is the end product, the grinding of the tool bit must take into consideration the clearance angle side rake form and other factors to achieve the result desired.

CLEARANCE

Because of the rapid advance of the cutting tool across the workpiece* the clearance angle is an important factor. Clean, accurate threads are impossible unless both sides and front of the tool are given enough clearance to permit the tool to move freely in the groove being formed. When the tool is fed into the work at an angle, as for form threads, the tool should have 3 to 5° of side clearance.



Threading Point Gauge

V thread tools are ground flat across the top, with about a 5-degree side-clearance angle. If the top of the tool is at an angle, the thread will be incorrect.

CENTER GAGE FOR CHECKING CUTTING TOOLS

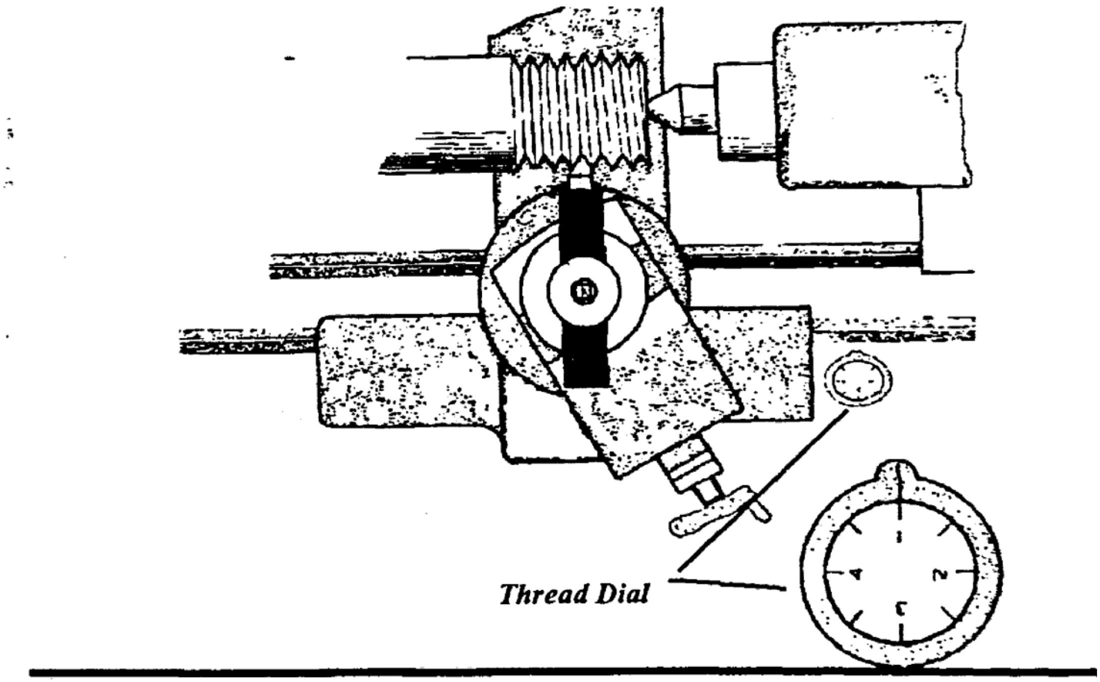
The center gage is a tool used for checking the points of cutter bits for the American standard screw threads that must be ground to an angle of 60°. The 60-degree included angle on one end of the gage is used when grinding the tool to the exact angle. This same tool is useful in checking the angle of lathe centers.

The V on the edge of the gage is used for setting the tool in the tool post of the lathe. The center line of the V-point is perpendicular to the axis of rotation of the cylinder to be threaded. This setting will give the correct thread form.

The two opposite sides of the gage are parallel and have graduations on the edges for checking the number of threads per inch.

One side of the gage at the point is engraved to show the sizes of tap drills for 60-degree V-threads (internal threads), and to show, in thousandths of an inch, the double depth of commonly used threads.

The 60-degree angle at the point is useful to gage the thread after it is cut. A single-point cutting form tool corresponding to the contour of one desired thread space is used to accomplish thread chasing. A thread-cutting tool is set on center and adjusted with a center gage so that the thread will not lean. This adjustment will ensure that the center line of each thread will be perpendicular to the rotational axis of the workpiece.



When thread chasing, a number of successive light helical cuts is taken around the cylindrical surface to be threaded. Depth of cut for the first cut may be about 0·05 inch, depending upon the thread size. This becomes progressively smaller for succeeding cuts, and it should be about from 0·0005 to 0·001 inch for the last cut. Usually the cutting tool is moved in for depth of cut with the use of the compound rest that is set at an angle.

This causes the majority of the metal to be cut on one side of the cutting tool. This in turn allows the chip to curl more easily, and positive rake, or other desired rake, can be provided for the cutting edge on the side where the majority of metal removal occurs. The cutting edge at the right side of the cutting tool is made to cut lightly by setting the compound rest to 29° instead of to 30°.

THREAD CUTTING IN A LATHE

Threads that must be accurate with some turned part must be turned in the lathe at the same setting used to turn the work. For this work, a proper cutting tool is set up in the tool post, and the lead screw is geared to the head-stock spindle to give the required lead to the thread.

Cutting threads on the engine lathe is usually done when only a few threads are to be cut or when special forms of threads are necessary. For the operation, the lead screw is revolved at a desired ratio with the spindle of the lathe through a series of gears. Quick change gear boxes enable the operator to produce various pitches of threads, using control levers.

In the thread-cutting operation, the lathe should be set at the correct ratio of feed to speed for the desired thread to be cut.

Next, the compound is adjusted so the micrometer dial on its collar is at zero. The tool is then brought into contact with the workpiece by adjusting the crossslide and setting its micrometer dial to zero. All adjustments for depth of cut can be made from these settings.

It is common practice to use both the cross-slide, and the compound. The tool is backed off the workpiece and the carriage is moved to where the tool is, at a point beyond the end of the workpiece. The cross-slide is then advanced until the micrometer dial reads the same as where the tool was touching the workpiece.

Next, the compound is advanced ·002 to ·003″ and a trial cut is taken. At the end of the cut, the cross-slide is backed off and the tool returned to its starting point. The cross-slide is then adjusted to its zero reading and the compound advanced a distance equal to the next cut. The operation is repeated until the proper depth of thread is obtained.

The gears to use for each thread are shown on a plate attached to the lathe by the maker. You should know how to gear the lathe without the plate. There is a gear of a known number of teeth permanently attached to the head-stock spindle. -

If the lead screw can be made to advance the cutting tool one thread-space, or thread-pitch, per revolution of the work, the result will be a thread of that pitch.

The rule for finding the ratio of teeth in the lead-screw gear to the teeth in the stud gear is to multiply the pitch of the thread to be cut by the teeth in the stud gear and divide by the pitch of the lead screw, which will give the number of teeth in the lead-screw gear.

A gear with 40 teeth on the screw will cut a thread with a pitch of 10 threads per inch.

TO CHECK THE GEAR TRAIN

Pitch of lead screw, multiplied by the product of all the driving gears, divided by the product of all the driven gears will give the pitch of the thread to be cut.

With lathes equipped with quick-change gears, most common threads can be set up by moving the required gears. When it is necessary to cut a thread of an odd pitch, the train of gears between the stud gear and the quick change-gear box must be changed. Metric threads can be cut on a lathe having a lead screw with the pitch in inches by using the ratio of 50 on the stud to 127 on the screw. This is because there are 127 centimeters in 50 inches.

The cutting tool is ground to the shape required for the form of screw thread being cut. For cutting 60° V threads, a center gage is used for checking the angle when grinding the tool to shape.

In cutting a right-hand exterior thread, the compound is turned in the direction of the headstock and set at an angle of 29°.

The tool is set so its center line is at a right angle to the axis of the workpiece. This setting can be obtained by the use of the center gage as shown.

When the tool point fits uniformly into the v notch of the gage, the tool is at a 90° angle.

NOTE: The point of the tool should be at the same elevation as the center line of the workpiece.

The compound slide is set to an angle of 60°, and the tool is set square with the work, using the "V" notch of the thread gauge to set the tool. The point of the tool must be at the same height as the lathe centers. The tool is run up to the work with the cross feed, and the cross-feed stop is set to always bring the cross feed back to the same position after backing out the tool to return for another cut. -

The compound slide is used to feed the tool into the work. By feeding the tool on the 60° angle to which the compound slide is set, the tool cuts on one side only, and it can be given a side rake to make the chip clear the thread groove. If the tool is fed in square with the work, it must cut on both sides. No side rake can be used, and the two opposing chips will interfere and jam in the cut.

The carriage is attached to the feed screw by closing the half-nuts. There is a safety interlock between the friction feed for turning and the half-nuts for thread cutting, so the two cannot be engaged simultaneously, which would wreck something.

At the end of each cut, the half-nuts are opened, and the tool is withdrawn from the cut, so the carriage can be returned to the start for another cut. If the tool is not withdrawn from the cut, the backlash of the feed gears would leave the tool out of line with the thread and if the lathe was reversed, the tool would damage the thread.

If your lathe is not equipped with a thread cutting dial, you must reverse the lathe to return the tool to the start for another cut. Without the thread dial, the half-nuts cannot be opened until the thread is completed,

The thread-cutting dial indicator is a dial geared to the lead screw. When the carriage is stationary, the dial revolves, but when the carriage is cutting a thread, the dial is still. There are several graduations on the dial, each numbered. As the dial revolves, the half-nuts are closed when the correct number comes up to the index mark. For most even numbered threads, there are several places on the dial that can be used to close the half nuts. For odd- numbered threads there is only one position, and the half-nuts must always be closed on the same number used to start the first cut.

After making the first cut return the tool until it nearly touches but does not cut and try the several numbers on the dial to see which ones track the tool in the first cut. Then any of these can be used for the rest of the job.

In starting the first cut, close the half-nuts on the number 1 line of the dial and feed the tool with the compound until the tool just scratches a fine line, indicating the thread. Shut down the lathe and test this line with the thread- pitch gauge to see that the lathe is cutting correctly. The cross feed of the carriage must always be up tight to the cross-feed stop before moving the tool with the compound feed.

At the end of the cut, if the thread just fades out, it is necessary to withdraw the tool while the feed is still on so that the thread will finish with a taper. To do this requires working at slow speed, and it helps to put a chalk mark on the work so the work so the tool can be backed out at the same place each revolution.

When the thread finishes close to a shoulder, turn a small groove next to the shoulder as deep as the bottom of the thread so the tool can finish in this groove. Leaving a complete thread up to the shoulder so the nut will tighten up tight to the shoulder. It is wise to set the carriage stop so the tool cannot run into the shoulder. The cutting tool should be ground so the point will be in the groove while the side clears the shoulder.

When it is not necessary that the nut ran up tight against a shoulder, the tool can cut its own finishing groove if you open the half nuts at the same place each time.

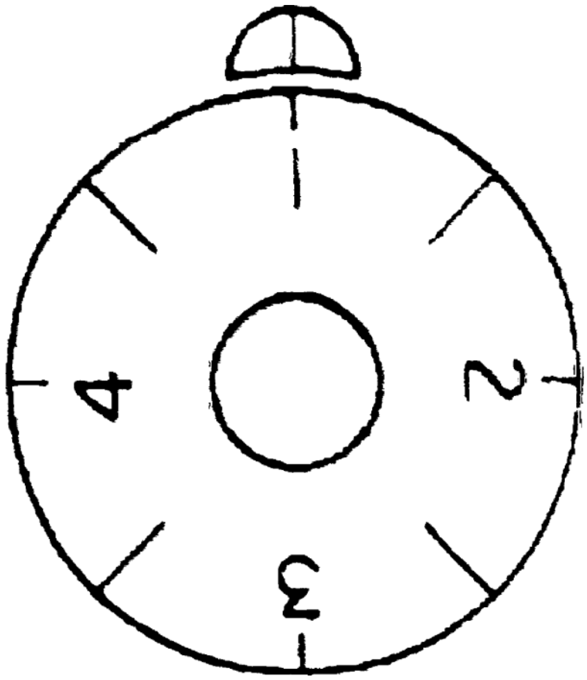
To use successfully any of these methods of finishing. a cut. you must work quickly with both hands, back the tool out with one hand while you open the half-nuts with the other, When you return the tool for another cut advance the compound slide by the amount of the chip. Never change the setting of the cross-feed stop after you have starred to cut a thread or you will throw- the tool out of alignment with previous cuts.

Since the tool is traveling on the 60° angle of the compound, you must divide half the DD figure by the sine of 60°, or ·866; the resulting figure will be the amount by which the tool is to be advanced on the compound slide.

Since there is always some springiness or backlash in both the cutting tool and the work, you will have to take several finishing cuts with the tool at its final setting, without advancing it any deeper, so It can work out the oversize cutting when possible, keep handy a nut of the required size that has an accurate thread, and use this to test the new thread.

If the work is driven by a lathe dog while on centers, be sure the slot in the driving plate is marked so the dog will be replaced in the same slot each time after removal for testing.

Use plenty of cutting oil when cutting steel. Brass and cast iron can be cut dry, but aluminum tends to stick to the tool and leave a rough surface unless a little oil is used.



THREAD DIAL

THREAD DIAL INDICATOR

Most modem lathes are equipped with a thread dial indicator, which saves time when cutting long screw threads. The thread-chasing dial is an indicator with a revolving dial. It is either fastened to the carriage of the lathe or built into it. The dial of the indicator, serves as a guide to the operator. It tells him when to close the half-nut in the apron onto the lead screw so he may take successive cuts in the same groove, or to space grooves properly for multiple threads.

When this is done, the lead screw and carriage bear the same relative positions as before. In other words, when the operator engages the half-nut as the proper mark on the dial comes into position, the threading tool moves into the same groove made on previous cuts.

The face of the dial is numbered to show positions at which the half nuts may be engaged. When the lathe is set up for cutting screw threads, the thread dial shows the relative position of the lead screw, spindle, and carriage of the lathe.

This permits disengaging the half nuts from the lead screw at the end of the cut, returning the carriage quickly to the starting point by hand, and reengaging the half nuts with the lead screw at a point that will assure that the tool follows exactly in the original cut.

The position at which the half nut should be closed depends upon the pitch of thread.

For odd-numbered threads, close half nuts on any numbered line, such as 3½ threads per inch, the half-nut must be engaged at any odd-numbered line on the dial.

For quarter-threads, such as 2¼ or 3¼ threads per inch, the half-nut must be engaged at the same point on the dial each time a cut is started.

For all numbered threads, close half nuts upon any line on the dial.

For threads involving half threads-any numbered line.

For threads involving quarter threads-return to the original starting point before closing half nuts.

The dial is engaged to the lead screw, as the operator desires, by a feed lever on the apron. This lever is called a half-nut lever. When the lathe is running but the half-nut lever is not engaged, the dial revolves. When the half-nut lever is engaged, the carriage moves but the dial remains motionless.

Before you can start the thread-cutting operation, you must decide, from the number of threads required per inch, at what point on the dial to engage the half-nut. For chasing all even numbers of threads per inch, such as 4, 6, 8, 10, etc., the half-nut is engaged for the first, and for all successive cuts, at any of the eight graduation marks on the face of the dial.

For an odd number of threads per inch, engage the half nut at any quarter-turn or numbered line on the dial (the main graduation marks.)

SETTING THE COMPOUND FOR RIGHT AND LEFT-HAND THREADS

To cut right-hand external threads on the lathe the compound rest is turned at a 29-degree angle. This prevents tearing of the thread and makes it easier to rechase the thread if the tool must be reset. The carriage is made to travel from right to left or, toward the headstock.

Before each successive cut, the tool is fed in with the compound rest. To cut a left-hand thread, the compound is turned at a 29-degree angle toward the headstock, and the carriage is made to travel from left to right.

SETTING THREADING TOOL

The threading tool is mounted in the tool post. Adjust the cutter point vertically to the exact center of the work. Then place a center gage with its back edge in contact with, or parallel to, the work or the tailstock spindle.

Now adjust the tool horizontally by fitting the cutter point exactly into the 60 degree angle notch in the front edge of the center gage. Tighten the tool post screw. Be sure not to change the position of the holder. Recheck the tool setting after tightening the tool post screw.

CUTTING AN EXTERNAL THREADS

After setting the compound rest and positioning the threading tool properly in relation to the work, it is necessary to select the proper speed and feed. Good thread-cutting practice requires that the back gears be engaged for this operation.

This reduces the r. p. m. or speed to a minimum, and is necessary if best results are to be obtained. The correct selection of feed is determined by a gear box on all modem lathes. Directions are shown on the gear box regarding the setting of levers to obtain the correct feed, depending upon the threads per inch to be cut.

Now the compound feed-screw graduated collar is set to zero and the tool point is brought into contact with the work by turning the cross feed screw. The tool point should contact the work lightly. Then run the carriage to the right, using the carriage hand wheel, until the tool clears the end of the work. Notice w hat the setting is on the cross-feed collar.

This adjustment must be remembered so at the end of each cut, the cross-feed is always brought back to the same number or setting. Feed in on the compound ·002″ and then start the machine and take the first trial cut.

If using the thread dial, be sure to engage the half-nut lever at the correct line on the dial, depending upon the threads per inch you are cutting. This causes the carriage to start in motion.

A check should be made after this first trial cut to see that the correct pitch of thread is being machined. This is done by using a thread-pitch gage or a rule.

To decide the total number of thousandths of an inch that the threading tool must be fed in by the compound feed screw, and in order to cut the thread to the desired depth, a simple formula is used. This formula is: divide the constant . ·750 by the number of threads per inch.

If it is required that 8 threads per inch are to be cut, then divide ·750 by 8, which gives a result of ·0937″ (094″ can be used).

This is the total number of thousandths of an inch the tool is to be fed into the work to cut 8 threads per inch. However, the compound feed is used until the tool has been fed in ·090″. Then the cross-feed is used to remove the final four thousandths of an inch from the thread, making our cuts and feeding the tool in ·001″ on each cut. This will help to polish the right side of the thread.

A good grade of lubricant should be used on the tool when threads are being cut. Mineral lard oil is a very good lubricant for threading.

RESETTING THE TOOL

If it is necessary for any reason to remove the tool before the thread is finished, reset the tool to the gage regardless of the part of the thread already cut. Having the compound rest at an angle of 29° makes it easy to reset the tool if it needs regrinding. The tool is clamped in the tool post after it is reground. Then it can be set with the center gage as before.

In resetting the tool, proceed as follows: first, reset the tool to the gage; then back the tool away from the workpiece. Start the machine and engage the thread-chasing lever as before. Let the workpiece make two or three turns and shut off the power with the threading lever still engaged.

Adjust the tool into the thread previously cut by moving the compound and cross-feed until the tool is lined up properly in the thread groove. Back the tool out from the workpiece slightly and turn on the power. When the tool is aligned Properly, proceed as before.

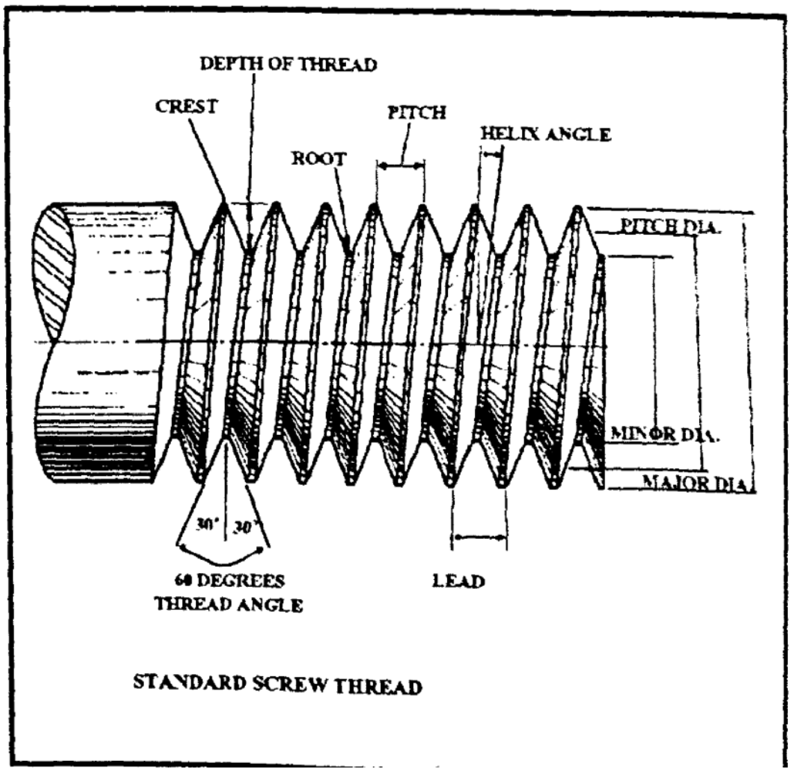
To move in for depth of cut for another type of thread, such as acme, square, and worm threads, the tool is moved in a direction that is perpendicular to the rotational axis of the spindle. The angle of the compound rest is set at 0°. For thread chasing, the lathe carriage is moved at the proper speed in relation to the rotation of the work piece with the use of the lead screw instead of the feed rod.

A method for quickly changing the gearing provides a large number of available leads, or numbers of threads per inch. To take a cut, the carriage is connected to the rotating lead screw with the two halves of the split nut that are fastened to the carriage. When the two halves of the split nut are closed on the lead screw, they act as one complete nut, and the carriage is moved as the lead screw rotates.

Since the lead screw is geared to the spindle, the carriage will move a predetermined distance per revolution of the workpiece. This distance is equal to the lead of the desired thread. A thread dial shows the correct time at which the lead screw should be engaged with the split nut. It is used because after each cut the split nut is disengaged from the lead screw.

The thread dial, mounted on the carriage, is connected to the lead screw with a small worm gear. Whenever the lead screw rotates and the split nut is not engaged, the thread dial rotates. One complete revolution of the thread dial corresponds to the movement of a certain number of threads on the lead screw, such as 24, past the split nut.

With this arrangement it will be possible to engage the split nut at 24 different places on the lead screw in one complete revolution of the thread dial. Eight equally spaced marks alternately numbered and unnumbered around the circumference of the thread dial show when to engage the split nut.



Standard Screw Thread Parts

CUTTING THE THREADS

Set up a V tool bit ground right for threading. This is a 60 degree tool. Set the point of the tool right on center, using the center on the tail stock. The compound should be set at 30°, when you clamp the tool. Set the quick change gears for 12 threads per inch, make sure your feeding dial is engaged, and you are ready to cut the threads.

Set both dials to where the tool is just touching the shank. Set the dials on 000. Set the tool in so you will take a ·005 cut. Have the lathe in back gear, running it at the slowest speed. When the threading dial comes around to 0 engage the feed. It feeds fast, so be ready to disengage the feed and back out the tool in one operation.

When it has completed set the tool in ·010 and when the dial comes back to 0, engage the feed. The carriage is then returned to its starting position for another cut. The threading tool is again moved in with the cross slide so that the cross slide is always returned to the same position for each successive cut.

The depth of cut is adjusted with the compound rest handwheel and graduated dial. Another cut then is taken by engaging the split nut with the lead screw. These successive cuts are continued until the thread reaches its correct depth.

Keep repeating the process until you have gone to the correct depth and make a final clean up pass at the last setting. Use a good cutting oil when cutting threads. Always turn in the compound dial, leave the other at 0, and back this out each time.

If engaged at an incorrect time, the threading tool will not enter the helical groove of the cut previously, produced. Instead it may remove some of the desired threads. After each cut, the split nut is disengaged and the threading tool is withdrawn sufficiently to clear the thread on the workpiece by using the crossslide handwheel and graduated dial.

SIZE OF WORKPIECE BEFORE THREADING

Before the threading operation is begun, the workpiece needs to be turned to the maximum or major diameter of the thread specified on the blueprint or other specifications. This size is usually given as O.D. (outside diameter) expressed in inches and decimal parts of an inch. When cutting V type threads, it is also good practice, to chamfer or bevel the ends of the shaft to be threaded. The chamfer can be at any angle from 30 to 45°. For instructions on straight turning, see the section on that operation.

SETUP FOR CUTTING AN EXTERNAL (V) THREAD ON THE LATHE

The setup for external threading will depend largely upon the shape and size of the workpiece. Threads are often turned on shafts mounted between centers and revolved by a lathe dog.

After this setup is made and the work is turned to the proper outside diameter, the workpiece is left right in the lathe and the threading operation is performed. The jobs are threaded while held in a three or four jaw chuck in which they were mounted to be turned to size.

Due to the pressure exerted by any threading tool, it is necessary, when threading long, narrow shafts, to use a steady rest.

THREAD CUTTING TERMINOLOGY

The following will give you information on the threads so they can be better understood. Thread cutting on the lathe is the most common way to make screws, etc. in the shop. Just about any type of screw, and thread that is needed in the shop, can be made on a lathe.

MAJOR DIAMETER

The major or outside diameter (O.D.) of an external thread is the diameter of the piece on which the thread is cut. LTL is the largest diameter of the thread.

DEPTH OF THREAD

The depth of a thread is the distance from the top or crest of the thread to the root measured vertically.

MINOR DIAMETER

The minor diameter is the smallest diameter of the thread of the screw. LTL is sometimes called the root diameter (R.D.) and can be found by subtracting twice the depth of the thread from the major diameter.

NUMBER OF THREADS PER INCH

The number of threads per inch (N) can be counted by Placing a rule against the threaded part, and counting the threads in one inch. The first thread is not counted since, in reality, not the crests but the spaces between the crests are what is being counted. A second method is to use a screw pitch gage. This method is more suitable for checking the finer pitches of screw threads.

PITCH

The pitch (P) of a thread is the distance from a point on a screw thread to a corresponding point on the next thread, measured parallel to the work axis. The pitch of a thread in inches can be found by dividing the whole number i by the number of threads per inch (N).

PITCH DIAMETER

On a screw thread, the pitch diameter is that of an imaginary cylinder. The outer surface of this cylinder would pass through the threads at such points as to make equal the width of the threads and the width of the spaces. On a 60 degree V type thread and on National form threads, the pitch diameter can be found by subtracting the single depth of the thread from the major diameter of the thread.

LEAD

The lead of a thread is the distance a screw will advance into a nut in one complete revolution. The lead is the same as the pitch on a single-thread screw.

The lead for a screw with 9 threads per inch is ·1111, which is the same as the pitch in inches (·1111).

On a double-thread screw; the lead is twice the pitch. On a screw with 9 threads per inch with a double-thread screw the lead would be doubled or ·2222. On a triple-thread screw, the lead is three times the pitch, and so on.

SINGLE SCREW THREADS

Most screw threads are single. The single screw thread has a single ridge and groove.

ANGLE OF THREAD

The thread angle is the angle included between the sides of the thread. The thread angle of the Unified National Form is 60°.

FORMS OF SCREW THREADS

Screw threads are so widely used to connect parts that they are of prime importance when produced in the machine shop.

Great amounts of study and research have been devoted to the standardization of screw thread forms.

The basic form of the screw and nut, with a 60 degree angle of thread 55. The crest of the thread may be flat or rounded. The flat crest is preferred in American practice, and the rounded crest is given preference in British practice.

Over many years, different screw thread forms and standards have been adopted in the United States. These different forms originated chiefly because of special requirements or because they were considered superior to other forms. In addition to the American Standard thread, they are the Acme, Square, and the 29° Worm threads.

AMERICAN STANDARD THREAD FORM

The American Standard thread form is used as the standard locking thread form in the United States. This thread form is used on practically all mating parts in modem machine construction. The Unified thread form is essentially identical to the former standard, the American National thread form. The two forms are interchangeable for most diameter-pitch combinations. A 1-64 National Coarse thread is interchangeable with a 1-64 Unified National Coarse thread.
Unified National Coarse (UNC)
Unified National Fine (UNF)
Unified National Extra-Fine (UNEF)
Unified National 8-Pitch (8 UN)
Unified National 12-Pitch (12 UN):
Unified National 16-Pitch (16 UN)

In the coarse, fine, and extra-fine series, the number of threads per inch increases as the diameters decrease. The coarse and fine threads are widely employed for general use.

THREAD FITS

There are three classes of fit designated by American Standards now being used. Classes 1A, 2A, and 3A apply to external threads only, and class 1B, 2B, and 3B apply to internal threads only. The three classes of fit are:

Class 1. Loose Fit
Class 2. Medium Fit
Class 3. Close Fit

LOOSE FIT

This class possesses the largest allowance and is used where rapid assembly of parts is required and looseness is not objectionable.

MEDIUM FIT

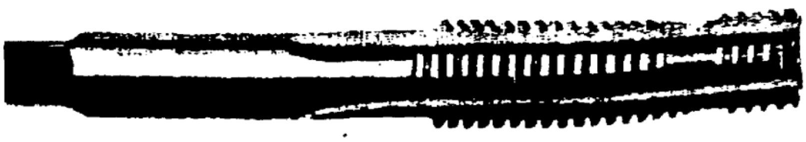
This is used on the bulk of standard screws, bolts and nuts. A very small amount of looseness or shake may be present or if the parts are carefully made no movement can be noted, yet the nut can he screwed on by hand.

CLOSE FIT

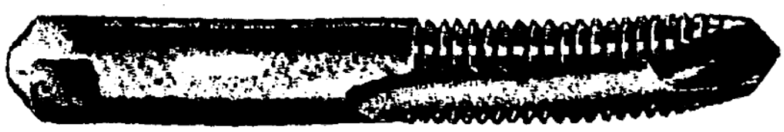
This is used on fasteners where accuracy of fit is highly important, and where no looseness is permitted. A wrench or another tool must be used to force the nut onto the bolt.

Tighter fits than those mentioned require special fabrication, and are carefully specified for the job being assembled.

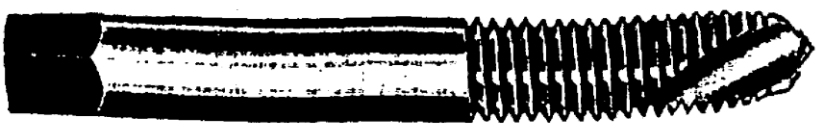
TAPS AND THREADING



ACME TAP



PLUG GUN TAP



GUN FLUTE PLUG TAP

Different Types Of Taps

The tap is used to cut internal threads. There are many forms of taps used in manufacturing work, but the hand tap is the only one used in a home workshop. Three taps are used for each size of thread. The Taper Tap is used to start the thread. This has a long taper ground on the point that will enter the drilled hole and act as a pilot to guide and center the tap. Even with this self-guiding feature, you must hold it in line with the hole or it will start crooked.

The tapered end will try to get it back to the center of the hole, and this will bend the tap and is liable to break it.

Due to the long taper of the starting, or taper, tap, it cannot reach very far into the hole. A tap with a shorter taper, called a Plug Tap, is used after the thread has been started with the taper tap.

It is possible to start a thread with a plug tap by carefully guiding it and using sufficient pressure. It is not possible to start a thread with a bottom tap. If your budget does not permit the purchasing of the three taps, get a plug tap and a bottom tap.

Taps are easily broken and must be handled very carefully. Never use a tap wrench too large for the tap and do not force it as it cuts a chip. After the chip has formed, turn the tap backwards until it breaks the chip. With very small taps, it helps to run the plug tap in as far as it will stand without breaking. Then back it out and cut with the bottom tap until it reaches its limit.

This way, each tap cuts less metal each time and is less liable to be broken by twisting. It takes longer, but is worth the extra time by saving broken taps.

It helps to have two small tap wrenches so the two taps can be left set up in the wrenches. Bending a small tap sideways is one of the easiest ways of breaking it. Turn the tap with the fingertips of one hand and use the other hand to steady the first by resting your wrist on the vise.

When tapping a hole where the tap must line up with a clearance hole, place the two parts together and tap the thread through the clearance hole, which acts as a guide. If the two holes are not in line, the tap will be forced against the tight side of the clearance hole and will cut threads that will allow the screw to enter the threaded lower part.

These part threads cut in the clearance hole may prevent the screw from clamping the two parts tight, and it may be necessary to remove them with a round file.

When a thread must be exactly parallel to a drilled hole, it is best to start the tap in the drill press before the work has been released from the clamps used to drill the hole. Insert the tap in the drill chuck and adjust the chuck so the tap will slip if it is biting too hard. Use a short rod that will fit into the key holes of the chuck to turn it, simultaneously using a little pressure on the drill-press feed to start the tap. The entire thread can be cut this way, which is rather slow or, after it is well started, the work can be removed from the drill press and the thread finished by hand in the vise.

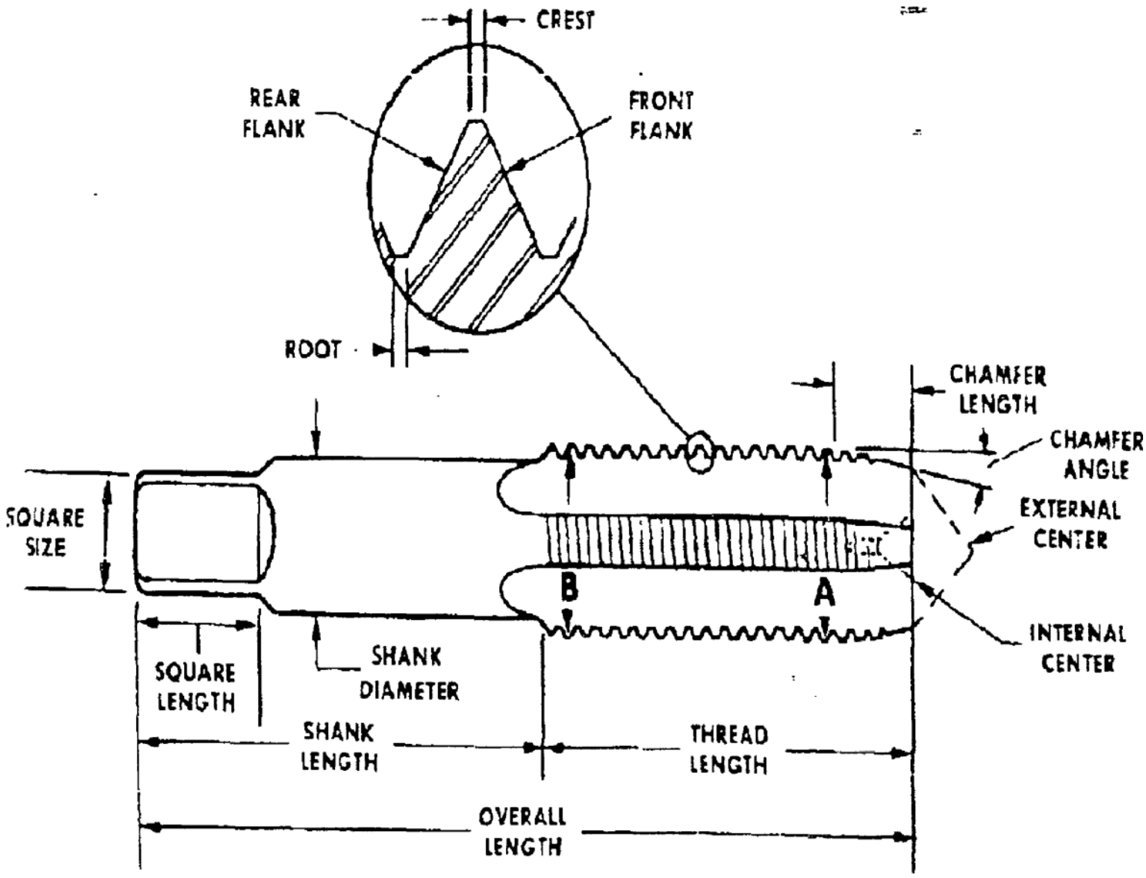
If it was necessary to move the drill-press table to remove the drill and insert the tap, make sure it is returned to the exact position used to drill the hole. Adjust the collar on the column so the table will not drop down and fix the turn indicator to its position, before moving the table.

If there is no turn indicator, make a chalk mark on the column. This same method of starting a tap in a drilled hole applies to holes drilled in the lathe. In this case the tap is placed in the tail-stock chuck, and the work is turned by hand, using a wrench on turn with and feeding with the tail stock.

Larger taps, like pipe taps, cannot be held in the drill-press chuck. These taps generally have a center reamed in the head end, and a 60° stub center can be set up in the drill press to line up the tap. Then the tap is turned with a wrench on its square end. This stub center is a piece of 1/2 inch stock turned to a 60° point. It does not need to be hardened, although it would last longer if it were hard. This stub center is very useful in locating a punch mark under the center of the drill. Set the stub center in the drill chuck and center the work under it before clamping to the table.

Many firms supplying drills have printed cards showing the size of hole to be drilled for taps and the clearance holes for threads. These also show the size of drills and their decimal equivalent.

Get two of these cards, one showing the tap drills and the other the decimal equivalents. Mount them on the wall near the drill press.

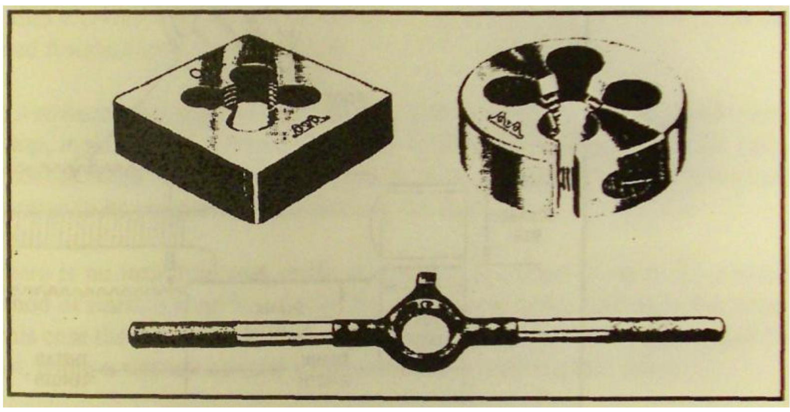


Different Parts Of A Tap

Dies are used to cut external threads. The small button dies of 1 inch diameter are very handy for threads up to 1/2 inch, but for threads over 1/2 inch, larger dies are required. Even the button die of 1 inch for 1/2 inch threads is so narrow that only a few teeth can cut the thread. This size is very useful for rerunning a thread that has been damaged.

Only one die of each size is needed. One side of the die has a taper for starting and is used for cutting the thread. The other side has the full thread clear to the face of the die, and the die can be turned this side to the work to finish the thread up to a shoulder.

It is difficult to start a die square with the work. If it is not started square, it will try to follow its own threads, while simultaneously running off-center to the work, which tries to force it back to center. The result is a very crooked thread. There are die stocks that have three adjustable fingers on the starting side. -



DIE AND DIE WRENCH

These can be set to bear on the work and guide the die square. You should also have a plain die stock to finish threads in close quarters where the stock with the guide is too wide to use.

When you purchase the dies, get the kind that are split on one side so the size can be adjusted. There are a lot of these button dies sold at bargain stores that are not true to size. These can cause a lot of trouble by cutting an over sized thread. Get dies made by one of the recognized, nationally established firms, so you will be sure of getting good ones. It is often required to cut a thread a little under size, say, for a free-running nut, and the adjustable die can do this.

It is also possible to cut an undersize thread by rerunning the thread with the die held at an angle. This makes the die cut some off the side of the previously cut thread, but it will not be a true and even thread. It is to be used only when an adjustable die is not at hand and never where accurate threads are required.

For cutting threads in the lathe with these button dies, use a die holder that fits in the tail stock. This has a guide portion on which the die can slide as it feeds onto the work. The thread is started by putting pressure on the die with the tail stock.

After it is started, the die will feed itself. The lathe can be run under power, using the slowest speed of the back gears. On completion of the thread, the lathe is reversed and backed off the die.

When backing off the die, be sure to remove the amount of feed used to start the die, or the die will stop backing off when the free travel is used up and will chew up the threads.

It is best to unclamp the entire tail stock from the bed as you start to back off, you can then be sure the die can work all the way off without doing any damage. A little sulphur-base cutting oil should be used on both taps and dies. With brass and aluminum, the taps and dies tend to stick to the metal unless oil is used.

CHAPTER 12 THE TOOL GRINDER

If you need to do tool making in the shop, you will need a tool post grinder. It is a simple matter to make a small grinder for small tools. You must do a little looking around in some appliance shops to find a motor that will run at a speed of 15,000 to 20,000 RPM. You can find new high speed electric motors sold by w. w. Granger. Most large cities have a store, or an appliance store will be able to give you an address of the nearest store.

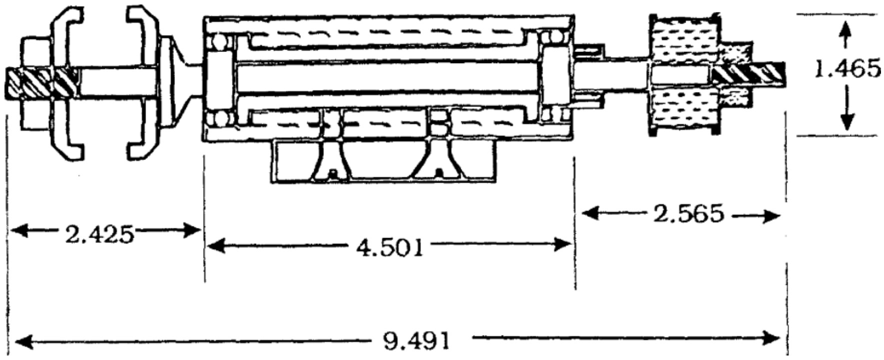
Most vacuum cleaner motors will run at this speed. It will require a bit of work to cut down the housing to be able to make one work.

There are several types of small motors that will run at this speed, and they will mount directly to the base with very little alternations.

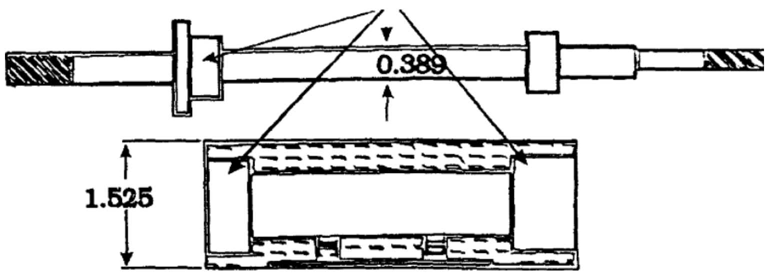
BEARING HOUSING

The bearing housing is made from a piece of mild steel shafting, bored and turned as showed. You might be able to find a ready built mandrel from some . bearing supply business, or a tool supplier.

This piece cannot be held in a chuck while boring, as it projects to far out, so hold the head end in the three jaw chuck and support the tail end in the steady rest. This means that the ends must be centered and the outside finished before you bore and finish the inside. The recess for the bearings should be a good push fit on the outer races.



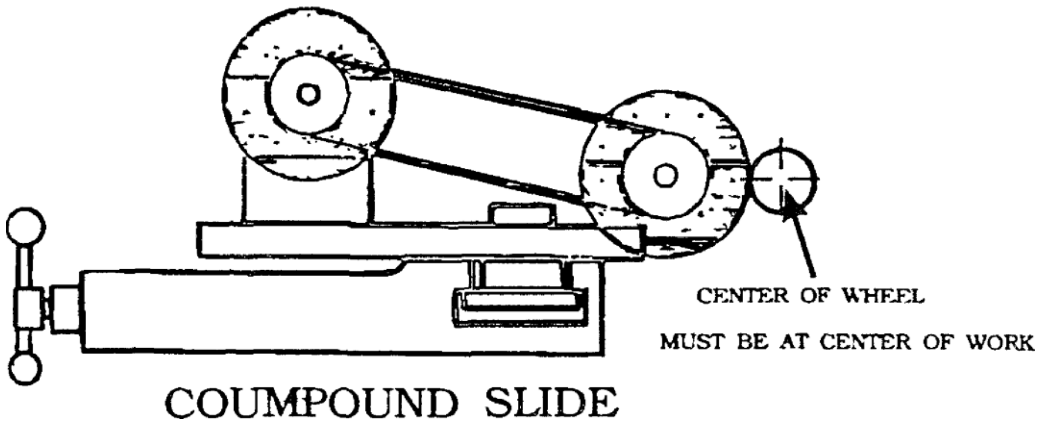
MAKE SHAFT SIZES TO FIT BEARINGS

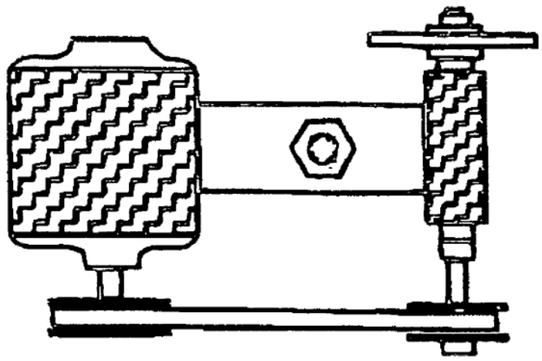


SPINDLE FOR SMALL GRINDER
Tool Grinder Arbor

PULLYS

The pulleys are made from mild steel. Since you need to use a flat belt, they should have a crown turned on the face under the belt. A flat belt always tries to run on the highest place on the pulley. If there is a crown in the center, making it the highest place, the belt will run there and not need the flanges on the sides.





TOOL POST GRINDER

Shop Made Tool Post Grinder

These are only to help if the belt stretches out of shape and one edge gets longer than the other, in which case it will run to the side off the crown. When the belt starts to do this, discard it, as it can start to whip at the high speed and cause vibration.

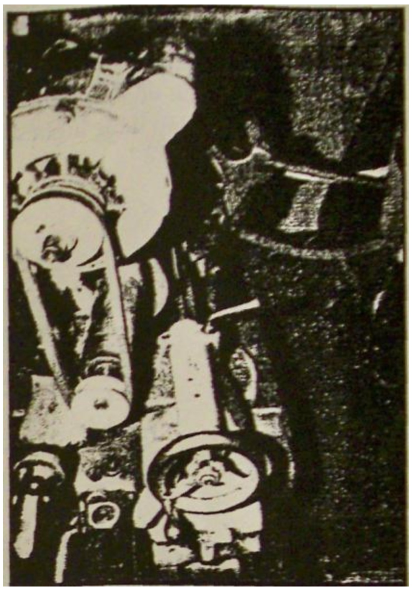
Make a pulley for the spindle, and a similar one for the motor, but fitted to the motor shaft. As these very small mounted grinding wheels must run very fast, you may have to make several sets of pulleys to find the maximum speed you can run it. This will be limited by the belt action.

Because of the high speed required, it is not possible to use a belt of much weight, such as a “V” belt. Such a belt would stand out away from the pulleys, due to centrifugal force, and would not only do very little work but would be apt to start whipping. The small flat rubber belts, made for some floor polishing machines, work very well at this speed. Since the total length of the belt is about 16 inches, you should get one before making the base of the grinder.

Then, when the pulley diameters are known, the distance between pulley centers can be worked out. This will govern the length of the 7/16 inch slot and the position of the bolt to the “T” slot of the compound slide.

Make up two collars with parallel faces to grip the grinding wheel. After boring and reaming the hole, mount them on a mandrel and finish both faces at the same setting. Use very light cuts so that the collars will not slip on the mandrel. The face of the collar that bears on the grinding wheel is relieved so that pressure on the wheel is concentrated well out on the wheel.

A layer of onion skin paper should be placed between wheel and collar to equalize the pressure. If the collars are not relieved, one collar may put pressure on the wheel near the shaft while the other bears near the rim. This could put a strain on the wheel and damage it.



GRINDING THE TOOL

CHAPTER 13 HEAT-TREAT FURNACE

Building a small heat-treat furnace for the shop. This furnace is made from . standard fire brick, 2½ x 4½ x 8 inches. Fire brick does vary in size, so the size shown in the drawing may have to be changed to fit your needs. Another style of furnace is where the main opening is at the top, and the part hangs down on the inside. This seems to give a more uniform heating.

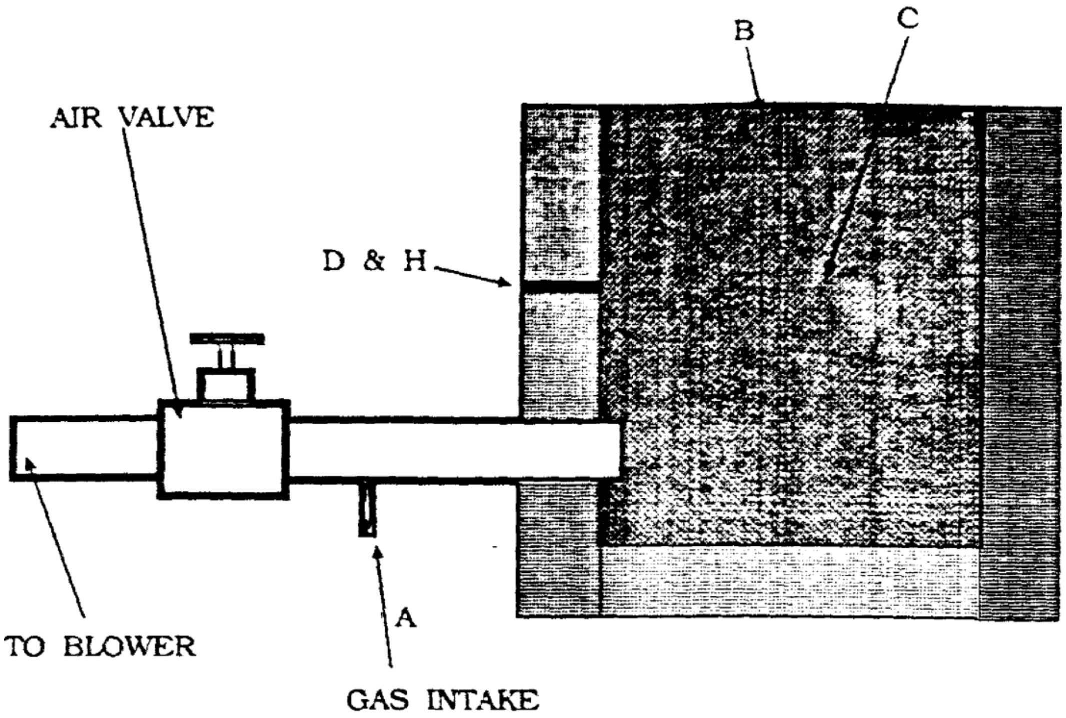
A-Gas/Air outlet; B-Furnace opening; C-Side view; D-Opening for temperature sensor; E-Blower; F-Gas intake; G-Heat sensor pipe; H-Heat sensor opening;

You will need 8 firebricks to make this. When completed you will have a furnace with an inside size of 4½ x 3 x 8 inches. This size will handle just about any job in the shop.

FIRE BRICK

The fire brick is mortared together with the cement used in fireplaces, and it should not be any problem to get from any business that sells fireplaces or wood burning stoves. The brick is placed 2, flat side by side, mortared, and then one brick on each side is stood on edge, (see drawing) mortared to the two laying flat.

Two more are then placed on top of the first two. To finish it off, two more are mortared and set on edge on one end, as per drawing. Let it set for 24 hours before starting any more work on it.



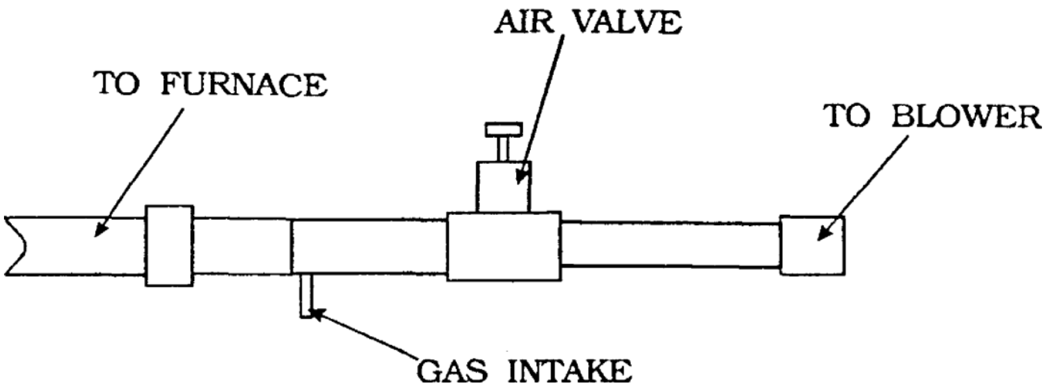
FURNACE AND GAS VALVE

Gas Furnace

Next you must get a cement drill 1½ inch, and drill one hole 1/4 inch above the base in the center on the side. This is where the 1 inch pipe from the blower will be inserted.

You will need to get your pipe cut and threaded to the correct length. Connect a 2 inch pipe to the 1 inch pipe, this will go to the blower.

Next you will need to cover the fire brick on the outside. Cut 1/8 inch metal to fit to the outside and top and weld this together, to form a box for the fire brick. Be sure to have a metal bottom already cut out to put under the brick so the sides can be welded to it.



FURNACE TORCH VALVE

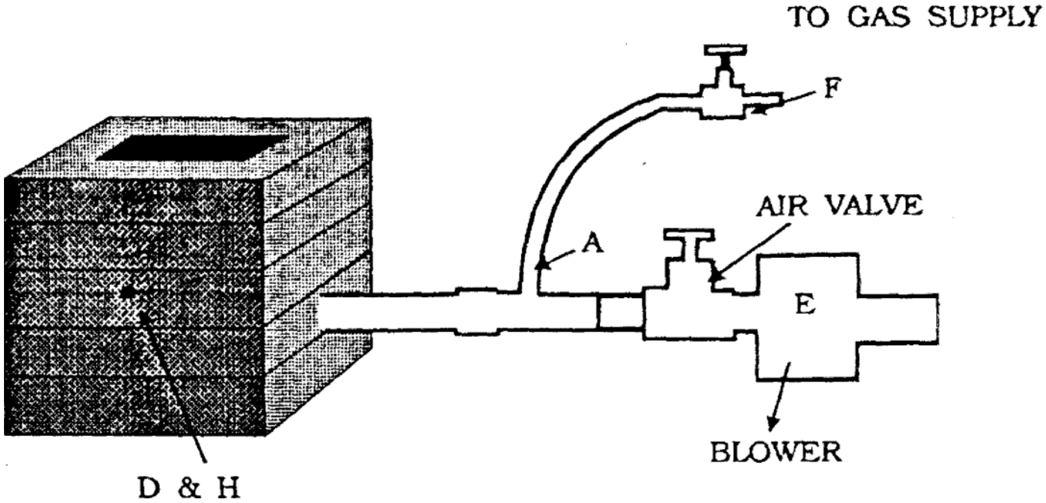
Gas Metering Valve

You can when building this go with the opening in the top, and then you need not make a door for the front. This is the simplest way to go and I believe the best. You hang the part down through the opening in the top. At the front there is a small opening for the temperature probe. This is a 1/4 or 3/8 inch hole.

FRONT LOADING FURNACE

If you decide to build a front loading furnace you will need a 2 inch opening in the top of the furnace for the gas. Also you must build a metal door for the front and line it fire brick. You now will need to find a used Kirby vacuum cleaner blower, or any other type of high speed blower to provide the air for the furnace.

A shop Vacuum blower also works great. You will need to find an adjustable rheostat to control the speed of the blower. The speed that it runs would make it impossible to light the oven. The final step is to tap into the air line with a line from your gas supply. .



COMPLETED HEAT-TREAT FURNACE

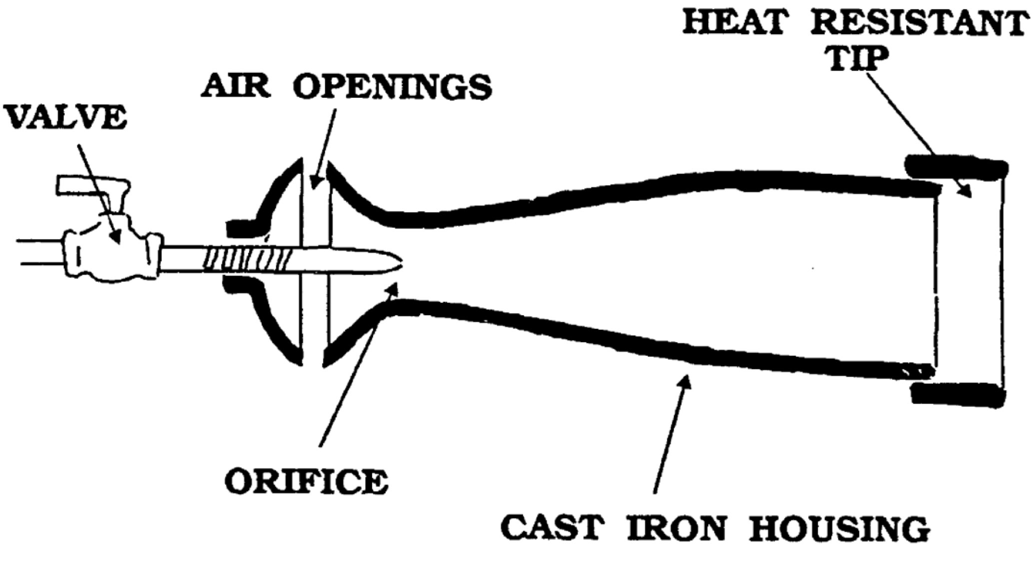
Top Loading Furnace

The basic tools for heat-treating is the gas torch. The torch is a simple apparatus consisting of a mixing tube into which fuel gas and a blast of air are introduced to be mixed and burned at the end of the tube. An old vacuum cleaner can be used for the air blast.

The hose used for the various attachments for the cleaner can be used to deliver the blast of air to the torch. You must make a fitting for attaching the hose at the dust-bag outlet and arrange the cleaner so that air can enter at the suction end.

At the torch, a gate of sheet metal is arranged to regulate the amount of air entering the mixing tube. You can also reduce the air flow by reducing the speed of the blower. A simple light dimmer found at any hardware store, will do the job very good if you have a DC blower motor.

The fuel is supplied through a rubber tube of a size to fit the supply pipe and the fitting on the torch. You should have a 1/4 inch needle valve to adjust the gas to air mixture. Using Butane or Propane from a 5 gallon bottle works better as there may not be enough pressure from natural gas. If you use butane, make sure all your fittings, and hose is for high pressure.



PROPANE BURNER

Propane Gas Burner

It has rubber ends that will fit on the gas nipples at each end. Any other good rubber tube can be used, such as a 3/8 inch garden hose. The mixing tube must be long enough so that the gas and air are thoroughly mixed by the time it gets to the burner end.

Artificial gas will bum at the end of a plain mixing tube, but for natural gas there must be a special tip on the burner end to maintain the flame, or else the air blast will snuff it out. This special tip consists of a jacket fitted around the end of the mixing tube with several small holes drilled into the mixing tube. This gives a low-velocity supply of gas and air to the jacket.

This will maintain a small circular flame around the end of the mixing tube, which will keep the mixture ignited as it comes out of the end of the main burner tube.

STARTING THE FURNACE

The air blast tends to blow the main flame so far away from the end of the mixing tube that it will mix with so much outside air that it will no longer be a combustible mixture and will be snuffed out. This annular ring of low velocity flame surrounding the outlet of the mixing tube will keep the main flame ignited unless so strong an air blast is used that the entire flame is blown away from the end of the mixing tube.

When you start up the burner, shut off the gas until you have the air adjusted, and then slowly turn on the gas, while holding a lit Butane torch over the opening of the furnace. It is best to reduce the air blast until the gas is ignited and then slowly open it until the desired flame is obtained. The flame should bum with a firm blue center cone, and the hottest spot will be at the tip of the blue cone. A yellow flame is not as hot and is very sooty. After the bricks of the furnace have become well heated, the air blast may be opened a little farther, and the blast will be increased.

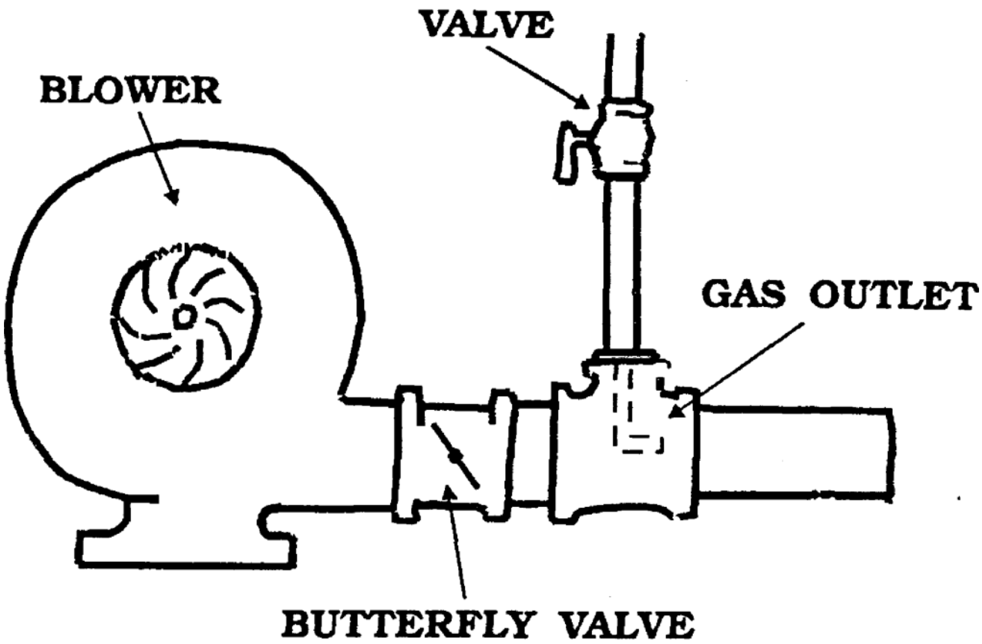
The flame from this torch is very hot and will heat steel to a white heat for forging, but it is not hot enough for welding, however a supply of oxygen for the air intake will increase the temperature.

Get a large enough cement drill to drill a hole through the fire brick about 2 inches from the top. The heat sensor will fit in there. I find a short length of iron pipe, that is large enough on the ID to allow the sensor to fit inside. This pipe is then mortared in place. See Drawing.

A High Temperature gauge can be purchase from an industrial supply dealer. If you can find one, a 2000 degree gauge is best.

In the air intake pipe (A) you will need some kind of manually controlled valve. You must be able to shut the air down quite a bit. If you tried to use the full amount of air, it would blow out the flame.

When you are ready to start the furnace, hook up the gas, check for leaks with soap, and if OK you are ready to fire up.



BLOWER FURNACE

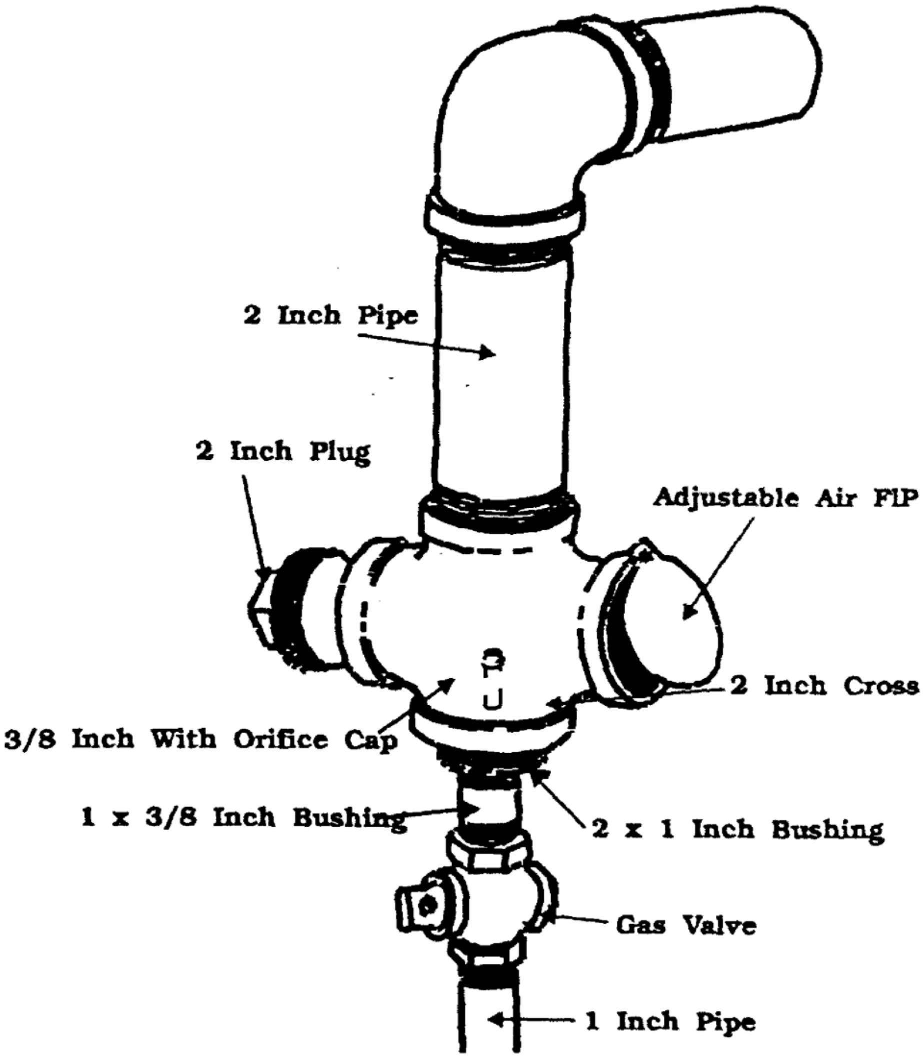
High Pressure Blower

NOTE It is important to do things at first very carefully to prevent an accident This is the way I have my furnace set up, and have had no problems. I have no control over what is done by other people, and cannot accept responsibility for what another person does. If you build a gas furnace you are on your own. Check it outside of the building for safety's sake.

Turn on the air, and close down the valve until there is almost no flow through the pipe. Now take a butane torch and light it. Open the main valve on the gas tank.

BE SURE THAT THE SMALL ADJUSTING VALVE IS CLOSED ALL THE WAY.

Holding the lit torch over the opening, slowly turn on the gas. In a few seconds it should light. There probably won't be much flame, so slowly open the air valve/gas valve until the furnace has a "roar" to it. Don't open the air valve too much as you need a slow heat. Adjust the fuel so that you get 3 to 4 inches of flame from the top. Let it heat up to dry out, and then it is ready to use.



BURNER FOR FURNACE

Gas Air Mixture For Furnace

Once it has cooled down, it can be moved back in the shop. This furnace can be changed to where it is front loading, made larger, or altered for whatever is needed.

This furnace is very fast. Normally it takes only 10 to 15 minutes to get to operating temperature. I find that hanging the knife parts down from the top will cause even heating. You can use 2 bricks to close down the opening more to confine the heat better. When through heat treating, close the top up with the bricks, and let cool down to tempering temperature.

This furnace works great with 01 and other tool steels. It can be used for forging steel as well.

COLORS FOR TEMPERING

430 degrees, Very pale yellow, extra file hard, dies, milling cutters, cut off tools.
440 degrees, Light Yellow, file hard, reamers, thread chasers, fly cutters, hollow mills.
450 degrees, Pale straw yellow, profile cutters for milling machines, rolling dies, knurling tools.
460 degrees, Straw yellow, knife hard, swages.
470 degrees, Deep straw yellow.
480 degrees, Dark yellow, cutting dies.
490 degrees, Yellow brown, extra hard, taps, dies.
500 degrees, Brown yellow, thread dies for general work.
510 degrees, Spotted red brown.
520 degrees, Brown purple, hard.
530 degrees, Light purple.

540 degrees, Light purple.
550 degrees, Dark purple.
560 degrees, Full purple.
570 degrees, Dark blue, half hard.
620 degrees, Blue gray, spring temper.

HIGH TEMPERATURES BY COLOR
752 degrees, Red heat, visible in the dark.
885 degrees, Red heat, visible in the twilight.
975 degrees, Red heat, visible in the daylight.
1077 degrees, Red heat, visible in the sunlight.
1292 degrees, Dark red.
1472 degrees, Dull cherry red.
1652 degrees, Cherry red.
1832 degrees, Bright cherry red.
2015 degrees, Orange red.
2192 degrees, Orange Yellow.
2372 degrees, Yellow white.
2552 degrees, White welding heat.
2732 degrees, Brilliant white.
2912 degrees, Dazzling white, bluish white.

In tempering I let the oven cool down to the temperature that I need to draw the temper for the job that I need, I then put the part in the oven, then let it cool down over night.

APPENDIX

MULTIPLY	BY	TO OBTAIN
Acres	43560	Square feet
Acres	4047	Square meters
Acres	0·0016	Square miles
Acres	4840	Square yards
Acre feet	43560	Cubic feet
Acre feet	1233. 48	Cubic meters
Atmospheres	76·0	Centimeters - mercury
Atmospheres	29·92	Inches - mercury
Atmospheres	14·70	Pounds/in.^{2}
Atmospheres	1·058	Tons/ft.^{2}
Barrels - oil	42	Gallons - oil
Board feet	144	Cubic inches
British Thermal Units	777·6	Foot-pounds
British Thermal Units	3·927 x 104	Horsepower-hours
British Thermal Units	2·928 x 104	Kilowatt-hours
Btu/min	12·96	Foot-pounds/s
Btu/min	0·0236	Horsepower
Btu/min	17·57	Watts
Centares (Centares)	1	Square meters
Centigrams	0·01	Grams
Centimeters	0·3937	Inches
Centimeters	0·01	Meters..
Centimeters	10	Millimeters
Centimeters - mercury	0·0132	Atmospheres
Centimeters - mercury	0·4460	Feet - water (4° C)
Centimeters - mercury	136·0	Kilograms/m^2
Centimeters - mercury	27·85	Pounds/ft^2
Centimeters-mercury	0·1934	Pounds/in.^{2}
Centimeters/s	0·0328	Feet/s
Centimeters/s	0·036	Kilometers/h
Centimeters/s	0·6	Meters/m in
Centimeters/s	0·0224	Miles/h
Centimeters/s	0·0004	Miles/min
Cubic centimeters	3·531 x 10^5	Cubic feet
Cubic centimeters	, 0610	Cubic inches
Cubic centimeters	1 x 106	Cubic meters
Cubic centimeters	1·3079 x 10^{-6}	Cubic yards
Cubic centimeters	2·642 x 10^{-4}	Gallons
Cubic centimeters	0·0010	Liters.
Cubic centimeters	0·0021	Pints (liq.)
Cubic centimeters	0·0011	Quarts (liq.)
Cubic feet	1728	Cubic inches
Cubic feet	0·0283	Cubic meters
Cubic feet	7·4805	Gallons
Cubic feet	28·32	Liters
Cubic feet	59·84	Pints (liq.)
Cubic feet	29·92	Quarts (liq.)
Cubic feet/min	0·1247	Gallons/s
Cubic feet/min	0·4719	Liters/s
Cubic feet/s	448·831	Gallons/m in
Cubic inches	16·39	Cubic centimeters
Cubic inches	0·0005787	Cubic feet
Cubic inches	1·6387 x 10^{-5}	Cubic meters
Cubic inches	2·1433 x 10^{-5}	Cubic yards
Cubic inches	0·004329	Gallons
Cubic inches	0·0164	Liters
Cubic inches	0·0346	Pints (liq.)
Cubic inches	0·0173	Quarts (liq.)
Cubic meters	1 x 10^6	Cubic centimeters
Cubic meters	35·31	Cubic feet
Cubic meters	61023	Cubic inches
Cubic meters	1·308	Cubic yards
Cubic meters	264·2	Gallons

MULTIPLY	BY	TO OBTAIN
Cubic meters	1000	Liters
Cubic meters	2113	Pints (liq.)
Cubic meters	1057	Quarts (liq.)
Cubic yards	27	Cubic feet
Cubic yards	46·656	Cubic inches
Cubic yards	0·7645	Cubic meters
Cubic yards	202·0	Gallons
Cubic yards	764·5 -	Liters
Cubic yards	1616	Pints (liq.)
Cubic yards	807·9	Quarts (liq.)
Cubic yards/min	0·45 “	Cubic feet/s
Cubic yards/min	3·367	Gallons/s
Cubic yards/min	12·74	Liter/s
Degrees (angle)	60	minutes
Degrees (angle)	0·0174	Radians
Degrees (angle)	3600	Seconds
Degree/s	0·1667	Revolutions/min
Degree/s	0·0028	Revolutions/s
Drams	27·34	Grains
Drams	0·0625	Ounces
Drams	1·7718	GRAMS
Fathoms	6	Feet
Feet	30·48	Centimeters
Feet	12	Inches
Feet	0·3048	meters
Feet	0·3333	Yards
Feet- water (4°C)	0·8826	Inches - mercury
Feet-water	62·43	Pounds/ft^2
Feet/min	0·5080	Centimeters/s
Feet/min	0, 0183	Kilometers/h
Feet/min	0·3048	Meters/min
Feet/min	0·0114	Miles/h
Feet/s	30·48	Centimeters/s
Feet/s	1·097	Kilometers/h
Feet/s	18·29	Meters/min
Fee/s	0·6818	Miles/h
Feet/s	0·0114	Miles/min
Foot-pounds	0·0013	British Thermal units
Footpounds	5·0505 x l0^{-7}	Horsepower-hours
Footpounds	3·766 x l0^{-7}	Kilowatt-hours
Footpounds/m in	0·0167	Footpounds/s
Footpounds/m in	3·030xl0^{-5}	Horsepower
Footpounds/min	2·2597x10^{-5}	Kilowatts
Gallons	3785	Cubic centimeters
Gallons	0·1337	Cubic feet
Gallons	231	Cubic inches
Gallons	0·0038	Cubic meters
Gallons	3·785	Liters
Gallons	8	Pints (liq.)
Gallons	4	Quarts (liq.)
Gallons Imperial	1·2009	u. s. gallons
Gallons U. S.	0·8327	Imperial gallons
Gallons-water	8·34	Pounds-water
Grains	980·7	Dynes
Grams	15·43	Grains
Grams	0·0353	Ounces
Grams	0·0322	Ounces (troy)
Grams	0·0022	Pounds
Grams/cm3	0·0361	Pounds, in^3
Hectares	2·471	Acres
Horsepower	42·44	Btu/min
Horsepower	33000	Foot-pounds/min
Horsepower	550	Foot-pounds/s
Horsepower	1·014	Horsepower (metric)
Horsepower	0·7457	Kilowatts

MULTIPLY	BY	TO OBTAIN
Horsepower -hours	0·7457	Kilowatt-hours
Inches	2·540	Centimeters
Inches - mercury	0, 033	Atmospheres
Inches - mercury	3·45. 3	Kilograms/m^2
Inches - mercury	70·73	Pounds/ft^2
Inches - water	0·0735	Inches - mercury
Kilograms	980665	Dynes
Kilograms	2·205	Pounds
Kilometers	3281	Feet
Kilometers	1000	Meters
Kilometers	0·6214	Miles
Miles/h	1·609	Kilometers/h
Miles/h	0·8690	Knots
Miles/h	6·82	Meters/min
Miles/min	2682	Centimeters/s
Miles/min	88	Feet/s
Meters/min	1·609	Kilometers/min
Meters/min	60	Miles/h
Milligrams	0·001	Grains
Milliliters	0·001	Liters
Millimeters	0·1	Centimeters
Millimeters	0·0394	Inches
Ounces	16	Drams
Ounces	437·5	Grains
Ounces	0·0625	Pounds
Ounces	0·9115	Ounces (troy)
Ounces	2·8349 x 10^{-5}	Tons (metric)
Ounces (troy)	1·0971	Ounces (avoir.)
Ounces (fluid)	1·805	Cubic inches
Ounces (fluid)	0·0296	Liters
Pounds	16	Ounces
Pounds	256	Drams
Pounds	7000	Grains
Pounds	0·0005	Tons (short)
Pounds	1·2153	Pounds (troy)
Pounds/in3	1728	Pounds/ft.^{3}
Pounds/ft.	1488	Kilograms/m
Pounds/in.	178·6	Grams/cm
Pounds/ft. 2	4·882	Kilograms/m^2
Pounds/in. 2	0·0680	Atmospheres
Pounds/in. 2	2·036	Inches-mercury
Quadrants (angle)	1·571	Radians
Quarts (liq.)	57·75	Cubic inches
Quintal, metric	220·46	Pounds
Radians	57·30	Degrees
Radians	3438	Minutes
Radians	0·637	Quadrants
Radians/s	9·549	Revolutions/min
Revolutions/s	360	Degrees/s
Revolutions/s	6·283	Radians/s
Revolutions/s	60	Revolutions/min
Seconds (angle)	4·8481 x 10^{-6}	Radians
Square centimeters	0·0011	Square feet
Square centimeters	0·1550	Square inches
Square centimeters	0·0001	Square meters
Square centimeters	100	Square millimeters
Square feet	2·2957xl0^5	Acres
Square feet	929·0	Square centimeters
Square feet	144	Square inches
Square feet	0·0929	Square meters
Square feet	3·5870x10^8	Square miles
Square feet	0·1111	Square yards
Square inches	6·452	Square centimeters
Square inches	0·0069	Square feet
Square Kilometers	247·1	Acres

MULTIPLY	BY	TO OBTAIN
Square kilometers	1·0764xl0^{7}	Square feet
Square kilometers	lxlO6	Square meters
Square Kilometers	0·3861	Square miles
Square kilometers	1·1960xl0^{6}	Square yards
Square meters	10·76	Square feet
Square meters	1·1960	Square yards
Square miles	640	Acres
Square miles	2·590	Square kilometers
Square miles	3·0976xl0^{6}	Square yards
Square millimeters	0·01	Square centimeters
Square millimeters	0·0016	Square inches
Square yards	9	Square feet
Square yards	0·8361	Square meters
Square yards	3·2283xl0^{7}	Square miles
Tons (metric)	1000	Kilograms
Tons (metric)	2205	Pounds
Tons (short)	2000	Pounds
Tons (short)	0·89286	Ton (long)
Tons (short)	0·9072	Tons (metric)
Watts	0·0586	Btu (mean)/min
Watts	0·7377	Foot-pounds/s
Watts	0·0013	Horsepower
Watts	0·001	Kilowatts
Watt-hours	3·4144	British Thermal units
Watt-hours	2655	Foot-pounds
Watt-hours	0·00134	Horsepower-hours
Watt-hours	0·001	Kilowatt-hours
Yards	91·44	Centimeters
Yards	3	Feet
Yards -	36	Inches
Yards	0·9144	Meters

Cutting Speeds for High-Speed Steel Drills				
Drill size (inches)	Brass	Cast iron	Mild steel	Stainless steel
1/16	12,000	6,000	4,800	3,000
1/8	6,000	3,000	2,400	1,500
¼	3,000	1,530	1,200	1,000
3/8	2,000	1,000	815	500
½	1,530	760	610	380
5/8	1,220	610	490	300
¾	1,000	500	400	250
7/8	875	440	350	220
1	760	380	300	190

Diameters with Relative Circumferences			
Diameter (inches	Circumference (feet)	Diameter (inches)	Circumference (feet)
1/2	·131	3½	·92
3/4	·196	4	1·05
1	·26	4½	1·18
1¼	·33	5	1·31
1½	·40	5½	1·45
1¾	·45	6	1·56
2	·53	6½	1·83
2½	·65	7	1·83
3	·79	7½	1·95

R. P. M. Speeds for Fraction Size Drills[1]												
Feet per Min.	30	40	50	60	70	80	90	100	110	120	130	140
Diam. (Inches)	Revolutions per Minute											
1/16	1833	2445	3056	3667	4278	4889	5500	6111	6722	7334	7945	8556
1/8	917	1222	1528	1833	2139	2445	2750	3056	3361	3667	3973	4278
3/16	611	815	1019	1222	1426	1630	1833	2037	2241	2445	2648	2852
¼	458	611	764	917	1070	1222	1375	1528	1681	1833	1986	2139
5/16	367	489	611	733	856	978	1100	1222	1345	1467	1589	1711
3/8	306	407	509	611	713	815	917	1019	1120	1222	1324	1426:
7/16	262	349	437	524	611	698	786	873	960	1048	1135	1222
½	229	306	382	458	535	611	688	764	840	917	993	1070:
5/8	183	244	306	367	428	489	550	611	672	733	794	856
¾	153	203	255	306	357	407	458	509	560	611	662	713
7/8	131	175	218	262	306	349	393	436	480	524	568	611
1	115	153	191	229	267	306	344	382	420	458	497	535
1⅛	102	136	170	204	238	272	306	340	373	407	441	475
1¼	92	122	153	183	214	244	275	306	336	367	397	428
1⅜	83	111	139	167	194	222	250	278	306	333	361	389
1½	76	102	127	153	178	204	229	255	280	306	331	357
1⅝	70	94	117	141	165	188	212	235	259	282	306	329
1¾	65	87	109	131	153	175	196	218	240	262	284	306
1⅞	61	81	102	122	143	163	183	204	224	244	265	285
2	57	76	95	115	134	153	172	191	210	229	248	267
2¼	51	68	85	102	119	136	153	170	187	204	221	238
2½	46	61	76	92	107	122	137	153	168	183	199	214
2¾	42	56	69	83	97	111	125	139	153	167	181	194
3	38	51	64	76	89	102	Ho	127	140	153	166	178

[1] Courtesy of Cleveland Twist. Drill Co,

Fractional Thread Sizes						
Screw size (inches)	Thread series	Tap drill	"V" DD	60°	USS: DD	60°
¼-20	NC	7	·087	·050	·065	·038
¼-28	NF	3	·062	·036	·046	·027
5/16-18	NC	F(1/4)	-096	·056	·072	·042
5/16-24	NF	I(9/32)	·072	·042	·054	·031
3/8-16	NC	5/16	·108	·063	·081	·047
3/8-24	NF	Q(11/32)	·072	·042	·054	·031
7/16-14	NC	U(3/8)	·124	·071	·093	·054
7/16-20	NF	25/64	·087	·050	·065	·038
½-13	NC	27/64	·133	·077	·100	·058
½-20	NF	29/64	·087	050	·065	·038
9/16-12	NC	31/64	·144	·083	·108	·062
9/16-18	NF	33/64	·096	·056	·072	·042
5/8-11	NC	17/32	·157	·091	·118	·068
5/8-18	NF	37/64	·096	·056	·072	·042
¾-10	NC	21/32	·173	·100	·130	·075
¾-16	NF	11/16	·108	·063	·081	·047
7/8-9	NC	49/64	·192	·111	·144	·083
7/8-14	NF	13/16	·124	·071	·093	·054
1-8	NC	7/8	·217	·125	·162	·094
1-14	NF	15/16	·124	·071	·093	·054
1⅛-7	NC	63/64	·247	·143	·186	·107
1⅛-12	NF	1 3/64	·144	·083	·108	·062
1¼-7	NC	1 7/64	·247	·143	·186	·107
1¼-12	NF	1 11/64	·144	·083	·108	·062
1⅜-6	NC	1 7/32	·289	·167	217	·125
1⅜-12	NF	1 19/64	·144	·083	·108	·062
1½-6	NC	1 11/32	·289	·167	·217	·125
1½-12	NF	1 27/64	·144	·083	·108	·062
1¾-5	NC	1 9/16	·346	·200	·260	·150
2-4½	NC	1 25/32	·385	·222	·289	·167

Lathe Cutting Speeds[1]										
Feet per Min.	40	50	60	70	80	90	100	120	140	160
Diam. Inches	REVOLUTIONS PER MINUTE									
¼	611	764	917	1070	1222	L375	1528	1833	2140	2444
⅜	408	509	611	713	815	916	1018	1222	1426	1630
½	306	382	458	535	611	088	764	916	1070	1222
⅝	244	306	367	428	489	550	611:	733	856	978
¾	204	254	306	357	407	458	509	611	714	814
⅞	175	218	262	306	349	393	436	523	612	698
1	153	191	229	267	306	344	382	458	534	612
1⅛	136	170	204	238	272	305	339	407	476	544
1¼	122	153	183	214	244	275	305	366	428	488
1⅜	111	139	166	194	222	249	277	332	388	444
1½	202	127	153	178	204	229	254	305	356	408
1¾	87	109	131	153	175	196	218	262	306	350
2	76	95	114	133	153	172	191	229	266	306
2¼	68	85	102	119	136	153	170	204	238	272
2½	61	76	92	107	122	137	153	183	214	244
2¾	55	69	83	97	111	125	139	166	194	222
3	51	64	76	89	102	115	127	153	178	204
3¼	47	59	70	82	94	106	117	141	164	188
3½	44	54	65	76	87	98	109	131	152	174
3¾	41	51	61	71	81	92	102	122	142	162
4	38	48	57	67	76	86	95	114	134	152
4½	34	42	51	59	68	76	85	102	118	136
5	30	38	46	53	61	69	76	92	106	122
5½	28	35	42	49	55	62	69	83	98	110
6	25	32	38	44	51	57	64	76	88	102
6½	23	29	35	41	47	53	69	70	82	94
7	22	27	33	38	44	49	54	65	76	88
7½	20·4	25	31	36	41	46	51	61	72	82
8	19·1	24	29	33	38	43	48	57	66	76
8½	18	22	27	31	36	40	45	54	62	72
9	17·0	21·2	25	30	34	38	42	51	60	68
9½	16·1	20·1	24	28	32	36	40	48	56	64
10	15·3	19·1	23	27	31	34	38	46	54	62
11	13·9	17·4	20·8	24	28	31	35	41	48	56
12	12·7	15·9	19·1	22	25	29	32	38	44	50
13	11·8	14·7	17·6	20·6	23	26	29	35	41	46
14	10·9	13·6	16·4	19·1	22	24	27	33	38	44
15	10·2	12·7	15·3	17·8	20·4	23	25	30	35	41
16	9·5	11·9	14·3	16·7	19·1	21·4	24	29	33	38
17	9·0	11·2	13·5	15·7	18·0	20·2	22	27	31	36
18	8 5	10·6	12·7	14·8	17·0	19·1	21	25	30	34

[1] Source: Turret Lathe Operator's Manual, The Warner & Swaney Co.

Machine-Screw Threads			
Screw size (inches)	Thread series	Tap drill	Clearance drill
2/64	NC	50	42
2/56	NF	50	42
4/40	NC	43	31
4/48	NF	42	31
6/32	NC	36	26
6/40	NF	38	26
8/32	NC	29	17
8/36	NF	29	17
10/24	NC	25	8
10/32	NF	21	8
12/24	NC	16	1
12/28	NF	14	1

R.P.M. Speeds for Number Size Drills												
Feet per Min	30	40	50	60	70	80	90	100	110	120	130	Decimal Equivalents
No Size	Revolutions per Minute											
1	503	670	838	1005	1173	1340	1508	1675	1843	2010	2179	·2290
2	518	691	864	1037	1210	1382	1555	1728	1901	2074	2247	·2210
3	538	717	897	1076	1255	1434	1614	1793	1974	2152	2331	·2130
4	548	731	914	1097	1280	1462	1645	1828	2010	2193	2376	·2090
5	558	744	930	1115	1301	1487	1673	1859	2045	2230	2416	·2055
e	562	749	936	1123	1310	1498	1685	1872	2060	2247	2434	·2040
7	570	760	950	1140	1330	1520	1710	1900	2090	2281	2470	·5010
8	576	768	960	1151	1343	1535	1727	1919	2111	2303	2495	·1990
9	585	780	975	1169	1364	1559	1764	1949	2144	2339	2534	·1960
10	592	790	987	1184	1382	1579	1777	1974	2171	2369	2566	·1935
11	600	800	1000	1200	1400	1600	1800	2000	2200	2400	2600	·1910
12	606	80S	1010	1213	1415	1617	1819	2021	2223	2425	2627	·1890
13	620	826	1032	1239	1450	1652	1859	2065	2271	2479	2684	·1850
14	630	840	1050	1259	1469	1679	1889	2099	2309	2518	2728	·1820
15	638	851	1064	1276	1489	1702	1914	2127	2334	2546	2759	·1800
16	647	863	1079	1295	1511	1726	1942	2158	2374	2590	2806	·1770
17	662	883	1104	1325	1546	1766	1987	2208	2429	2650	2870	·1730
18	678	904	1130	1356	1582	1808	2034	2260	2479	2704	2930	·1695
19	690	920	1151	1381	1611	1841	2071	2301	2531	2761	2991	·1660
20	712	949	1186	1423	1660	1898	2135	2372	2610	2847	3084	·1610
21	721	961	1201	1441	1681	1922	2162	2402	2644	2883	3123	·1590
22	730	973	1217	1460	1703	1946	2190	2433	2676	2920	3164	·1570
23	744	992	1240	1488	1736	1984	2232	2430	2728	2976	3224	·1540
24	754	1005	1257	1508	1759	2010	2262	2513	2764	3016	3267	·1520
25	767	1022	1276	1533	1789	2044	2300	2555	2810	3066	3322	·1495
26	779	1039	1299	1559	1819	2078	233£	2598	2858	3118	3378	·1470
27	796	1061	1327	1592	1857	2122	238$	2653	2919	3183	3448	·1440
28	816	1088	1360	1631	1903	2175	2447	2719	2990	3202	3534	·1405
29	843	1124	1405	1685	1966	2247	252$	2809	3090	3370	3651	·1360
30	892	1189	1487	1784	2081	2378	2676	2973	3270	3567	3864	·1285
31	955	1273	1592	1910	2228	2546	2865	3183	3501	3821	4138	·1200
32	988	1317	1647	1976	2305	2634	2964	3293	3622	3951	4281	, 1160
33	1014	1352	1690	2028	2366	2704	3043	3380	3718	4056	4394	·1130
34	1032	1376	1721	2065	2409	2753	3093	3442	3785	4129	4474	·1110
35	1042	1389	1736	2083	2430	2778	3125	3472	3821	4167	4514	·1100
36	1076	1435	1794	2152	2511	2870	3228	3587	3945	4304	4663	·1065
37	1102	1469	1837	2204	2571	2938	3306	3673	4040	4407	4775	·1040
38	1129	1505	1882	2258	2634	3010	3587	3763	4140	4516	4892	·1015
39	1152	1530	1920	2303	2687	3071	3455	3839	4222	4607	4991	·0995
40	1169	1559	1949	2339	2729	3118	3508	3898	4287	4677	5067	·0980
41	1194	1592	1990	2387	2785	3183	3581	3979	4377	4775	5172	·0960
42	1226	1634	2043	2451	2860	3268	3677	4085	4494	4902	5311	·0935
43	1288	1717	2146	2575	3004	3434	3863	4292	4721	5150	5579	·0890
44	1333	1777	2221	2665	3109	3554	3999	4442	4886	5330	3774	·0660
45	1397	1863	2329	2795	3261	3726	4192	4658	5124	5590	3056	·0820

[1] Courteny of Cleveland Twist Drill Co,

R.P.M. Speeds for Number Size Drills (Continued)												
Feet per Min	30	40	50	60	70	80	90	100	110	120	130	Decimal Equivalents
No. Size	Revolutions per Minute											
46	1415	1886	2358	2830	3301	3773	4244	4716	5187	5659	6130	·0810
47	1460	1946	2433	2920	3406	3893	4379	4866	5352	5839	6326	·07S5
48	1508	2010	2513	3016	3518	4021	4523	5026	5528	6031	6534	·0760
49	1570	2093	2617	3140	3663	4186	4710	5233	5756	6279	6808	·0730
50	1637	2183	2729	3274	3820	4366	4911	5457	6002	6548	7094	·0700
51	1710	2280	2851	3421	3991	4561	5131	5701	6271	6841	7413	·0670
52	1805	2406	3008	3609	4211	4812	5414	6015	6619	7218	7820	·0635
53	1924	2566	3207	3848	4490	5131	5773	6414	7062	7704	8346	·0595
54	2084	2778	3473	4167	4862	5556	6251	6945	7639	8334	9028	·0550
55	2204	2938	3673	4408	5142	5877	6611	7346	8080	8815	9549	·0520
56	2465	3286	4108	4929	5751	6572	7394	8215	9036	9857	10678	·0465
57	2671	3561	4452	5342	6232	7122	8013	8903	9771	10660	11548	·0430
58	2729	3637	4547	5456	6367	7275	8186	9095	10004	10913	11823	·0420
59	2795	3726	4658	5590	6521	7453	8388	9316	10248	11180	12111	·0410
60	2865	3820	4775	5729	6684	7639	8594	9549	10504	11459	12414	·0400
61	2938	3918	4897	5876	6856	7835	8810	9794	10774	11753	12732	·0390
62	3015	4020	5025	6030	7035	8040	9045	10050	11057	12060	13068	·0380
63	3096	4128	5160	6192	7224	8256	9288	10320	11366	12398	13421	·0370
64	3183	4244	5305	6366	7427	8488	0549	10610	11671	12732	13793	·0360
65	3273	4364	5455	6546	7637	8728	9819	10910	12005	13096	14187	·0350
66	3474	4632	5790	6948	8106	9264	10422	11580	12732	13890	15047	·0330
67	3582	4776	5970	7164	8358	0552	10746	11940	13130	14324	15517	·0320
68	3696	4928	6160	7392	8624	9856	11088	12320	13554	14786	16018	·0310
69	3918	5224	6530	7836	9142	10488	11754	13060	14389	15697	17006	·0292
70	4091	5456	6820	8184	9548	10912	12276	13640	15006	16370	17734	·0280
71	4419	5892	7365	8838	10311	11784	13257	14730	16160	17629	19099	·0260
72	4584	6112	7640	9168	10696	12224	13752	15280	16807	15335	19863	·0250
73	4776	6368	7960	9552	11144	12736	14328	15920	17507	19099	20690	·0240
74	5106	6808	8510	10212	11914	13616	15318	17020	18674	20372	22069	·0225
75	5457	7276	9095	10914	12733	14552	16371	18190	20008	21827	23646	·0210
76	5730	7640	9550	11460	13370	15280	17100	19100	21008	22918	24828	·0200
77	6366	8488	10610	12732	14854	16976	19098	21220	23343	25465	27587	·0180
78	7161	9548	11935	14322	16709	19096	21483	23870	26260	28648	31035	·0160
79	7902	10536	13170	15804	18438	21072	23706	26340	28988	31611	34246	·0145
80	8490	11320	14150	16980	19810	22640	25470	28300	21123	33953	36782	·0135
Courtesy of Cleveland Twist Drill Co.												

Milling Machine R.P.M Necessary to Give a Desired Cutting Speed[1]						
	Cutting Speeds is Feet Per Minute					
Diameter (Inches)	40	50	60	70	80	90
	Revolutions per Minute					
¼	611	764	917	1,070	1,222	1,375
5/16	489	611	733	856	978	1,100
3/8	407	509	611	713	815	917
7/16	349	437	524	611	698	786
½	306	382	458	535	611	688
5/8	244	306	367	428	489	550
¾	204	255	306	357	407	458
7/8	175	218	262	306	349	393
1	153	191	229	267	306	344
1⅛	136	170	204	238	272	306
1¼	122	153	183	214	244	275
1⅜	111	139	167	194	222	250
1½	102	127	153	178	204	229
1⅝	94	117	141	165	188	212
1¾	87	109	131	153	175	196
1⅞	81	102	122	143	163	183
2	76	95	115	134	153	172
2¼	68	85	102	119	136	153
2½	61	76	92	107	122	137
2¾	56	69	83	97	111	125
3	51	64	76	89	102	115
3½	44	55	65	76	87	98
4	38	48	57	67	76	86
4½	84	42	51	59	68	77
5	31	38	46	54	61	69
5½	28	35	42	49	56	63
6	25	32	28	45	51	57
7	22	27	33	38	44	49
8	19	24	29	33	38	43
9	17	21	25	30	34	38
10	15	19	23	27	31	34
11	14	17	21	24	28	31
12	13	16	19	22	25	29
13	12	15	18	21	24	27
16	10	12	14	17	19	22
18	8	11	13	15	17	19
[1] Source: American Machinist.						

Suggested Drill Speeds for Various Materials[1]	
Material to be Drilled	Cutting Speed (Surface Feet per Minute)
Aluminum and its alloys	200-300
Bakalite	100-150
Brass and bronze, soft	200-300
Bronze, high tensile	70-150
Carbon, pure (carbide drills)	100
Cast iron, soft	100-150
Cast iron, hard	70-100
Cast iron, chilled	30-40
Copper graphite alloy (carbide drills)	60-70
Glass (carbide drills)	20-30
Magnesium and its alloys	250-400
Malleable iron	80-90
Marble	15-25
Marble (carbide drills)	60-80
Nickel and monel	40-60
Slate	15-25
Slate (carbide drills)	40
Steel, machinery (0·2-0·3 c)	80-110
Steel, annealed (0·4-0·5 c)	70-80
Steel, tool (1·2 c)	50-60
Steel, forged	50-60
Steel, alloy (300 to 400 Brinnel)	20-30
Steel, stainless, free machining	30-40
Steel, stainless, hard	30-40
Steel, manganese	15
Stone	15-25
Stone (carbide drills)	30
Wood	300-400
[1] Source: American Machinist.	

Tap Drill Sizes									
American National Coarse Standard Thread (N.C,) Formerly U.S. Standard					American National Fine Standard Thread (N.F.) Formerly S.A.E. Thread				
Sizes	Threads per Inch	Outside Diameter of Screw	Tap Drill Sixes	Decimal Equivalent of Drill	Sizes	Threads per Inch	Outside Diameter of Screw	Tap Drill Sizes	Decimal equivalent of Drill
1	64	·073	53	·0595	0	80	·060	3/64	·0469
2	66	·086	50	·0700	1	72	·073	53	·0595
3	48	·099	47	·0785	2	64	080	50	·0700
4	40	·112	43	·0890	3	58	·099	45	·0820
5	40	·125	38	·1035	4	48	·112	42	·0935
6	32	·138	36	·1065	5	44	·125	37	·1040
8	32	·164	29	·1360	6	40	·138	33	·1130
10	24	·190	25	·1495	8	36	·164	29	·1360
12	24	·216	16	·1770	10	32	·190	21	·1590
¼	20	·250	7	·2010	12	28	·216	14	·1820
5/16	18	·3125	F	·2570	¼	28	·250	3	·2130
⅜	16	·375	5/16	·3125	5/16	24	·3125	I	·2720
7/16	14	·4375	U	·3080	3/8	24	, 375	Q	·3320
½	13	·500	27/64	·4219	7/16	20	·4375	25/64	·3006
9/16	12	·5625	51/64	·4843	½	20	·500	29/64	·4531
⅝	11	·625	17/32	·5312	9/16	18	·5625	·5062	·5062
¾	10	·750	21/32	·6562	5/8	18	·625	·5687	·5687
⅞	9	·875	49/64	·7636	¾	16	·750	11/16	·6575
1	8	1·000	7/8	·875	7/8	14	·875	·8020	·8020
1⅛	7	1·125	63/64	·9843	1	14	1·000	·9274	·9274
1¼	7	1·250	1 7/64	1·1093	1⅛	12	1·123	1 3/64	1·0468
					1¼	12	1·250	1 11/64	1·1718
Courtesy of South Bend Lathe Works									

Unified and American Standard Coarse Thread Series-Basic Dimensions								
Size	Basic Major Diam	Thds. per Inch	Basic Pitch Diam.	Minor Diam. Ext. Thds	Minor Diam. Int. Thds.	Lead Angle, Basic Pitch Diam.		Area Minor Diam.
	Inches		Inches	Inches	Inches	Deg.	Min.	Sq. In.
1 (073)	·0730	64	·0629	·0538	·0561	4	31	·0022
2 (086)	·0860	56	·0744	·0641	·0667	4	22	·0031
3 (099)	·0990	48	·0855	·0734	· 0764	4	26	·0041
4 (112)	·1120	40	·0958	·0813	·0849	4	45	·0050
5 (125)	·1250	40	·1088	·0943	·0979	4	11	·0067
6 (138)	·1380	32	·1177	·0997	·1042	4	50	·0075
8 (164)	·1640	32	·1437	·1257	·1302	3	58	·0120
10 (190)	·1900	24	·1629	·1389	·1449	4	39	·0145
12 (216)	·2160	24	·1889	·1649	·1709	4	1	·0206
¼	·2500	20	·2175	·1887	·1959	4	11	·0269
5/16	·3125	18	·2764	·2443	·2524	3	40	·0454
3/8	·3750	16	·3344	·2983	·3073	3	24	·0678
7/16	·4375	14	·3911	·3499	·3602	3	20	·0933
½	·5000	13	·4500	·4056	·4167	3	7	·1257
9/16	·5625	12	·5084	·4603	·4723	2	59	·1620
5/8	·6250	11	·5660	·5135	·5266	2	56	·2018
¾	·7500	10	·6850	·6273	·6417	2	40	·3020
7/8	·8750	9	·8023	·7387	·7547	2	31	·4193
1	1·0000	8	·9188	·8466	·8647	2	29	·5510
1⅛	1·1250	7	1·0322	·9497	·9704	2	31	·6931
1¼	1·2500	7	1·1572	1·0747	1·0954	2	15	·8898
1⅜	1·3750	6	1·2667	1·1705	1·1946	2	24	1·0541
1½	1·5000	6	1·3917	1·2955	1·3196	2	11	1·2938
1¾	1·7500	5	1·6201	1·5046	1·5335	2	15	1·7441
2	2·0000	4½	1·8557	1·7274	1·7594	2	11	2·3001
2¼	2·2500	4½	2·1057	1·9774	2·0094	1	55	3·0212
2½	2·5000	4	2·3376	2·1933	2·2294	1	57	3·7161
2¾	2·7500	4	2·5876	2·4433	2·4794	1	46	4·6194
3	3·0000	4	2·8376	2 6933	2·7294	1	36	5·6209
3¼	3·2500	4	3·0876	2·9433	2·9794	1	29	6·7205
3½	3·5000	4	3·3376	3·1933	3·2294	1	22	7·9183
3¾	3·7500	4	3·5876	3·4433	3·4794	1	16	9·2143
4	4·0000	4	3·8376	3·6933	3·7294	1	11	0·6084
Source: Machinery's Handbook.								

Unified and AMERICAN Standard Fine and Extra-Fine Thread Series

Size	Basic Major Diam. Inches	Thds. per Inch	Basic Pitch Diam. Inches	Mirror Diam. Ext. Thds. Inches	Minor Diam. Int. Thds. Inches	Lead Angle. Pitch Diam. Deg. Min.		Area, Minor Diam. Sq. In.
	FINE THREAD SERIES							
0 (·060)	·0600	80	·0519	·0447	·0465	4	23	·0015
1 (·073)	·0730	72	·0640	·0560	·0580	3	57	·0024
2 (·086)	·0860	64	·0759	·0658	·0691	3	45	·0034
3 (·099)	·0990	56	·0874	·0771	·0797	3	43	·0045
4 (·112)	·1120	48	·0985	·0864	·0894	3	51	·0067
5 (·125)	·1250	44	1102	·0971	·1004	3	45	·0072
6 (·138)	·1380	40	·1218	·1073	·1109	3	44	·0087
8 (·164)	·1640	36	·1460	·1299	·1339	3	28	·0128
10 (·190)	·1900	32	·1697	·1517	·1562	3	21	·0175
12 (·216)	·2160	28	·1928	·1722	·1773	3	22	·0226
¼	·2500	28	·2268	·2062	·2113	2	52	·0326
5/16	3125	24	·2854	·2614	·2674	2	40	·0524
3/8	·4750	24	·3479	·3239	·3299	2	11	·0809
7/16	·4375	20	·4050	·3762	·4834	2	15	·1090
½	·5000	20	·4675	·4387	·4459	1	57	·1486
9/16	·5625	18	·5264	·4943	·5024	1	55	·1888
5/8	·6250	18	·5889	·5568	·5649	1	43	·2400
¾	·7500	16	·7094	·6733	·6823	1	36	·3513
7/8	·8750	14	·8286	·7874	·7977	1	34	·4805
1	1·0000	14	·9536	·9124	·9227	1	22	·6464
1	1·0000	12	·9459	·8978	·9098	1	36	·6245
1⅛	1·1250	12	1·0709	1·0228	1·0348	1	25	·8118
1¼	1·2500	12	1·1959	1·1478	1·1598	1	16	1·0237
1⅜	1-3750	12	1·3209	1·2728	1·2848	1	9	1·2602
1½	1·5000	12	1·4459	1·3978	1·4096	1	3	1·5212

EXTRA-FINE THREAD SERIES								
12 (·216)	·2160	32	·1957	·1777	·1822	2	55	·0242
¼	·2500	32	·2297	·2117	·2162	2	29	·0344
5/16	·3125	32	·2922	·2742	·2787	1	57	·0581
3/8	·3750	32	·3547	·3367	·3412	1	36	·0678
7/16	·4375	28	·4143	·3937	·3988	1	34	·1201
½	·5000	28	·4768	·4562	·4613	1	22	·1616
9/16	·5625	24	·5354	·5114	·5174	1	25	·2030
5/8	·6330	24	·5079	·5739	·5799	1	16	·2560
11/16	·6875	24	·6604	·6364	·6424	1	9	·3151
¾	·7500	20	·7175	·6887	·6959	1	16	·3585
13/16	·8125	20	·7800	·7512	·7584	1	10	·4388
7/8	·8750	20	·8425	·8137	·8209	1	5	·5153
15/16	·9375	20	·9050	·8762	·8834	1	0	·5979
1	1·0000	20	·9675	·9387	·9459	0	57	·6866
1 1/16	1·0625	18	1·0264	·9943	1·0024	0	59	·7702
1 1/8	1·1250	18	1·0889	1·0568	1·0649	0	56	·8705
1 3/16	1·1875	18	1·1514	1·1193	1·1274	0	53	·9770
1 ¼	1·2500	18	1·2139	1·1818	1·1899	0	50	1·0895
1 5/16	1·3125	18	1·2764	1·2443	1·2524	0	48	1·2082
1 3/8	1·3750	18	1·3389	1·3068	1·3149	0	45	1·3330
1 7/16	1·4375	18	1·4014	1·3693	1·3774	0	43	1·4640
1 ½	1·5000	18	1·4639	1·4318	1·4399	0	42	1·6011
1 9/16	1·6625	18	1·5264	1·4943	1·5024	0	40	1·7444
1 3/8	1·6250	18	1·5889	1·5568	1-5649	0	38	1·8937
1 11/16	1·6875	18	1·6514	1·6193	1·6274	0	37	2·0493
1 3/4	1·7500	16	1·7094	1·6733	1·6823	0	40	2·1873
2	2·0000	16	1·9594	1·9233	1·9323	0	35	2·8917

Source: Machinery's Handbooks

Hardness Conversion Relations for Tool Steels						
Rockwell C Scale, diamond	Rockwell B Scale, 1/16-in. ball	Brinell		Vickers diamond pyramid hardness	Scleroscope hardness number	Superficial Rockwell 30N scale
		Impression diam, mm	Hardness number			
68				940	97	84
67				900	95	84
66				865	92	83
65				332	91	82
64				800	88	81
63				772	67	60
62				746	85	79
61				720	83	78
60				697	81	78
59				674	90	77
58				653	78	76
57				633	76	75
56				613	75	74
55				595	74	73
54				577	72	72
53				560	71	71
52				544	69	70
51				528	68	69
50				513	67	69
49		2 85	461	438	66	68
48			451	464	64	67
47		2·90	444	471	63	66
46		2· 95	429	458	62	65
45			421	446	60	64
44		3·00	415	434	58	63
43		3·05	401	423	57	62
42		3·10	338	412	56	61
41			381	402	55	60
40		3·15	375	392	54	60
39		3·20	363	382	52	59
38		3·25	352	372	51	58
37		3·30	341	363	50	57
36	109		336	354	49	56
35	109	3·35	331	345	48	55
34	108	3·40	321	336	47	54
33	108	3·45	311	327	46	53
32	107	3·50	302	318	44	52
31	106	3·55	293	310	43	51
30	106	3·60	28$	302	42	50
29	105	3·65	277	294	41	50
28	104	3·70	269	286	41	49
27	103	3·75	262	279	40	46
26	103		258	272	39	47
25	102	3·80	255	266	38	46
24	101	3·85	248	260	37	45
23	100	3·90	241	254	36	44
22	99	3·95	235	248	35	43

ROCKWELL HARDNESS-CONTINUED

Rockwell C Scale, diamond	Rockwell B Scale, 1/16-in. ball	Brinell		Vickers diamond pyramid hardness	Scleroscope hardness number	Superficial Rockwell 30N scale
		Impression diam, mm	Hardness number			
21	98	4·00	228	243	35	42
20	97	4·05	223	238	34	42
	96	4·10	217	230	33	
	95	4·15	212	222	32	
	94	4·20	207	217	31	
	93	4·25	202	213	31	
	92	4·30	196	209	30	
	91	4·35	192	204	29	
	90	4·40	187	195	28	
	89	4·45	183	191	27	
	88	4·50	179	187	27	
	87	4·55	174	184	26	
	86	4·60	170	180	26	
	85	4·65	166	173	25	
	84	4·70	163	170	25	
	83	4·75	159	168	24	
	82	4·80	156	166	24	
	81	4·85	153	161	23	
	81	4·90	149	156	23	
	80	4·95	146	153	22	
	79	5·00	143	150	22	

SPEEDS				
Material	Brn. Range	H.S.S. Cutter S.F.M. Range	Cast Alloy Cutter S.F.M. Range	Carbide Cutter S.F.M. Range
Aluminum	100—150	1000—550	2000—1100	4000—2200
Brass	100—175	650—250	1300—500	2600—1000
Low-Carbon Steel	100—200	325—100	650—200	1300—400
Free-Cutting Steel	150—200	250—150	500—300	1000—600
Alloy Steel	150—250	175—70	350—140	700—280
Alloy Ste«l	250—350	70—40	140—80	280—160
Cast Iron	125—175	100—60	200—120	400—240
Cast Iron	175—200	60—45	120—90	240—180
Cast Iron	200—225	45—40	90—80	180—160
Cast Iron	225—250	40—35	80—70	160—140

Speeds For Machining

Milling Machine R.P.M- Necessary to Give a Desired Cutting Speed						
	Cutting Speeds in Feet per Minute					
Diameter (Inches)	40	50	60	70	80	90
	Revolutions per Minute					
1/4	611	764	917	1,070	1,222	1,375
5/16	489	611	733	856	978	1,100
3/8	407	509	611	713	815	917
7/16	349	437	524	611	698	786
½	306	382	458	535	611	688
5/8	244	306	367	428	489	550
¾	204	255	306	357	407	458
7/8	175	218	262	306	349	393
1	153	191	229	267	306	344
1 ⅛	136	170	204	238	272	306
1 ¼	122	153	183	214	244	275
1 ⅜	111	139	167	194	222	250
1 ½	102	127	153	178	204	229
1 ⅝	94	117	141	165	188	212
1 ¾	87	109	131	153	175	196
1 ⅞	81	102	122	143	163	183
2	76	95	115	134	153	172
2 ¼	68	85	102	119	136	153
2 ½	61	76	92	107	122	137
2 ¾	56	69	83	97	111	125
3	51	64	76	89	102	115
3 ½	44	55	65	76	87	98
4	38	48	57	67	76	86
4 ½	34	42	51	59	68	77
5	31	38	46	54	61	69
5 ½	28	35	42	49	56	63
6	25	32	28	45	51	57
7	22	27	33	38	44	49
8	19	24	29	33	38	43
9	17	21	25	30	34	38
10	15	19	23	27	31	34
11	14	17	21	24	28	31
12	13	10	19	22	25	29
13	12	15	18	21	24	27
16	10	12	14	17	19	22
18	8	11	13	15	17	19