

BARREL FITTING



Barrel Fitting

Copyright (c) 2000 Harold Hoffman

We have many good Mini and other books on Gunsmithing, Knife making, History, Out of date, and Crafts books. The purpose is to give you the basic information on subject that is covered here. I hope you enjoy and learn from these Ebooks. H. Hoffman

All rights reserved. No parts of this publication may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopy, recording, or any information storage and retrieval system, without the written consent of the publisher.

Action Books

7174 Hoffman Rd.

San Angelo, TX. 76905

Phone/Fax 915-655-5953

[Home Site](#)

[Email](#)

Zaibatsu Release

I am the alpha, the omega,
the beginning and the end

BARREL FITTING



ABOUT THE AUTHOR

Harold Hoffman has through his 30 plus years of experience as a Gunsmith, Toolmaker and Custom Knife maker has passed on to you through his books information that soon may be lost or forgotten. His books are not intended for the person wanting to make a complete firearm, but for learning basic shop tool making.

The information found within his books is for instructional purpose only. -- The titles DO NOT actual cover gun repair on firearms, but how to make needed parts for firearms which is about 40% of all gun repair. Without this information you will be severely limited in gun repair. He first started gun repair when he was 18 years old doing minor repair for the farmers and local hunters in the Bucklin, Kansas area. His main interest was how to make rifle barrels, as he was an avid hunter. Moving into a bigger shop he bought a lathe and proceeded to learn how to use it. He wanted to find out how to make rifling buttons to rifle barrels, tool making, and learn everything about making barrels.

Over the years he became an expert toolmaker and how to build most everything that was needed in the shop. The information found in his books will show you how to make most of the equipment and tools needed in most shops. After an eye accident he quit Gunsmithing and started writing books on everything that he knew. He had so much difficulty finding any information that he wanted all this information that he had learned in over 30 years to be available to everyone otherwise it would be lost. His books are now about the only books available on Gunsmithing/Tool making, as most publishers do not publish Gun or Gunsmithing books anymore.

BARREL FITTING

BARREL FITTING

One of the major operations on guns is barrel replacement. When the new barrel is a factory standard for the gun in the case this is usually a simple job on rifle or shotgun, as these factory barrels practically always screw right up to the proper place. All gunsmiths usually do some barrel fitting, to keep factory-made arms in use. The arms manufacturers do not sell barrels for high-powered rifles, but require that the actions be sent them so they can fit and head space the actions. Nearly all .22 rim-fire barrels and lever action rifle barrels are easily fitted, with very little machine work.

Removing the old barrel is not always so simple. If it is an octagon barrel, remove the tubular magazine. If the rifle is so equipped, and withdraw the breech bolt far enough so the extractor clears its slot in the end of the barrel by a safe margin. Next, place copper jaw-facings on the bench vise jaws and grip the barrel tightly in the vise, with the receiver close to the jaws. Pad the sides of the receiver with sheet lead or copper and place a heavy crescent wrench (at least a 24"), on the receiver, being careful to place it as close to the front edge of the receiver as possible. This is done to avoid crushing in the thin sides of the receiver back of the receiver ring:

The finished barrels as ordered fit the old receiver as they come. Often you may find an old receiver, which allows the new barrel to turn and go past the index mark so sights will not line up. If it turns in only little past, or just lining up without tension on the wrench, it is possible to set the barrel in tightly by removing it and carefully peening the shoulder of the new barrel. This swages the metal out so that it will offer greater bearing surface against the front of the receiver.

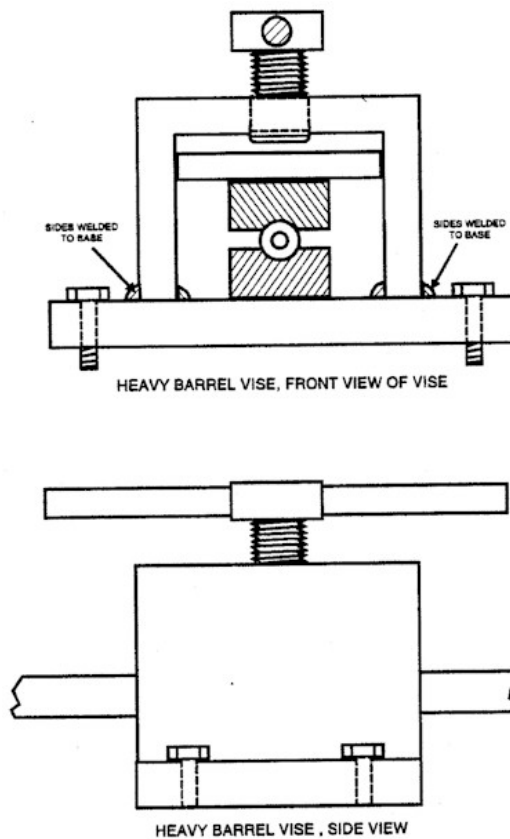
If the barrel goes too far past for lining up by peening, the best thing to do is to make a washer of thin shim stock to go around the barrel shank between the shoulder and the receiver. If the barrel will not screw in to the index, you can file or take off a little in the lathe, which will take an even amount off all the way around. If the work can be done on the lathe it is better, but be careful to only take only a very little off at a time. What ever you do, do not remove metal from the front of the receiver back always work on the barrel. If you have a pin-in unthreaded barrel that fits loosely, due to a worn pinhole. Set the barrel in tight against the receiver, clamp it there the best way you can and drill out the receiver and the barrel together for the next size larger pin.

BARREL FITTING

CHANGING BARRELS

To change a Springfield, Enfield or Mauser type barrels is usually a simple matter. To replace these barrels, you need a sturdy barrel vise, as well as an action wrench. Always clamp the barrel in the vise and turn the receiver. Two or three good commercial barrel vises are on the market, but I find it is easy and much cheaper to make your own, as shown in the drawing. You can use the regular style rigid base machinist's bench type vise for barrel work. A vise, of the size and strength necessary for barrel removal and replacement would cost at least twice as much as a commercial barrel vise. Do not try to remove Mauser or Enfield barrels in your regular four or five-inch bench vise, as they are not stout enough.

The barrel vise or clamp must be mounted very rigidly, and if you can, in cement. A very heavy, solid bench that is bolted to the floor will work also. The jaws for these vises may be of very good grooved hardwood that is covered with powdered rosin before fitting to the barrel.

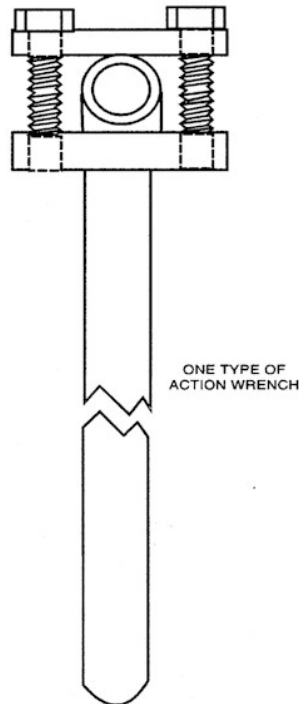


Better yet, of lead or Babbitt metal, cast to the contour of the barrel involved. You have to hold the barrel tight enough to prevent it turning when you bear down on the wrench. Jaws should be smooth for fitting finished barrels without marring them, but can be of drilled steel for removal of unusable military barrels, which are usually set in very tightly. A barrel should be clamped as close to the receiver as conveniently possible, to minimize twisting action.

When the barrel just will not unscrew, the welding torch can sometimes be applied, the receiver ring over the threads and given a fast heating to expand it slightly. Only a little heat is necessary-not enough to change the color of the metal, but it should be fast, so the receiver

BARREL FITTING

gets the benefit and not the barrel shank. A butane torch can be used, but it spreads a lot of heat around and does not concentrate it, as does the acetylene tip. Keep the receiver just hot enough so the color does not change, until the barrel is cool to the touch. Quickly place the wrench on the receiver and give it a sudden, heavy pull-which will usually break it loose.

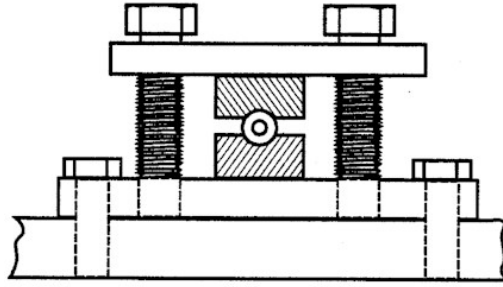


If this method fails to loosen a stubborn barrel, the only thing left to do is cut a groove at the front of the receiver. A cut off tool can be used to cut a recess in front of the receiver, this being done while the barrel is clamped in the late, and running at slow speed. Nearly all rifle barrels are fitted to the receiver with a right-hand thread; hut there is always the exception, as I once found out when removing the barrel from a 6.5 MM Norwegian Krag rifle. As it happened, I was able to move this barrel slightly in a left-hand direction and then it tightened up and would move no farther so, in spite of disbelief, I reversed operations and the barrel unscrewed to the right. The old, hammer model, Marlin shotgun is also a left hand thread.

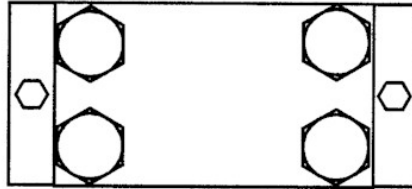
Setting new barrels in bolt-action receivers is simply the reverse of removing the old one, naturally, and as a rule is much easier, the principal problem is keeping the barrel clear of marks from the blocks. Hardwood blocks are almost the only type, which will not mar a finished barrel. The ideal fit is to have the barrel thread into the receiver so that it can barely be started by hand but turned easily by the receiver wrench until within 316 to 1/8" of lining up. If going by the index marks on the barrel and the receiver.

It should then tighten until the marks line up. A 30" length of pipe for a cheater handle can be used to bring the shoulder of the barrel against the receiver ring tightly. The shoulder should seat firmly against the receiver all the way around. This is important in .22 rim-fire match rifles, or any target arm. It can be spotted by painting the shoulder with layout fluid, or shoe polish and turning in until the shoulder bears against the receiver lightly. Then removed for inspection, the spots showing wear being reduced slightly. The barrel shoulders will be more accurately machined than the front of the receivers, on large caliber rifles, so clean the front of the action good before fitting the barrel.

BARREL FITTING



THIS IS A HEAVY DUTY CLAMP TYPE VISE, FRONT VIEW



TOP VIEW OF SAME VISE

BARREL FITTING

SHOTGUN BARRELS

Shotgun barrels usually unscrew easier than rifle barrels do and, as they are round, the same method is used as for round rifle barrels. A hardwood block is bored out to fit the shotgun barrel, and then split.

Before clamping a barrel in one of these clamps, coat the surface of the sleeve that is to be against the barrel with powdered rosin. This gives it a much better grip and protects the surface of the barrel, not even marring the bluing. This rosin may afterwards be cleaned from the surface of a barrel with high-test gasoline or turpentine.

Be careful not to set the clamp up too tightly on a shotgun barrel if it is a new barrel, or a barrel you are going to use again, as you may crush it. When setting it up on a high-power rifle barrel, oil the threads on the bolts holding the two parts of the clamp together, and pull the nuts up tightly with a long-handled socket wrench.

A shotgun barrel has a fairly straight taper from the breech, so the clamp can be fitted back close to the receiver.

After the old barrel has been removed, clean out the threads in the receiver with an old toothbrush, dipped in solvent. Also, clean the threads on the new barrel with brush and solvent, to remove the heavy oil and any dirt that may have collected upon them. Oil the threads of the new barrel with a light oil, and screw the barrel into the receiver as far as it will go by hand. Place the shotgun barrel in the barrel vise and screw the receiver up to place with the crescent wrench. If you cannot make the receiver screw quite up to the mark on the barrel, unscrew it from the barrel and, using a large, flat, dead smooth file or a large Carborundum stone, dress off the barrel, and, after cleaning it, screw it onto the barrel again.

Repeat this process until you can screw the receiver up to its proper position. If the receiver screws up too far on the new barrel, peen it slightly. The correct procedure is to place the barrel in the lathe, face-off the end of the barrel and the barrel shoulder far enough so that the barrel can make another complete turn. This is less the amount required to make the marks on receiver and barrel meet.

Knowing how many threads to the inch are on the barrel, it is easy to figure just how many thousandths of an inch the barrel will move in the receiver in making one complete revolution in the threads. After the barrel is chucked in the lathe, bring the facing tool lightly against the end of the barrel, or against the barrel shoulder, and set the micrometer carriage stop against the forward end of the lathe carriage.

Next, back off the micrometer stop the required number of thousandths of an inch for one revolution of the barrel, less the amount required to make the marks on receiver and barrel meet. This number of thousandths of an inch is then deducted from the number of thousandths of an inch the barrel moves in a full revolution. The answer is the amount the micrometer stop is to be backed away from the lathe carriage.

A thread must be machined from the rear end of the barrel, on such barrels, which have a

BARREL FITTING

shank section at the rear machined to a diameter of the root diameter of the thread.

If the rear end of the barrel is slotted for an extractor, this slot must be deepened. This is done while the barrel is still in the lathe, using a tool post grinder, with a grinding wheel narrow enough to enter the extractor slot, or done on a milling machine. It may also be done on the milling machine.

This method of bringing a barrel to its correct place in the receiver when it screws past the register mark can only be used if a chambering reamer is available. After shortening the barrel at the breech end the chamber must be deepened the same amount.

If no chambering reamer is available, instead of shortening the barrel so it can be seated one turn deeper in the receiver. A steel shim must be placed between the barrel shoulder and the part of the receiver against which this shoulder bears. When using steel shims, lubricate it either with oil or colloidal graphite to prevent it tearing under the twisting pressure.

BARREL FITTING

HOLDING THE MATERIAL FOR MACHINING

The stock for machining may be held and rotated using one of several methods. Stock may be held between two centers. Center holes that fit the 60 degree tapered points of hardened steel centers are made with a combination drill and countersink. The drill portion of this cutting tool makes a space at the point of the center holes to provide a clearance and to serve as a reservoir for lubricant. Since the center that is held by the tail stock does not rotate with the work piece, it functions as a bearing. Lubricant should be provided, and excessive pressures should be avoided.

FACE PLATES

Face Plates are used to support irregularly shaped work that cannot be gripped easily in chucks or collets. The work can be bolted or clamped directly on the face plate or can be supported on an auxiliary fixture that is attached to the faceplate. The latter procedure is timesaving when several identical pieces are to be machined.

LATHE CHUCKS

Lathe chucks are used to support a wider variety of work piece shapes and to permit more operations to be performed than can be accomplished when the work is held between centers. Proper care and oiling of the chucks are very important. The manufacturers recommend that light oil should be used on the jaws and threads. This will fly out very quickly as the chuck revolves. A stiff gear grease will stay in the chuck, but the chuck must be taken apart to get the grease on the working parts, and chips tend to stick in the grease.

You should do as the maker advises and use a light oil. Chips collecting in the threads of the chuck can cause much damage. The type of material being machined has a lot to do with the chips accumulating. Cast iron and some of the harder steel alloys crumble when cutting and form small chips and metal dust that will work into the grooves of the jaws and into the screws that actuate the jaws.

After machining a metal that forms these small chips, the chuck must be dismantled and thoroughly washed out with a solvent and oiled as it is reassembled. Chucks are usually attached to the spindle by an accurate thread and a locating taper in conjunction with a cam lock. A work piece chucked in a three-jaw chuck is not gripped as securely as it would be in a four-jaw chuck. Chucks are made with a central hole, which it allows work pieces or bars, if needed, to extend through them into the hollow spindle of the lathe.

Two basic types of chucks are used for working with barrels.

THE FOUR JAW CHUCK

The type in which all the jaws move simultaneously equal distance when the chuck key is turned. This type of chuck usually has three jaws, and in use this chuck automatically locates cylindrical, hexagonal, and other sections concentric with the rotational axis of the spindle. It

BARREL FITTING

is useful for holding bar stock with these sections. The jaws that hold the work piece move in channels in the chuck body toward or away from the center when the chuck key is turned.

Slight deviations from the needed concentric location are sometimes encountered, as the chuck is used and wear develops. The four-jaw chuck has four independent screws that move the jaws. Moving a jaw is accomplished by turning its corresponding screw with the chuck key.

This type of chuck has four jaws, and it is useful for holding irregular shapes, such as castings and forging's, square sections, and many other sections. It is used for holding cylindrical portions of work pieces to be machined not concentric with the gripped surfaces. The jaws can be taken out and reversed to hold either inside or outside work. While the jaws are out, it is possible to wash out the slides and screws with some of the solvent on an old brush.

This is all the dismantling that can be done with the four-jaw chuck. It is hard to clean under the screws so work the brush down under as far as possible. If you have compressed air in your shop, use it to blow out the chips under the screws. An old toothbrush is very good for cleaning the slides. It can be dipped in the solvent and run down the inside of the slides.

Since individual adjustment of each jaw is possible, work pieces can be accurately adjusted with respect to the rotational axis of the spindle. Considerable time is usually required to set up this type of chuck. By using a dial indicator properly supported and in contact with a cylindrical surface, it is possible to adjust the four chuck jaws so that concentricity with the rotational axis of the spindle will be within 0.001 inch or less.

THE THREE JAW CHUCK

The three-jaw chuck has a scroll or spiral thread inside that moves the jaws. As this spiral changes its diameter as the jaws move, the teeth on the inside of the jaws must be curved to fit the small diameter when near the center and to fit the large diameter when near the outer end. The teeth can be curved only one way, so that it is necessary to have a different set of jaws for inside and for outside work.

When the jaws must be changed, screw out the ones not wanted, clean the spiral by revolving it while washing with solvent on a brush, and oil both it and the new teeth. Have a small box or compartment in a convenient drawer for storing the jaws when not in use. Washing them before they are stored will keep them clean. The jaws and corresponding slots are numbered, and the numbers on the jaws and slot must match or else the jaws will not center.

To insert the jaws, turn the spiral until the start of the thread shows in the #1 slot. Back it up a little so that the #1 jaw can enter until it touches the thread, Advance the spiral, while pressing in the jaw to make it engage the spiral. Watch at slot #2 until the end of the spiral appears in the slot; repeat with #2 jaw and #3.

Wind in on the spiral until all the jaws come even with the outside of the body of the chuck. At this point they should all be flush with the outside of the chuck. This is a test to see if the jaws are properly entered and centered. If one of them is not flush, you must wind them all out and

BARREL FITTING

start over again.

Never use a chuck wrench bigger than the one furnished with the chuck. The manufacturer has determined the maximum size it is safe to use with that particular chuck, so use only that size and don't take a chance of wining the chuck by over straining it. For the tedious job of winding out the jaws when changing them, you can make a speed wrench that is a big help.

Get a piece of round, cold-rolled steel that is big enough to file a square on one end that will fit the square socket in the screws of the chuck. Then bend it to a U-shaped crank similar to that of a carpenter's bit brace. This gives you a handle to hold with one hand while winding with the other.

If the sockets in the screws of the four-jaw chuck are the same as for the three-jaw chuck, you need only one speed wrench, but if they are not the same, make one for each chuck. Be sure that the arm having the square end for the four-jaw chuck is long enough to reach over the jaw when it is being removed and is in the extreme out position. If this arm is too short, the wrench will strike the jaw before it is screwed all the way out, and you must change to the regular 'T' chuck wrench to finish removing and inserting the jaw.

When putting either chuck on the spindle, make sure it has been cleaned from the threads inside the chuck and on the nose of the spindle. A good stiff brush will remove any loose chips, but when a chip sticks in the threads it must be pried out with a sharp-pointed tool.

If the chips have much oil on them, it will be necessary to dip the brush in solvent in order to remove them. Put a drop of oil on the spindle threads before screwing on the chuck. Always see that the chuck is turned uptight against the shoulder on the spindle. Use a suitable size wrench on one of the jaws to set it tight.

If it is not tight, the chuck may spin off when the lathe is suddenly stopped or reversed. Do not use the chuck wrench in one of the screws to unscrew the chuck, as it is a bad practice. It will wear the socket and will eventually ruin it. Use a wrench or pry-bar on the jaws to turn the chuck off. To lock the spindle while screwing a chuck or face plate on or off, throw in the back gears while the cone pulley latch is engaged.

This locks the spindle, but, if too much force is used, it will throw the back gears out of mesh, and they must be held in. A chuck should never set up so tight that this will happen, but sometimes heavy cuts will set one on tight, and it must be forced off.

When centering work in the four-jaw chuck, as you approach the final setting, only very small movements of the chuck wrench are used, slacking off on the low side and tightening the high side, revolving the work half the amount of the error shown on the dial indicator.

LATHE CENTERS

Stock that is long with respect to their diameters usually is machined between centers. Two

BARREL FITTING

centers are used, one in the spindle hole and the other in the hole in the tail stock quill. Two types are used. The solid or dead center, and is made of hardened steel with a Morse taper on one end so that it will fit into the spindle hole. The other end is ground to a 60 taper; sometimes the tip of this taper is made of tungsten carbide to provide better wear resistance. Before a center is placed in position, the spindle hole needs to be wiped carefully to make sure that no dirt or chips are in the hole. The presence of dirt, etc., will prevent the center from seating properly and it will not be aligned accurately.

Before a work piece can be mounted between lathe centers, a 60-degree 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.

The work and the center at the head stock end rotate together, so no lubricant is needed in the center hole at this end. However, because the center in the tail stock quill is dead with respect to the rotating work piece, 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 work piece center hole and the center, and inaccuracy and serious damage may occur.

Proper tightness must be maintained between the centers and the work piece. The work piece must rotate freely, yet no looseness should exist. Looseness will be first noticed by chattering of the material during cutting. Tightness of the centers should be checked after cutting has been done for a short time. The resulting heating and expansion of the work piece will increase the tightness.

Live Centers are sometimes used in place of the dead center in the tail stock quill. The end that fits into the work piece 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.

A connection must be provided between the spindle and the work piece to cause it to rotate. This is accomplished by a lathe dog and a faceplate. The dog is a forging that fits over the end of the work piece 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.

As the work piece is machined, it may expand so much, because of the heat created, that it will be necessary to back off the tail stock center dead center and readjust to obtain a proper pressure between the center hole and the center. Machining cuts are made, if possible, with the feed directed toward the head stock rather than toward the tail stock. A work piece held between the centers is rotated by a dog clamped at one end. Center holes establish the axis about which the work piece rotates. Since center holes serve to locate, work pieces with center holes may be readily transferred from one machine to another.

BARREL FITTING

Centers have shanks with self-holding tapers, and they fit accurately into the tapered holes in the spindle and tail stock 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.

STEADY AND FOLLOW RESTS

A follower rest is used in preference to a steady rest for certain work pieces, since it has some advantages. The steady rest is handy for holding the end of a shaft while drilling and reaming the center in its end. For this purpose, the head end of the shaft is held in the three-jaw chuck, and the tail end is held in the steady rest. The center should be marked on the end of the shaft but not center punched. A steady rest or a follower rest is used for two reasons.

1. To prevent the work piece from bending away from the cutting tool, so that the needed accuracy will be obtained.
2. To prevent vibration, or chattering, with a resulting poor surface smoothness. A steady rest may also be used to support the outer end of a long work piece, which is held at its other end by a chuck.

If you turn a long, slender piece between centers, the radial force exerted by the cutting tool, or the weight of the work piece itself, may cause it to be deflected out of line. Steady and/or follower rests provide means for supporting such work between the head stock and the tail stock. The steady rest is clamped to the lathe ways and has three movable fingers that are adjusted to contact the work and align it.

The follow rest when used is bolted to the lathe carriage. It has two contact fingers that are adjusted to bear against the work piece, opposite the cutting tool, so as to prevent the work from being deflected away from the cutting tool by the cutting forces.

A light cut should be taken before adjusting the finger's to provide a smooth contact-surface area. To set up the work piece for the center drill, the shaft has to be marked. A small prick-punch mark is all that can be used. The steady rest guides are adjusted until the mark on the end of the shaft runs true with the point of the tail center. The punch mark on the end of the shaft used for setting up must run perfectly true.

If it is off-center at all, the shaft will run concentric with the guides of the steady rest, and the punch mark will wobble in a little circle.

The pilot drill of the center reamer will try to follow this punch mark, and if it is large enough to grip the point of the drill, the side pressure may break the pilot drill.

The punch mark must be small, just large enough to be seen. With the guides of the steady rest adjusted, the end of the shaft can be faced off, drilled, or turned. This will make a hole large enough to allow the drill point of the reamer to clear and not cut. The reamer can then

BARREL FITTING

be used to ream the center true with the steady rest. The reamer must be held in a chuck that is steady enough so that it will not spring out of line as the drilled hole wobbles.

If the chuck is not rigid enough, true up the drilled hole with a tool in the tool post of the carriage, and when it is running true, finish-ream it to a 60 ° angle of the center. If there is an error in the punch mark on the end of the shaft, do not try to drill it with the combined drill and center reamer, since you may break the drill. Use a larger drill that is long enough to be flexible and follow the punch mark.

A steady rest also can be used in place of the tail stock as a means of supporting the end of long pieces, having too large an internal hole to permit using a regular dead center, or work where the end must be open for boring. In such cases the head stock end of the work must be held in a chuck to prevent it's moving longitudinally, and tool feed should be toward the head stock. A shaft that is not round cannot be set up in the steady rest. The jaws must have a cylindrical surface to run on.

Try setting up the shaft in the four-jaw chuck in a reversed position, with the end requiring the cylindrical place next to the chuck. The outer end can be steadied by a center drilled accurately at the center of the shaft. It is now possible to turn the shaft round and running true to it's setting in the chuck. This round place should be close to the chuck and should be wide enough to take the jaws of the steady rest.

The shaft can now be reversed end for end, with the turned part resting in the steady rest and the other end accurately centered in the four-jaw chuck. The center reamed to steady the outer end at the first setting will not be accurate with the second setting in the four-jaw chuck, and so, when the work is completed, it should be corrected by again reversing the ends in the lathe and truing up the center while it runs true in the steady rest. If this end cannot be set up in the steady rest, the temporary center should be removed or destroyed, since it cannot be reused without spoiling the work.

You can use a steady rest with an irregular shape is to fit a sleeve made from a short piece of pipe over the work. This sleeve will bear on the jaws of the steady rest, and, by the use of setscrews tapped through the pipe; the shaft can be centered in the sleeve. There should be four setscrews at each end of the sleeve for a round shaft or three at each end for a hexagon bar.

BARREL FITTING

BARREL TURNING

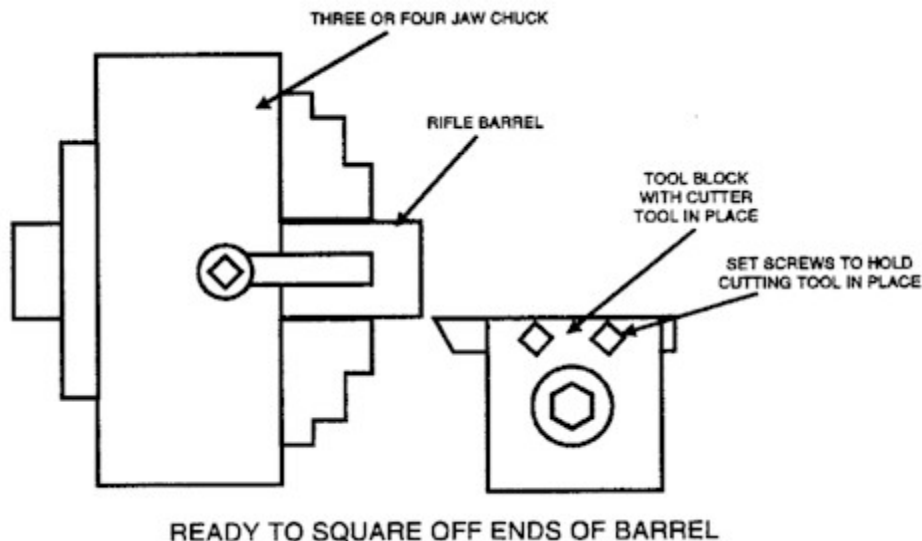
Now we will take the barrel and turn it into a finish rifle barrel. I will try to cover the process step by step up to the finish. I will assume that this barrel is a 308 caliber. I will give you instructions for a 24-inch barrel. The first thing to do is cut the barrel to 24 ½ inches long. You will need the extra ¼ inch in truing up each end, and the final finishing of the barrel.

You have the barrel cut to 24½ inches, so now chuck the barrel in the lathe and face off both ends of the barrel. If the bore has run quite a bit off center, you will need to repeat this process after a few passes on the O.D.

The reason for this is that if the angle on the end of the barrel is off, one side of the barrel will be thicker than the other. In turning, the barrel will probably warp. Now that the barrel is squared, put the faceplate on the lathe. Take a bar of steel that is 24 inches long that has centers in each end and has been turned true.

FACING

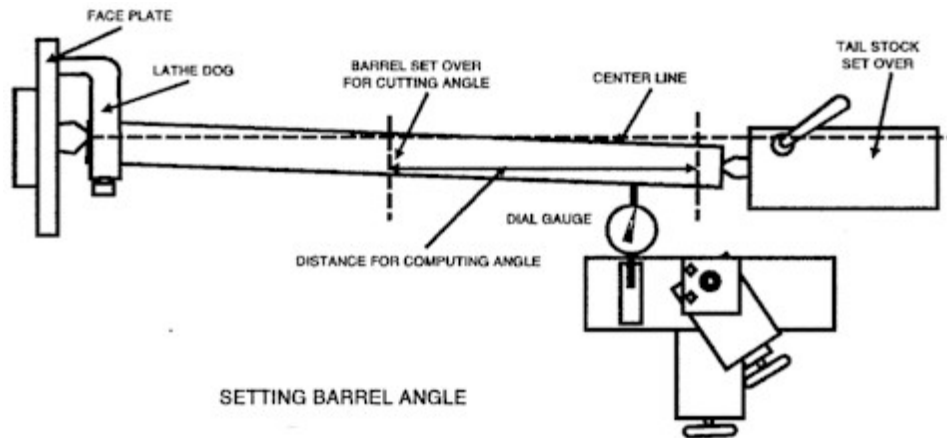
Facing is the producing of a flat surface as the result of the tool being fed across the end of the rotating work piece. The work may be held in a chuck, on a faceplate, or between centers. Unless the work is held on a mandrel, if both ends of the work are to be trued, it must be turned end for end after the first end is completed and the facing operation repeated.



Because most facing operations are performed on surfaces that are away from the head stock, 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. In either case, 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 slightly and thus producing a surface that is not flat.

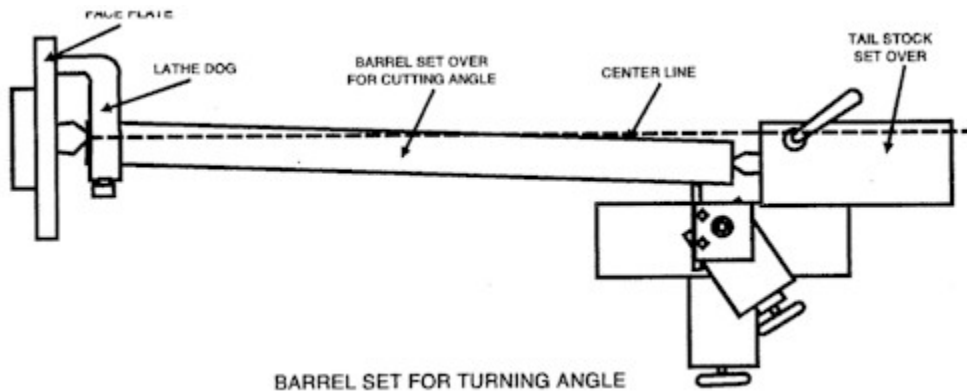
BARREL FITTING

When facing castings or other materials that have a hard surface, the depth of the first cut should be sufficient to penetrate the hard material to avoid excessive tool wear.



LATHE CHUCKS

Lathe chucks are used to support a wider variety of work piece shapes and to permit more operations to be performed than can be accomplished when the work is held between centers. Proper care and oiling of the chucks are very important. The manufacturers recommend that a light oil should be used on the jaws and threads. This will fly out very quickly as the chuck revolves. A stiff gear grease will stay in the chuck, but the chuck must be taken apart to get the grease on the working parts, and chips tend to stick in the grease.



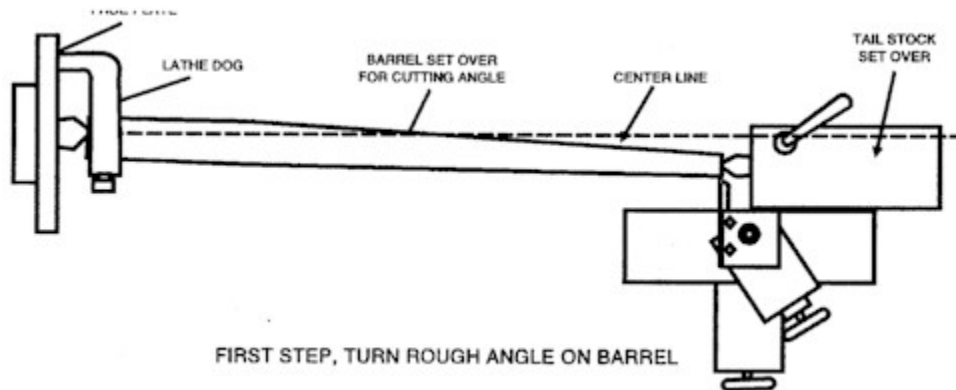
You should do as the maker advises and use a light oil. Chips collecting in the threads of the chuck can cause much damage. The type of material being machined has a lot to do with the chips accumulating. Cast iron and some of the harder steel alloys crumble when cutting and form small chips and metal dust that will work into the grooves of the jaws and into the screws that actuate the jaws.

After machining a metal that forms these small chips, the chuck must be dismantled and thoroughly washed out with a solvent and oiled as it is reassembled. Chucks are usually attached to the spindle by an accurate thread and a locating taper in conjunction with a cam lock.

BARREL FITTING

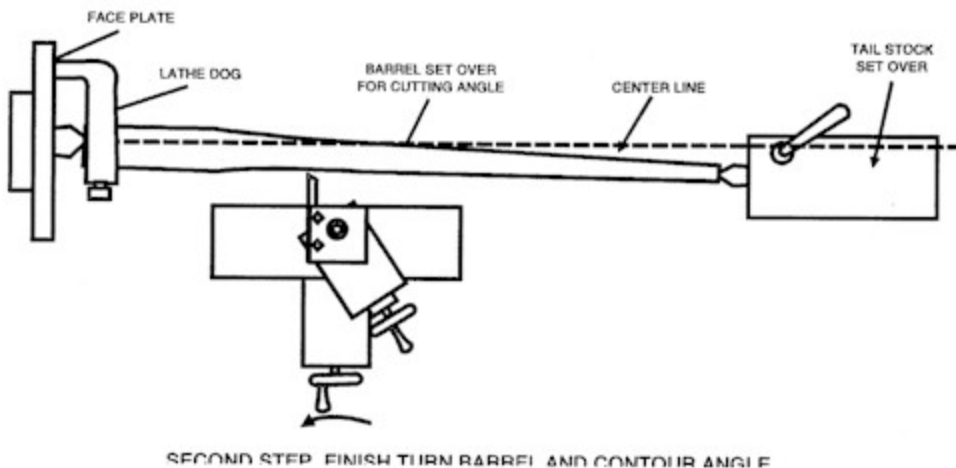
A work piece chucked in a three-jaw chuck is not gripped as securely as it would be in a four-jaw chuck. Chucks are made with a central hole, which it allows work pieces or bars, if needed, to extend through them into the hollow spindle of the lathe.

Two basic types of chucks are used to hold the work.



THE FOUR JAW CHUCK

The type in which all the jaws move simultaneously equal distance when the chuck key is turned. This type of chuck usually has three jaws, and in use this chuck automatically locates cylindrical, hexagonal, and other sections concentric with the rotational axis of the spindle. It is useful for holding bar stock with these sections. The jaws that hold the work piece move in channels in the chuck body toward or away from the center when the chuck key is turned.



Slight deviations from the needed concentric location are sometimes encountered, as the chuck is used and wear develops. The four-jaw chuck has four independent screws that move the jaws. Moving a jaw is accomplished by turning its corresponding screw with the chuck key.

This type of chuck has four jaws, and it is useful for holding irregular shapes, such as castings and forging's, square sections, and many other sections. It is used for holding cylindrical portions of work pieces to be machined not concentric with the gripped surfaces.

BARREL FITTING

The jaws can be taken out and reversed to hold either inside or outside work. While the jaws are out, it is possible to wash out the slides and screws with some of the solvent on an old brush.

This is all the dismantling that can be done with the four-jaw chuck. It is hard to clean under the screws so work the brush down under as far as possible. If you have compressed air in your shop, use it to blow out the chips under the screws. An old toothbrush is very good for cleaning the slides. It can be dipped in the solvent and run down the inside of the slides.

Since individual adjustment of each jaw is possible, work pieces can be accurately adjusted with respect to the rotational axis of the spindle. Considerable time is usually required to set up this type of chuck. By using a dial indicator properly supported and in contact with a cylindrical surface, it is possible to adjust the four chuck jaws so that concentricity with the rotational axis of the spindle will be within 0.001 inch or less.

THE THREE JAW CHUCK

The three-jaw chuck has a scroll or spiral thread inside that moves the jaws. As this spiral changes its diameter as the jaws move, the teeth on the inside of the jaws must be curved to fit the small diameter when near the center and to fit the large diameter when near the outer end. The teeth can be curved only one way, so that it is necessary to have a different set of jaws for inside and for outside work.

When the jaws must be changed, screw out the ones not wanted, clean the spiral by revolving it while washing with solvent on a brush, and oil both it and the new teeth. Have a small box or compartment in a convenient drawer for storing the jaws when not in use. Washing them before they are stored will keep them clean. The jaws and corresponding slots are numbered, and the numbers on the jaws and slot must match or else the jaws will not center.

To insert the jaws, turn the spiral until the start of the thread shows in the #1 slot. Back it up a little so that the #1 jaw can enter until it touches the thread, Advance the spiral, while pressing in the jaw to make it engage the spiral. Watch at slot #2 until the end of the spiral appears in the slot; repeat with #2 jaw and #3.

Wind in on the spiral until all the jaws come even with the outside of the body of the chuck. At this point they should all be flush with the outside of the chuck. This is a test to see if the jaws are properly entered and centered. If one of them is not flush, you must wind them all out and start over again.

Never use a chuck wrench bigger than the one furnished with the chuck. The manufacturer has determined the maximum size it is safe to use with that particular chuck, so use only that size and don't take a chance of ruining the chuck by over straining it. For the tedious job of winding out the jaws when changing them, you can make a speed wrench that is a big help.

Get a piece of round, cold-rolled steel that is big enough to file a square on one end that will fit the square socket in the screws of the chuck. Then bend it to a U-shaped crank similar to that of a carpenter's bit brace. This gives you a handle to hold with one hand while winding

BARREL FITTING

with the other.

If the sockets in the screws of the four-jaw chuck are the same as for the three-jaw chuck, you need only one speed wrench, but if they are not the same, make one for each chuck. Be sure that the arm having the square end for the four-jaw chuck is long enough to reach over the jaw when it is being removed and is in the extreme out position. If this arm is too short, the wrench will strike the jaw before it is screwed all the way out, and you must change to the regular 'T' chuck wrench to finish removing and inserting the jaw.

When putting either chuck on the spindle, make sure it has been cleaned from the threads inside the chuck and on the nose of the spindle. A good stiff brush will remove any loose chips, but when a chip sticks in the threads it must be pried out with a sharp-pointed tool.

If the chips have much oil on them, it will be necessary to dip the brush in solvent in order to remove them. Put a drop of oil on the spindle threads before screwing on the chuck. Always see that the chuck is turned up tight against the shoulder on the spindle. Use a suitable size wrench on one of the jaws to set it tight.

If it is not tight, the chuck may spin off when the lathe is suddenly stopped or reversed. Do not use the chuck wrench in one of the screws to unscrew the chuck, as it is a bad practice. It will wear the socket and will eventually ruin it. Use a wrench or pry-bar on the jaws to turn the chuck off. To lock the spindle while screwing a chuck or face plate on or off, throw in the back gears while the cone pulley latch is engaged.

This locks the spindle, but, if too much force is used, it will throw the back gears out of mesh, and they must be held in. A chuck should never set up so tight that this will happen, but sometimes heavy cuts will set one on tight, and it must be forced off.

When centering work in the four-jaw chuck, as you approach the final setting, only very small movements of the chuck wrench are used, slacking off on the low side and tightening the high side, revolving the work half the amount of the error shown on the dial indicator.

You will need a trued barrel for each length of barrel you turn. What this is for is to give you a means to duplicate any taper that you want. Example, if you found that the old barrel that you wanted to duplicate, had a taper of .125 from the muzzle, to 12 inches from the muzzle.

Go to the taper per foot chart. In the taper Per Foot column at the top, find 1/8. Follow this column down, and when you come to where the left column says – Length of Tapered Portion, at 1 foot, or 12 inches. Where these two meet find a figure of 0.0104. Now mark the 24-inch bar of steel exactly 12 inches from the tail stock end, to a point in the middle of the barrel. This is the distance you will need to travel with the dial indicator.

Set the dial indicator on the carriage of the lathe, set it to 000, and set the tail stock over a bit. Crank the carriage down towards the head stock to where the mark is on the barrel.

BARREL FITTING

Check the reading on the dial indicator, and if it is not right, reset the tail stock and repeat the process until you get 0.0104.

It would be wise to keep all of these figures on notes, or stamp the barrel with the correct figures, for future reference.

FACE PLATES

Face Plates are used to support irregularly shaped work that cannot be gripped easily in chucks or collets. The work can be bolted or clamped directly on the face plate or can be supported on an auxiliary fixture that is attached to the faceplate. The latter procedure is timesaving when several identical pieces are to be machined.

Having found the correct angle, set the barrel blank between the centers, and clamp a lathe dog on the head stock end. The tail stock center, I have found over the years work better if it is carbide. If you are careful, you can use the standard high-speed center. Put a little grease on it that contains MolyKote Z before setting it up.

I have tried to use live centers off and on, but I have found that I get too much chatter, and that will cause stress to build in the barrel, and thus warp the barrel.

It is very important to keep any stress from build up in the barrel (Live centers, too tight centers, dull tool bit, improperly sharpened tool bit, etc.) Now for the tool bit. I have used carbide most all the time in turning barrels. I prefer a good grade of high-speed steel as they are sharper and cut better, but require sharpening every pass, to keep a good edge. I had to use carbide due to the time saved.

I ground all my tools to the general shape. On high speed bits I used only about .010-nose radius, stoned on by hand. Set the tool up so it is about .010 above the tail stock center. If I have the time, I usually use a feed of .004, but you will need to experiment to see which feed works the best.

Usually I do not take over .020 passes per each side on each pass.

STEADY AND FOLLOWER RESTS

A follower rest is used in preference to a steady rest for certain work pieces, since it has some advantages. The steady rest is handy for holding the end of a shaft while drilling and reaming the center in its end. For this purpose, the head end of the shaft is held in the three-jaw chuck, and the tail end is held in the steady rest. The center should be marked on the end of the shaft but not center punched. A steady rest or a follower rest is used for two reasons.

1. To prevent the work piece from bending away from the cutting tool, so that the needed accuracy will be obtained.

BARREL FITTING

2. To prevent vibration, or chattering, with a resulting poor surface smoothness. A steady rest may also be used to support the outer end of a long work piece, which is held at its other end by a chuck.

If you turn a long, slender piece between centers, the radial force exerted by the cutting tool, or the weight of the work piece itself, may cause it to be deflected out of line. Steady and/or follower rests provide means for supporting such work between the head stock and the tail stock. The steady rest is clamped to the lathe ways and has three movable fingers that are adjusted to contact the work and align it.

The follow rest when used is bolted to the lathe carriage. It has two contact fingers that are adjusted to bear against the work piece, opposite the cutting tool, so as to prevent the work from being deflected away from the cutting tool by the cutting forces.

A light cut should be taken before adjusting the finger's to provide a smooth contact-surface area. To set up the work piece for the center drill, the shaft has to be marked. A small prick-punch mark is all that can be used. The steady rest guides are adjusted until the mark on the end of the shaft runs true with the point of the tail center.

If the chuck is not rigid enough, true up the drilled hole with a tool in the tool post of the carriage, and when it is running true, finish-ream it to a 60 angle of the center. If there is an error in the punch mark on the end of the shaft, do not try to drill it with the combined drill and center reamer, since you may break the drill. Use a larger drill that is long enough to be flexible and follow the punch mark.

A steady rest also can be used in place of the tail stock as a means of supporting the end of long pieces, having too large an internal hole to permit using a regular dead center, or work where the end must be open for boring.

In such cases the head stock end of the work must be held in a chuck to prevent it's moving longitudinally, and tool feed should be toward the head stock. A shaft that is not round cannot be set up in the steady rest. The jaws must have a cylindrical surface to run on.

What you have to watch for is that the cut is not heavy enough that it will cause the barrel to spring, especially on the barrel that is half way turned down. This springing causes stress. The tool must cut clean, for if it drags, or it is slightly dull it will cause the barrel to heat as it is turned. When the bit is sharpened right you will get some heat build up, but not much. Watch the chips, as they are coming off the barrel. If they turn a dark blue, you are taking too much a cut, or the tool is dull.

Keep close of the pressure on the tail stock. It must be snug on the barrel, but not tight. As the barrel heats its lengthens, and the tail stock must be loosened. If the tail stock center loosens too much, the tool bit will chatter. After the contour is turned up to where the shoulder will end, you will need to remove the barrel. You will then check to see if the bore is straight, and straighten if necessary.

If you have kept a sharp tool and a light feed, it should be straight. You must, after ever pass

BARREL FITTING

from now on the barrel should be checked to see if it is straight. If the barrel has warped, take a lighter pass after it is straighten.

When you start cutting the contour of the shoulder, unless you have a tracer attachment for your lathe, you will have to cut it by hand. This may sound hard, but with a little practice you will get a perfect contour. To this, as you get where the contour starts, you will start to back the feed out by hand. With pass put a little more contour to the shoulder.

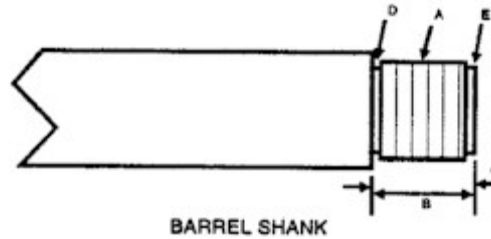
Remember to take the shoulder back a little farther than where it should be when finish, as you have to turn down the shank of the barrel. If the barrel should start to chatter when turning, lay a barrel blank on the barrel behind the cutter and that will usually stop the chatter.

When you have finish turning the barrel down to within a few thousands of the finish size, set the feed to the lightest feed. Sharpen the tool bit, hone it very good, and make a finish pass.

BARREL FITTING

THREAD DIMENSIONS

All of the thread sizes shown here were taken from the workbook that we made up as we threaded new barrels, and came directly from the old barrel. They may or may not agree with some of the factory specifications found in some books. The sizes given below were what we cut on the new barrels and reinstalled back on the old action.



ACTION	A	B	C	D	E	THREADS
STEVENS	.875	1.425	.500			16V
KRAG M 92	1.050	.700				12LHSQ
FV 10 SEMIAUTO	1.058	.825	.100	.100		18V
MARLIN 1894	.780	.725	.100			12SQ
WIN 43	.930	.840				RECS16V
WIN HIGH WALL	.935	.850				RECS16V
JAP 7.7	1.050		.700	.125		17V
MANLKR. STEYER	1.060		.600	.125		12V
REM.ROLL.BLK	1.053		1.400	.145	.425	12SQ
MARTINE SMALL	.745		.625			14V
SAKO L 461	.860		.825	.100		16V
SWEDISH MAUSER	.990		.620	.100	.100	12V
600 REM.	1.058		.875			16V
STEVENS 322	.875		1.950	.100		18V

BARREL FITTING

ACTION	A	B	C	D	E	THREADS
SAKO L 579	1.00		.780		.150	16V
WIN MOD 70	1.00		.740		.100	16V
SAVAGE 99	.900		.875		.125	12SQ
SAVAGE 110	1.065		.756	W/N	.312	20V
PERU MAUSER	.998		.835		.125	12V
REM. ROL.BLK	1.430		1.00	1.055	.430	12SQ
WIN 95	.933		.750	.125		16V
SAKO RIHIMAK	.862		.862	.845	.100	16V
MAUSER G33/40	.980		.635	.100	.100	12V
M1 GARAND	.970		.700	.100	.100	12SQ
WIN 92	.805		.635	.100		20V
SA COLT 45	.708		.707		.124	20VTAP
BRNO MAUSER	1.100		.625	.100	.100	12V
M 30 REM	1.115		.800	.100	.100	CON10SQ
93 MAUSER	.985		.625			12V
RUGER NO 1	1.00		.850		.160	16V
JAP ARISKA	1.017		.720	.125	.125	14V
SAVAGE	.870		.725			18V
REM 722	1.060		.875			16V
WIN 86	.880		.730			20V
SHUTZ & LARSON	1.040		.982		.100	14V
BSA HERTER	1.122		.980	.905	.200	12V
6.5 ITAL.MAUS.	1.062		.762	.120	.120	14V
TEX MAG.	1.00		1.100	.120		14V
98 MAUSER	1.100		.635	.100	.100	12V
SR MAUSER	.980		.635	.100	.100	12V
SPRINGFIELD	1.040		.740		.100	CON10SQ
ENFIELD	1.115		.800		.100	CON10SQ
WW GREENER	1.080		.690		.120	14V
LEE ENFIELD	.995		.685		.120	14V
WIN 43	.805		.641		.100	20V
SHARP B	.940		1.100	.100	.100	16V
BRNO	.990		.600	.100	.100	20V
MARTINI	1.00		.675	.100	.120	14V
MANLIKR.G98/40	1.055		.570	.100	.100	12V
WIN 94	.809		.504		.100	20V
SAKO L61	1.100		.695		.100	16V
SAKO L61 MAG.	1.050		.695		.100	16V

BARREL FITTING

ACTION	A	B	C	D	E	THREADS
SAV. MOD 20	.895		.930		.120	12SQ
KRAG JORG.	.980		.750	.100	.100	10SQ
REM XP 100	1.060	.900	.875		.340	RECS16V
REM 740	.874		.330	.551	.100	RECS20V
WTB MK V	1.055		.700		.130	RECS16V
WIN 670	1.00		.700		.100	16V
SAVAGE 99 F	.900		.920		.125	RECS14V
REM ROL.BLK.						
SMALL SHANK	.975		1.425		.350	12SQ
MANLICKE GRK	1.065		.580	.100	.100	12V
REM 40X	1.045	.995	1.650		.875	RECS16V
WIN 88	1.00		.715		.100	16V

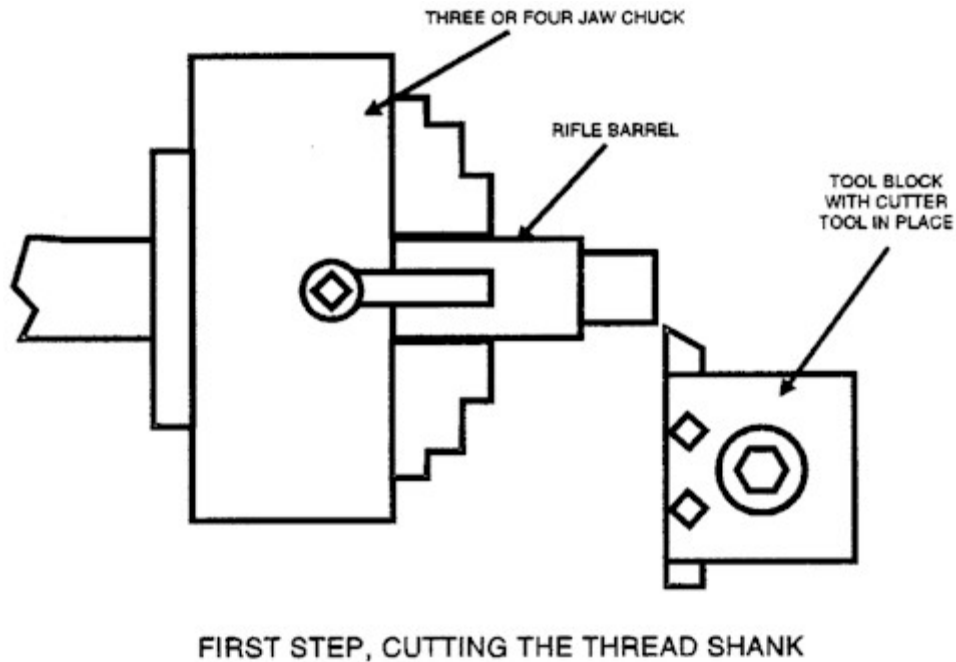
NOTE— The above dimensions were taken directly from the barrel and not from factory specifications.

BARREL FITTING

THREAD CUTTING

TEST BAR

Before you start the threading 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 tail stock 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 tail stock. 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 head stock 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 work piece 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.

THREAD-CUTTING TOOLS AND THREAD FORMS

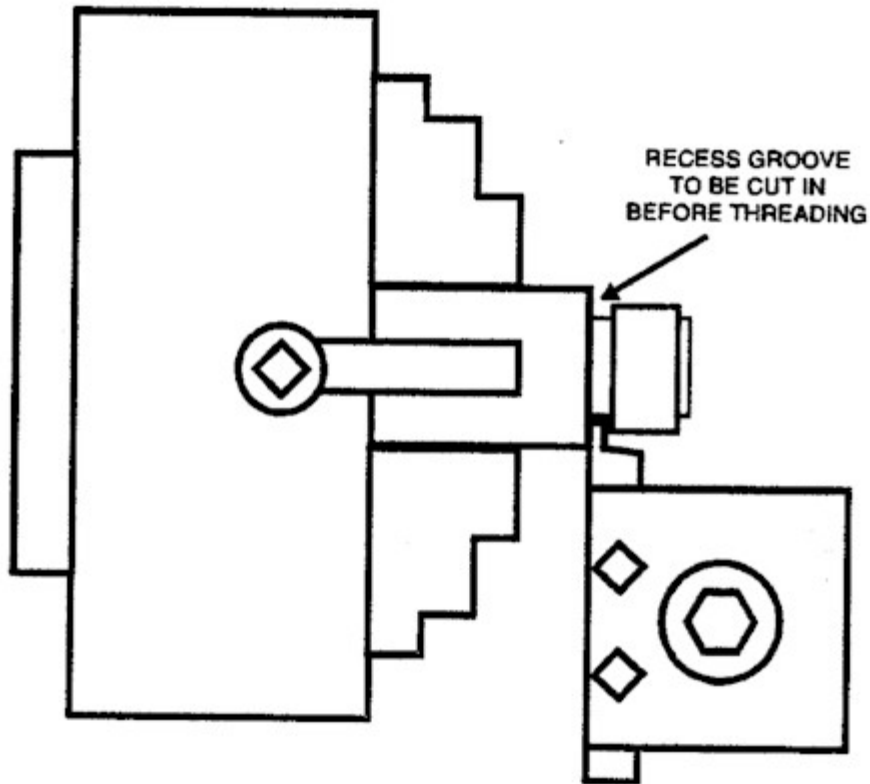
For each of the types of thread a tool bit to form that particular thread must be ground. Since the form of thread is the end product, the grinding of the tool bit must take into consideration

BARREL FITTING

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 work piece, 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 degrees of side clearance.



CUTTING RECESS GROOVE IN SHANK FOR CLEARANCE

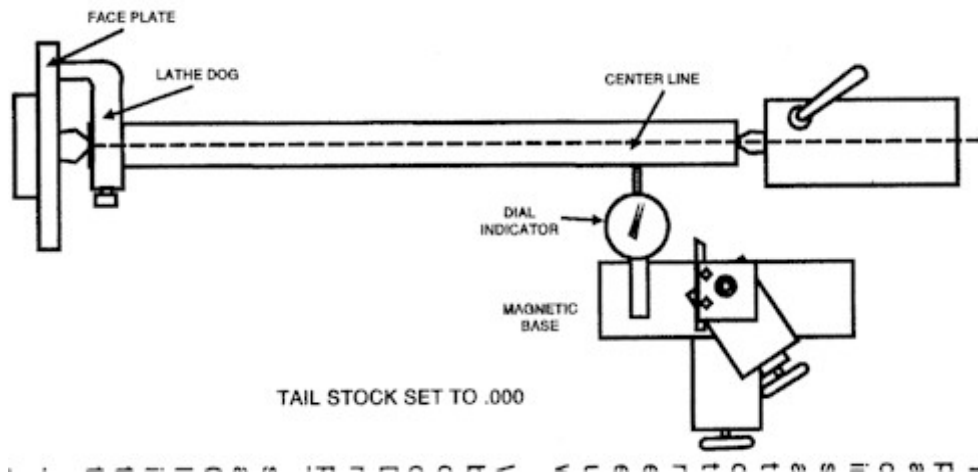
V thread tools are ground flat across the top, with about a 5-degree side clearance angle. If the top of the tool is at an angle, the thread will be incorrect.

CENTER GAGE FOR CHECKING CUTTING TOOLS.

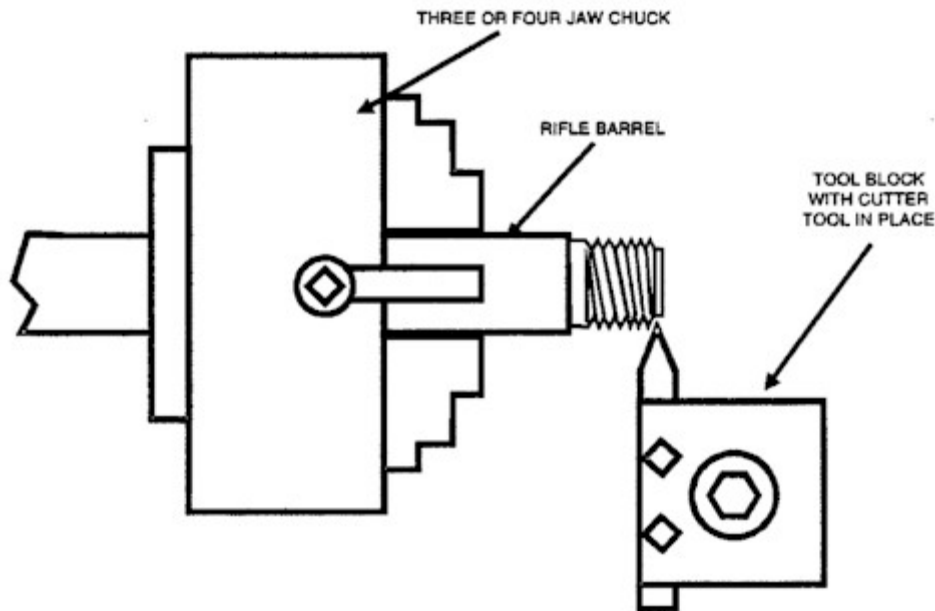
The center gage is a tool used for checking the points of cutter bits for the American standard screw threads that must be ground to an angle of .60 degrees. The 60-degree included angle on one end of the gage is used when grinding the tool to the exact angle. This same tool is useful in checking the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

BARREL FITTING



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



SECOND STEP, THREADING THE THREAD SHANK

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 used 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 centerline of each thread will be perpendicular to the rotational axis of the work piece.

BARREL FITTING

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 degrees instead of to 30 degrees.

BARREL FITTING

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 gearboxes 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 work piece by adjusting the cross-slide 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 work piece and the carriage is moved to where the tool is, at a point beyond the end of the work piece. The cross-slide is then advanced until the micrometer dial reads the same as where the tool was touching the work piece.

Next, the compound is advanced .002 to .003" and another 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.

BARREL FITTING

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 head stock and set at an angle of 29'.

The tool is set so its centerline is at a right angle to the axis of the work piece. 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 centerline of the work piece.

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 were 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 were 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,

BARREL FITTING

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 uptight 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 run 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 started 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.

BARREL FITTING

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.

BARREL FITTING

THREAD DIAL INDICATOR

Most modern 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.

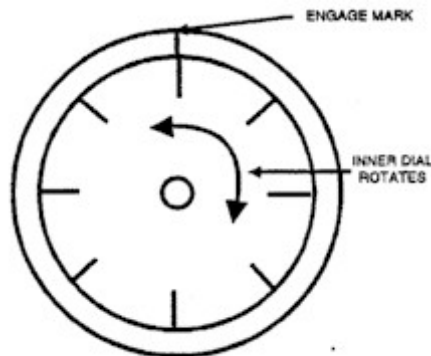
The face of the dial is numbered to show positions at which the half nuts may be engaged. When the lathe is set up for cutting screw threads, the thread dial shows the relative position of the lead screw, spindle, and carriage of the lathe.

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 3 ½ threads per inch, the half-nut must be engaged at any odd-numbered line on the dial.

For quarter-threads, such as 2 ¼ or 3 ¼ threads per inch, the half-nut must be engaged at the same point on the dial each time a cut is started.



THREADING DIAL

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

BARREL FITTING

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.)

BARREL FITTING

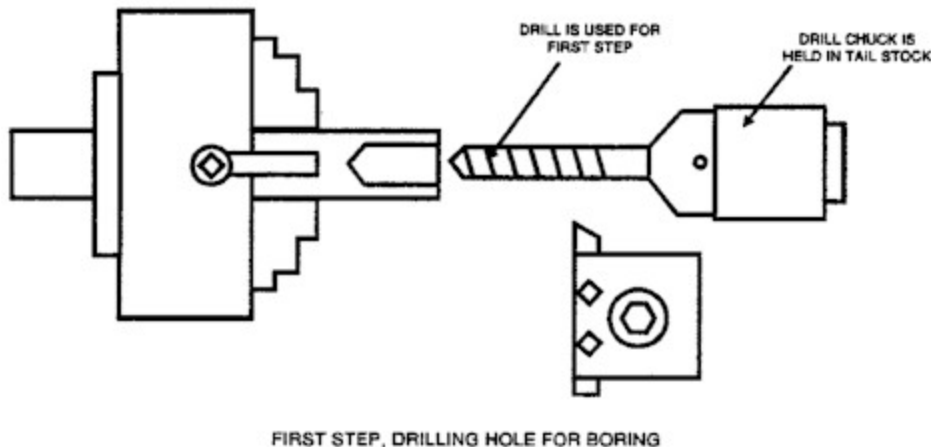
SETTING THE COMPOUND FOR RIGHT AND LEFT-HAND THREADS.

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 head stock.

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 head stock, 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 tail stock 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 EXTERNAL THREADS.

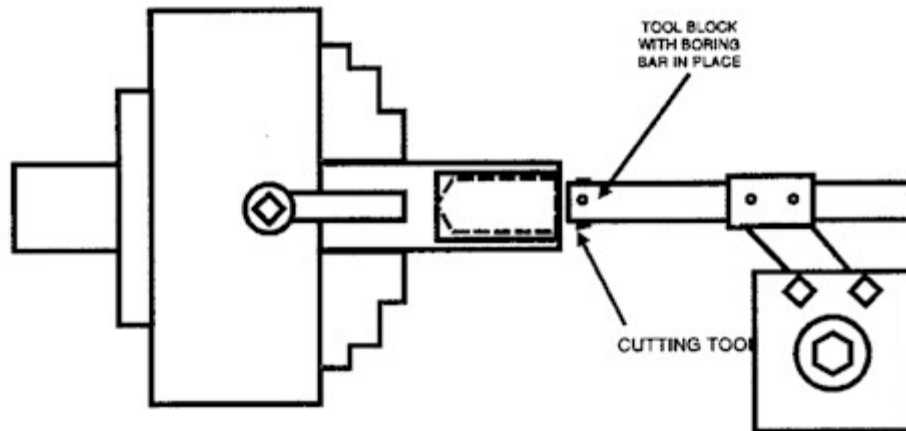
After setting the compound rest and positioning the threading tool properly in relation to the work, it is necessary to select the proper speed and feed. Good thread-cutting practice requires that the back gears be engaged for this operation.

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 gearbox on all modern lathes. Directions are shown on the gearbox 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

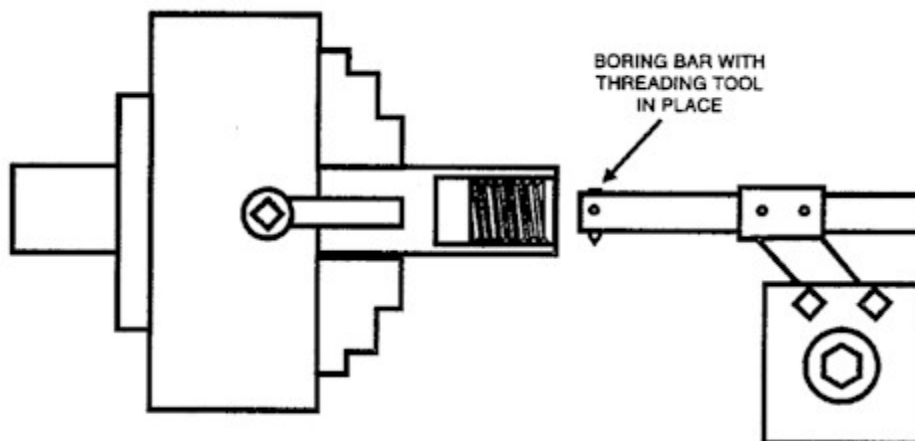
BARREL FITTING

work lightly. Then run the carriage to the right, using the carriage hand wheel, until the tool clears the end of the work. Notice what the setting is on the cross-feed collar.



SECOND STEP, BORING DRILLED HOLE TO SIZE

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 $.002^\circ$ and then start the machine and take the first trial cut.



THIRD STEP, CUTTING THE THREADS

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.

BARREL FITTING

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 degrees 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 work piece. Start the machine and engage the thread-chasing lever as before. Let the work piece 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 work piece 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 degrees. 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 work piece. 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.

BARREL FITTING

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.

BARREL FITTING

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 degrees, 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 hand wheel 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 work piece by using the cross slide hand wheel and graduated dial.

SIZE OF WORK PIECE BEFORE THREADING.

Before the threading operation is begun, the work piece 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 4-5 degrees. 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 work piece. 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 work piece is left right in the lathe and the threading operation is performed. The jobs are threaded

BARREL FITTING

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.

BARREL FITTING

THREAD CHASING

Before you start the threading operation, set the tail stock back to 000 using a 18-inch bar that is turned to the exactly 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 tail stock 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 tail stock. 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 head stock end (without the lathe being turned on) and move the carriage to the tail stock end. If there are any differences in size, adjust the tail stock and repeat until the dial indicator reads the same on each end.

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. In other words, this adjustment will ensure that the centerline of each thread will be perpendicular to the rotational axis of the work piece.

When thread chasing, a number of successive light helical cuts are 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 degrees instead of to 30 degrees.

To move in for depth of cut for some other threads, 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 degrees. 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 work piece. 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

BARREL FITTING

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.

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 work piece by using the cross slide hand wheel and graduated dial.

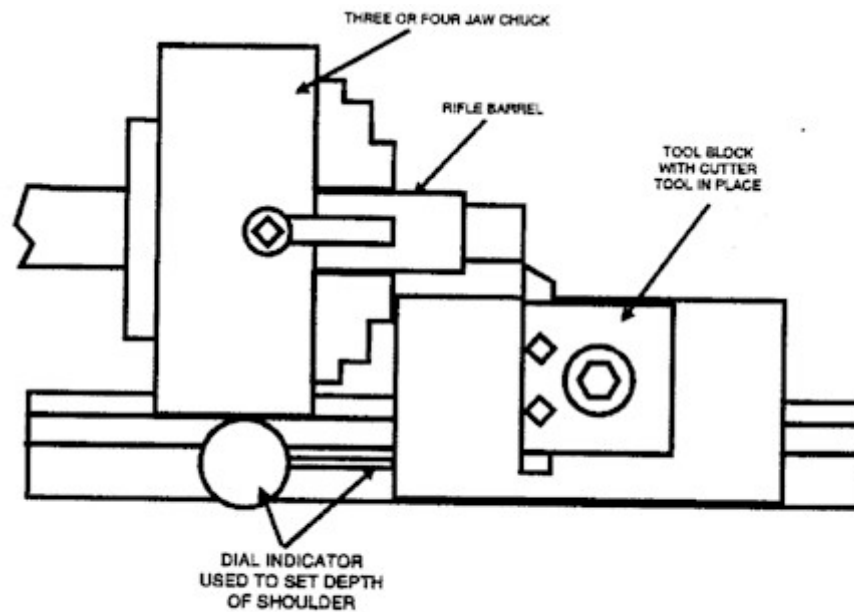
The carriage is then returned to its starting position for another cut. The threading tool is again moved in with the cross slide so that the cross slide is always returned to the same position for each successive cut. Depth of cut is adjusted with the compound rest hand wheel and graduated dial. Another 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.

BARREL FITTING

THREADING AND FITTING

When you have finish turning the barrel, and double-checking it for straightness, it is time to turn the shank. Set the tail stock back to 000, take the lathe dog off and put a smaller one on the muzzle end of the barrel. With a shim under the setscrew to keep the barrel from being marked by the setscrew. Put the barrel back in the lathe, and turn the shank to the correct diameter.

On a 98 Mauser, which we will thread to, take only a light clean up pass. This is assuming that you made your barrel out of 1 1/8 inch stock. It would be best if you have an old 98 Mauser barrel handy to look at, while you are doing this.



FIRST STEP, CUTTING THE THREAD SHANK TO SIZE

Knowing how many threads to the inch are on the barrel, it is easy to figure just how many thousands of an inch the barrel will move in the receiver in making one complete revolution in the threads. After the barrel is chucked in the lathe, bring the facing tool lightly against the end of the barrel, or against the barrel shoulder, and set the micrometer carriage stop against the forward end of the lathe carriage.

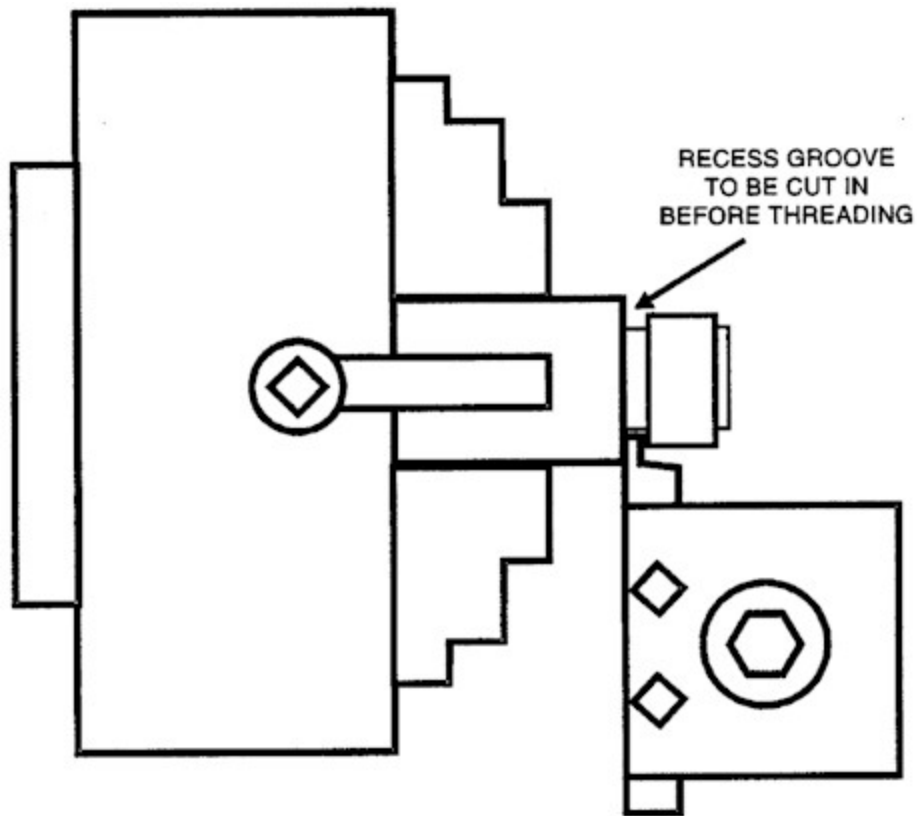
Next, back off the micrometer stop the required number of thousandths of an inch for one revolution of the barrel, less the amount required to make the marks on receiver and barrel meet. This number of thousandths of an inch is then deducted from the number of thousandths of an inch the barrel moves in a full revolution. The answer is the amount the micrometer stop is to be backed away from the lathe carriage.

A thread must be machined from the rear end of the barrel, on such barrels, which have a shank section at the rear machined to a diameter of the root diameter of the thread.

If the rear end of the barrel is slotted for an extractor, this slot must be deepened. This is

BARREL FITTING

done while the barrel is still in the lathe, using a tool post grinder, with a grinding wheel narrow enough to enter the extractor slot, or done on a milling machine. It may also be done on the milling machine.



CUTTING RECESS GROOVE IN SHANK FOR CLEARANCE

If you look at the chart for threading the 98 Mauser, See chapter on barrel thread dimensions, The O.D. of the shank is 1.100. That only leaves less than .020 for the shoulder. This is Ok as on the 98 Mauser we seat the barrel on the inside ring.

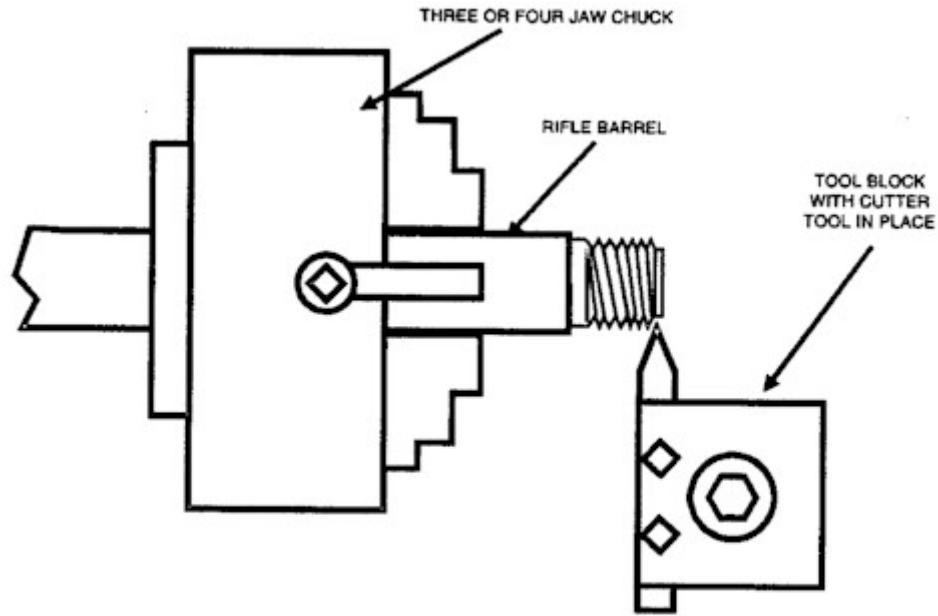
Turn the thread end down to 1.100 now. Using a dial indicator as you are turning cut a .625 long shoulder. When you have the thread shank turned to the correct size, take a small square end tool and relieve the threads on the front end of the shoulder, and the back end of the thread shank. This relief should be about .060 deep per side.

It is time to cut the threads on the 98 Mauser. The 98 has 12 threads per inch V type. Setup 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 degrees, 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. See chart for amount that you will have to turn in to get the proper depth. 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

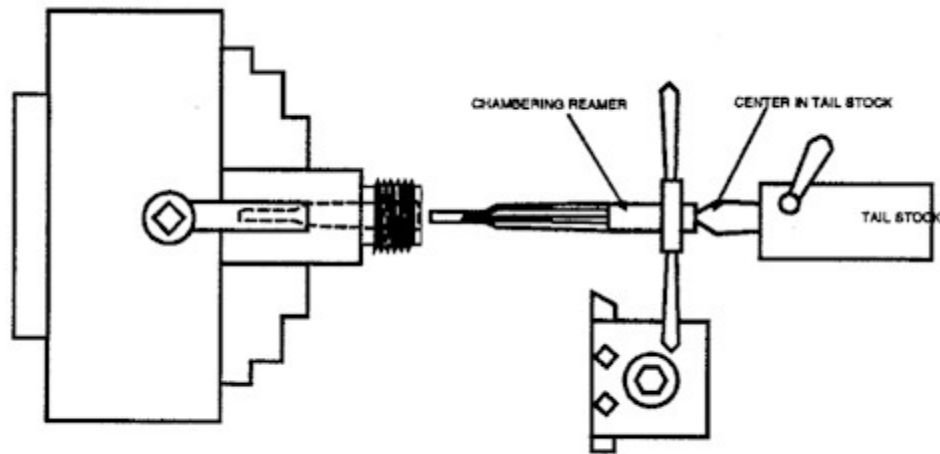
BARREL FITTING

back out the tool in one operation.



SECOND STEP, THREADING THE THREAD SHANK

When it has completed set the tool in .010 and when the dial comes back to 0, engage the feed. Keep repeating the process until you have gone .062; make a 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.



SECOND STEP, CHAMBERING BARREL

BARREL FITTING

FITTING THE BARREL TO THE ACTION

Take the action and see if it will screw on to the barrel. It should but if it doesn't take another pass until it does. **DO NOT TURN ON THE LATHE OR CHANGE ANYTHING UNTIL THE ACTION WILL SCREW ON.**

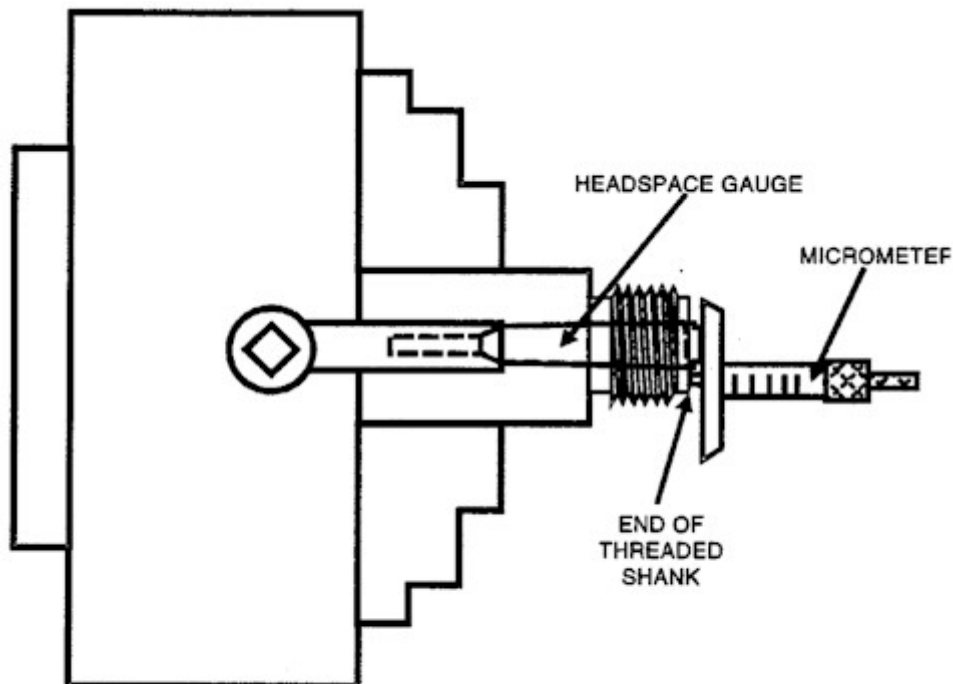
The barrel screws onto the action and butts up against the shoulder on the inside of the action.

Take a depth mike and measure the depth from the front of the action, to the inside ring. Check this against the measurement of .062 that you turned on the thread shank. The thread shank should be .003 longer than the measurement from the face of the action to the ring.

When you tighten the barrel up on the action, you will see the compression marks on the face of the barrel. When you have this the barrel is completed except chambering.

CHAMBERING THE BARREL

The reamers are easily made. If you can make barrel reamers, you can make chambering reamers. What you need now is a 308 Winchester chambering (finish), and head space gauges (go and no go).



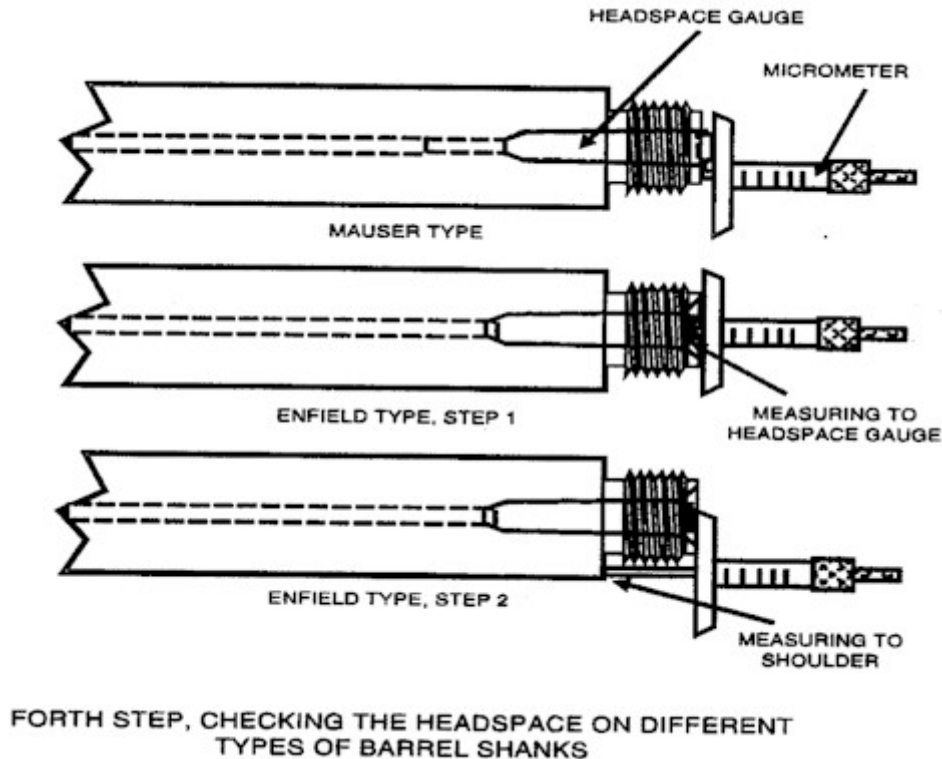
THIRD STEP, CHECKING THE HEADSPACE

You will need a 7/16 drill bit, unless you are planning to use a roughing reamer to remove the extra metal. You will need now to chuck the barrel on the shank in front of the threads. When

BARREL FITTING

using a dial indicator be sure the barrel is running true on both ends.

If it is not running true, you will cut an out of round chamber. Chuck the drill in the tail stock chuck, as you will have to drill out the chamber end about 1.250. This will remove enough metal so the finish reamer won't have to cut much.



Now set a small lathe dog or clamp on the reamer, and with a center in the tail stock, move the tail stock up close enough to the barrel so the reamer will be in the barrel next to the shoulder of the reamer. Have the reamer in the tail stock center. Leave the lathe in back gear setting that you used for threading and with the tail stock fastened down. Slowly feed the reamer into the barrel, keeping plenty of cutting oil on the reamer.

You can rest the lathe dog or clamp on the compound, while feeding in. Be sure that the reamer is on the tail stock center when you feed it in as if it isn't it can grab and break the reamer.

ALWAYS HOLD THE REAMER BACK AGAINST THE TAIL STOCK CENTER.

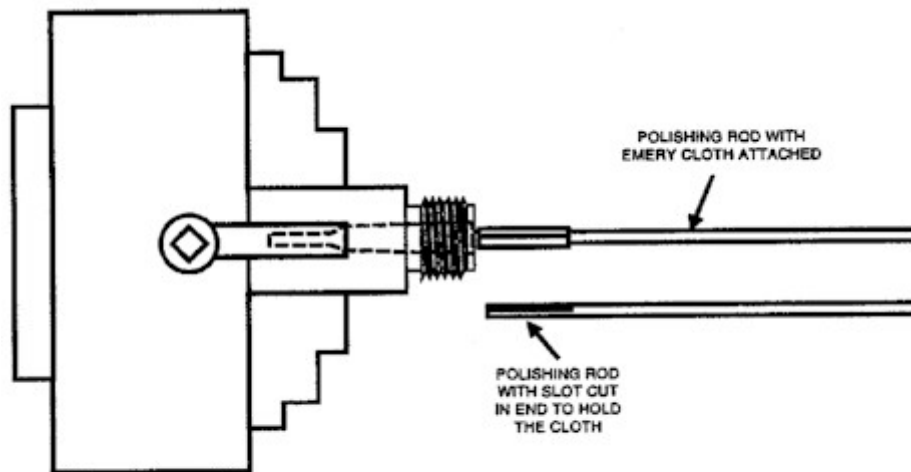
Usually at first you will be able to feed in for a depth of about 1/8 inch, before removing the reamer and blowing out the chips. As you get down to the shoulder area, the feed in is less.

BARREL FITTING

CHECKING THE HEAD SPACE

When you get close, you will have to use the head space gauge. You will need to know what the correct head space will be. Take the action with the bolt cocked and closed, measure the distance from the inner ring, what the barrel is butted up against, and the bolt face. Take the difference, which in this case is .100.

Now knowing the depth is .100 we can keep feeding the reamer in until the head space gauge will be the right depth. The correct depth would be .095. Allowing for .003 for compression of the barrel to the action, this would leave .002 on the head space, which is about right. To set the depth, use a depth mike, placing the depth mike on the end of the head space gauge, and measuring the depth until it read .095.



FIFTH STEP, CHAMBERING BARREL

When you get close it would be best to use a dial indicator to go the final few thousandths. This is how the Mauser is chambered and head spaced. Other types of actions are done the same, with a little variation.

FITTING SPRINGFIELDS & ENFIELDS

On some barrels like Springfield's or Enfields, we use the same way of measuring from the front of the action to the bolt face. Because the barrel is coned, we have to measure from the cone to the head space gauge. This distance is subtracted from the distance from the shoulder on the barrel, to the face of the cone.

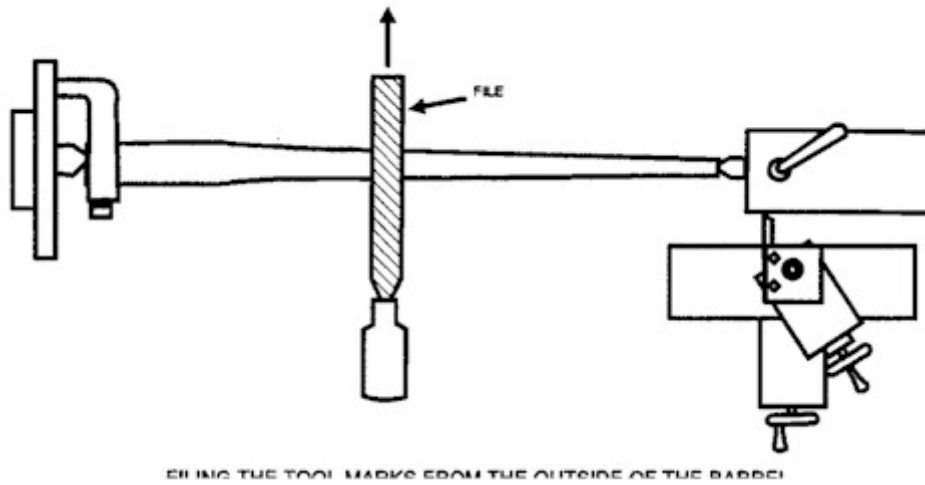
POLISHING THE CHAMBER

After the barrel is correctly head spaced, the chamber should be finely polished. This is done with the lathe running at a high RPM.

Note—The barrel that is sticking out the back of the lathe should be centered with a collar that

BARREL FITTING

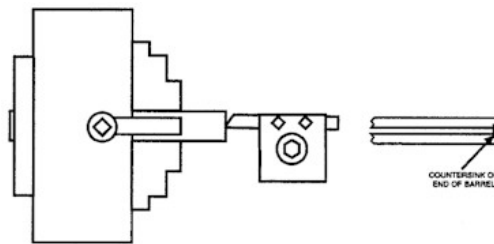
has set screws to hold the barrel from starting to run out of center with the high RPM. Start with a fine polishing cloth on the end of a slotted rod.



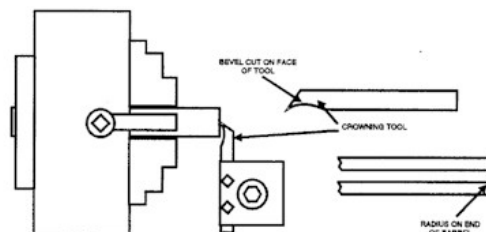
Do not over polish, as this will change the dimensions of the chamber. You will be able to see all the scratches, and will be able to measure any part of the chamber.

Screw the barrel on the action, seat it real tight, remove the extractor, and the cocking piece. Drop a new loaded round into the chamber and close the bolt. It should close any pressure. Remove the loaded round, and drop the No Go gauge in the chamber.

It should not allow the bolt to close. If for some reason it does, remeasure and face a little off the face of the barrel.



CROWNING THE END OF THE BARREL BY COUNTER BORING



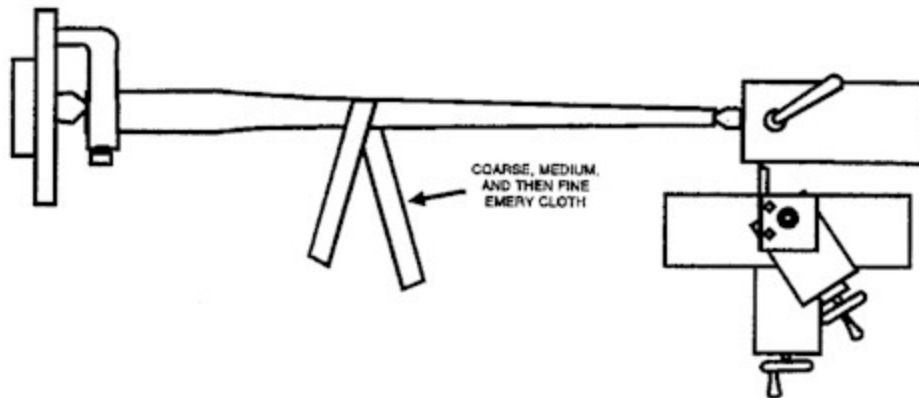
CROWNING THE END OF THE BARREL

BARREL FITTING

FINISHING THE OUTSIDE OF THE BARREL

All that we have to do now is to polish the machine marks from the barrel and crown the muzzle. I find that I can chuck the barrel in the lathe, using a live center and remove the machine marks with coarse grit paper, then finish polishing with finer grit. If the barrel is too rough it can be finished smooth and free of tool marks with a body or lead file. Then when the machining marks are removed, finish the barrel with coarse emery, following with finer grits.

This does not take long if you have not left on the outside of the barrel. If there are chatter marks on the outside, the barrel will need to be filed while it is spinning.



FINAL FINISHING THE OUTSIDE OF THE BARREL WITH EMERY CLOTH

When the barrel is as smooth as you want it, remove the barrel, turn it around, chuck it up on the muzzle and crown it. The easy way is to face the muzzle off flush, and counter bore it to protect the bore. You are ready to polish and blue.

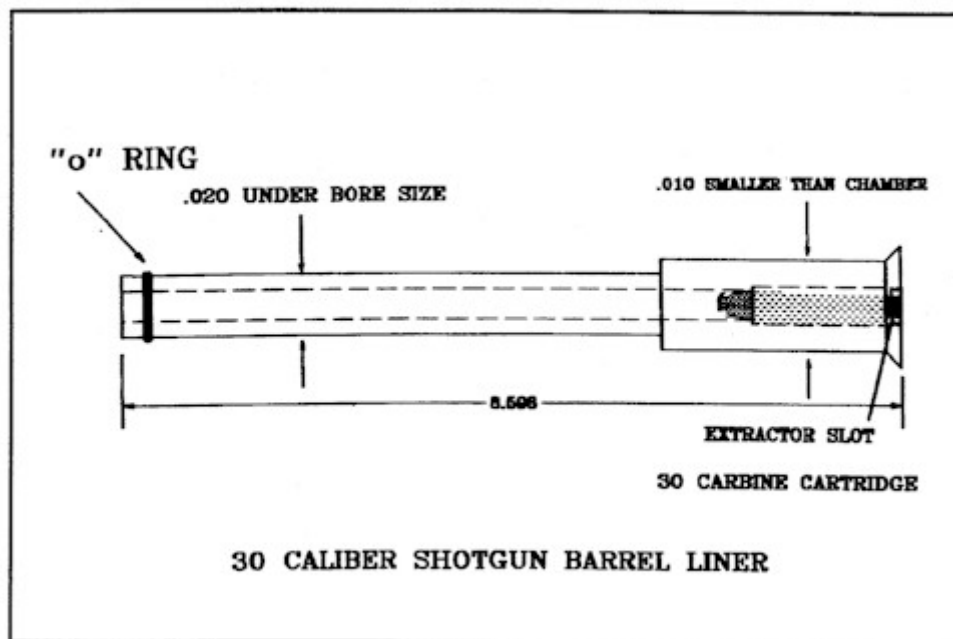
BARREL FITTING

BARREL INSERTS & LINERS

SHOTGUN INSERTS

You are out hunting with your shotgun and all you are using is No.6 shot. Up ahead you see some game but they are all out of range for your shotgun. You shrug your shoulders and walk on, wishing you had your high power rifle.

This has happened to me many times while hunting birds. Many a time I could have gotten me a Fox or Coyote if I had just had a rifle. I got an idea one day that I could have a rifle barrel in the double barrel, and much the same as a German Drilling.



When I got home, I found an old 30 Carbine barrel that would be ideal for a barrel insert for a shotgun. I cut the carbine barrel off right behind the gas port. I then turned the barrel down to where the barrel would just slip into the barrel on my 20 gauge double barrel.

I then turned the back end of the barrel down to the shape of a 20 gauge shot gun shell casing, but where it would just slip into the chamber of the shotgun. I carefully trimmed a little at a time off the front of the rim until I could just close the barrel up with a 30 carbine case chambered in the You can also make a chamber liner for a 38 or 357 cal. This is a chamber insert made like the long insert; with the exception it is only 6" long. Use a "O" on the front to center it.

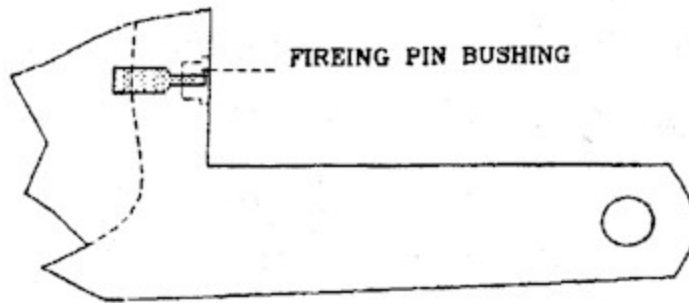
On a high pressure cartridge the firing pin hole in the breechblock needs to be bushed to keep the primer from flowing back and locking the action closed. Also any time that the primer flows back, there is always the chance that the primer will rupture. If that should happen, high-pressure gas may wreck the stock and action.

BARREL FITTING

There is always the chance of personal injury from the high-pressure gas. If you are using a cartridge with pressure over that of the 45-70 or 38 Special,

BUSH THE BREECH BLOCK.

The best cartridges for shotgun inserts are rimmed cases. The lower pressure cartridges will give good accuracy up to 100 yds. with plenty of power for most game.



BUSHING FIREING PIN HOLE ON SHOTGUN

The liner inserts give acceptable accuracy up to 100 yards, and have had many 5-inch groups from a 0/U shotgun. Some type of sights needs to be installed to get this type of accuracy. For close shooting good groups can be achieved with the front bead.

If you use the shotgun regularly, you can install an adjustable rear sight.

REVOLVER INSERTS

There is another type of insert that I use in my 357 revolvers. Each of the chambers has a .177 Cal. insert install in it. An old .177 pellet rifle will supply you with the necessary rifled barrel for this project. There is two ways of doing this project. One is to use only the inserts in the chamber, and the other is to also line the barrel.

Lining the barrel with a rifled liner gives the best accuracy, but the chamber method will give satisfactory accuracy up to 30 yds.

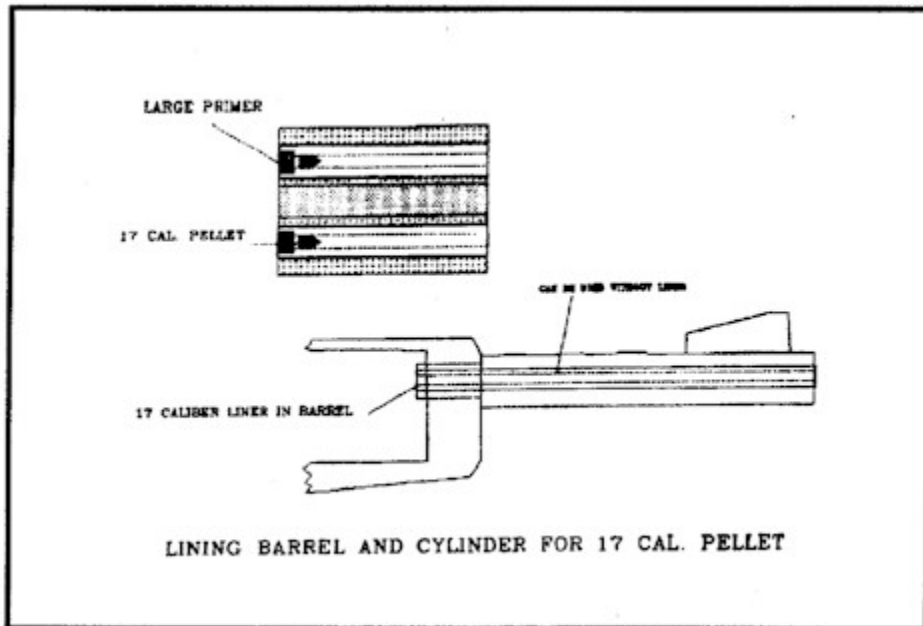
The short piece of barrel is centered in the lathe and is shaped just like a 38 or 357 case. It is made to just slip full length inside the chamber in the cylinder, with no play. The primer end is drilled to accept a large pistol primer. Drill just deep enough that it will be flush. Using a taper reamer, just ream the barrel in front of the primer so a .177 pellet will just start inside the barrel.

22 CAL. BARREL LINERS

When you have an old 22 of your Grand Dads that is in good shape, but the barrel is worn

BARREL FITTING

out, there is still hope. Brownell's and a few other suppliers have a 22 AF liner that will when installed give new barrel accuracy when properly set in the barrel.



The liner should be carefully chambered and then the chamber polished to remove the reamer marks. It should be chambered deep enough so when a cartridge is inserted in the chamber it will be flush with the end of the liner.

INSTALLING THE LINER

To install, all that is necessary is to measure the liner, get a twist drill the same size or a few thousandths larger than the liner, and grind a pilot on it (.218") and sharpen the cutting edge. Then a ¼ inch piece of drill rod is brazed to the end of the drill. You will need some type of "V" groove fixture to hold drill and rod in perfect alignment while brazing.

The old barrel can be put in the lathe, and rebored to the liner size. Drill from the muzzle to the chamber. You will have to clean the drill every ½ inch or so, and keep it well oiled with a good cutting oil.

When done, clean the oil from the barrel with solvent. You can the liner in the barrel already chambered. If you want now put the action back in the barrel, remove the extractor from the bolt. Insert a 22 case in the chamber of the liner. Now coat the inside of the barrel and the outside of the liner with epoxy. Slip the liner up to 1/8 inch of the bolt, or where the end of the chamber is.

Wipe off all the excess epoxy from the chamber end, close the bolt, and push the liner up flush against the bolt. Once it is flush on the bolt, open the bolt and wipe off any extra epoxy.

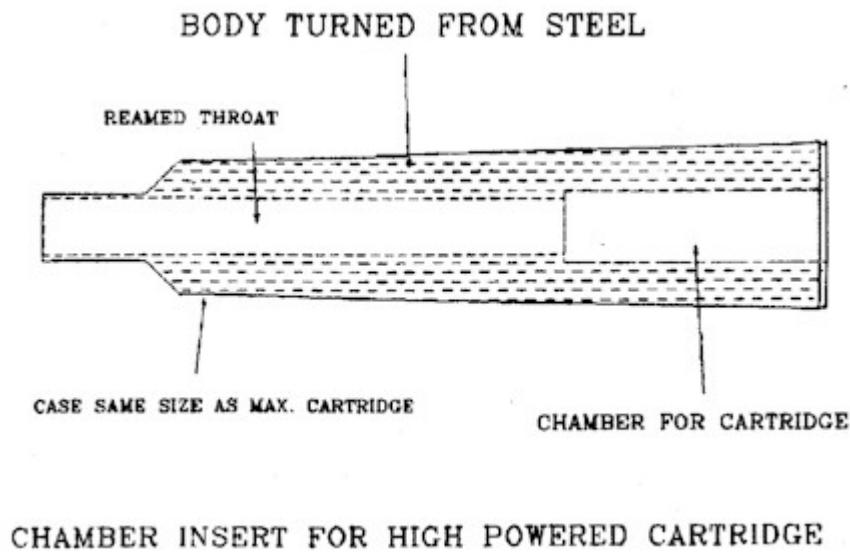
If you are careful the case can be removed also. All that is left to do is cut the extractor slot, and crown the barrel. This can be done by very carefully filling the liner flush with the muzzle,

BARREL FITTING

and then removing any burrs from the edge of the bore with a sharp knife.

RELINING BARRELS

Many old guns are around that have barrels that are rusted out, but otherwise the rest of the gun is in workable condition. There are others that have bores that are larger than the individual would like, and would like to change them to a smaller caliber to make them more serviceable.



There is a solution to correct the above problem. That is to rebores and reline the barrel to the caliber that you want. It is a simple matter to do this, as all that is necessary is to purchase tubing that has the correct ID, or slightly smaller. The bore is then reamed to the correct size, and then rifled. In relining barrels, the original barrel is bored out by a drill with a pilot to fit the original bore. The drill being started at the chamber end of the old barrel so that it will be well centered. The barrel is then reamed with a barrel reamer to bring it to the size of the tube to be inserted in it.

Once this is done the old barrel is then drilled out to the size of the OO of the liner tube. If you cannot find the tubing the size that is required, a rifled barrel or blank is made or purchased and turned down to the OO that is required.

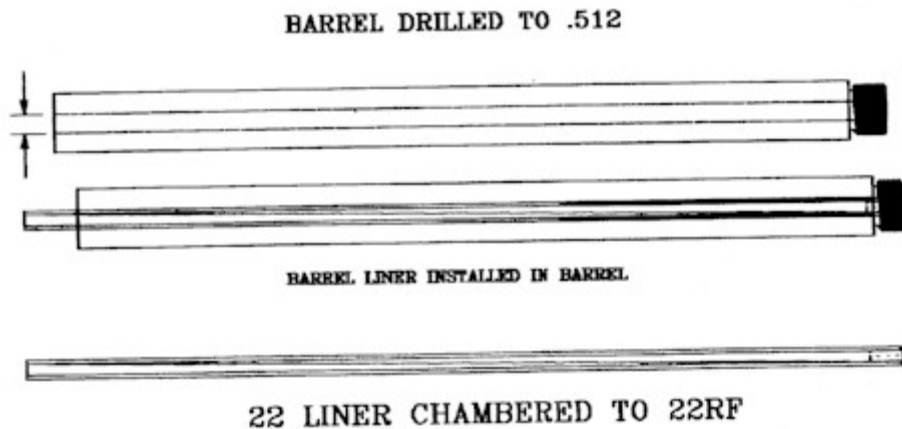
Be forewarned that to turn a barrel down to a smaller size does have certain problems such as warping; it is still a simple operation. Sharp tools, and a follower steady rest will eliminate much of the chances of warpage.

TUBING FOR RELINING

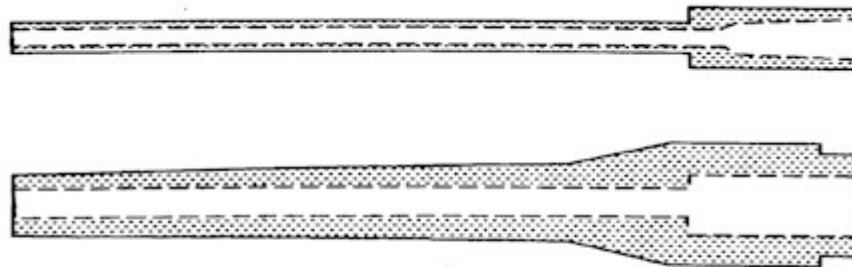
Most of the barrel liners for relining are long, and if possible it is best to get seamless tubing for the operation. The tubing can be purchased from any steel supply house, and of many different makes of manufactures. I have found is manufactured by Superior Tubing, and it has excellent interior surface finish. The surface finish is very good, and is usually better than the

BARREL FITTING

other tubing manufactures. This is very important as if the ID of the tube is rough and looking like an 1800-century washboard. It will be difficult to ream it and get the interior finish that is needed for the bore.



When you buy the tubing you may have to buy the tubing with a smaller ID than is desired, and bore it out to the size that you want, and then ream it to the correct size. It is best to use a short piece of tubing, and bore and ream it to be sure it will do a good job and get the desired interior finish. Most of the tubing will be 41 30 that is a Chrome-Moly steel. This steel will give good wear, but it is more difficult to get a good finish. If you are able to get the desired results, you can now bore out the old barrel to fit the OD of the tubing.



LINEING HIGH POWER BARREL

BORING TOOLS

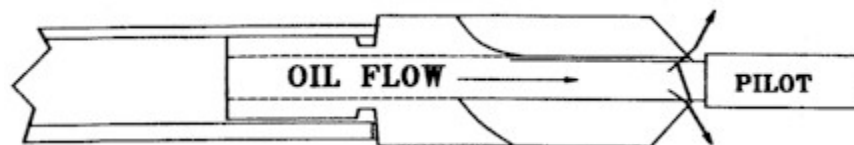
The boring tool that I mostly used was the standard drill bit with a pilot ground on it that was a close fit to the bore. This was done on the lathe with a Tool Post Grinder. Chuck the back end of the drill in a collet or three jaws chuck, and use a hardened female center in the Tail Stock. Grind the pilot at least $\frac{1}{2}$ " long, but $\frac{3}{4}$ " is better. A recess is cut at the shoulder of the pilot to make sharpening the drill easier. The drill is then removed from the lathe and the shoulder is resharpener to the correct angle of the original drill. It is wise to select the drill size to about .0025 thousands smaller than the finish size of the bore. The reamer will remove the extra stock, and finish the bore to a close fit for the liner.

BARREL FITTING

A shaft close to the size of the drill is now brazed to the end of the drill. The shaft is slightly beveled, and the drill and shaft is laid in a piece of angle iron. The shaft has to be raised slightly to be centered with the drill. After all is lined up the two are brazed together, and let cooled slowly to relieve any stress. Grind off any extra brass and drill is ready to use.

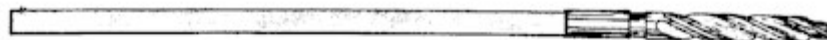
BORING THE BARREL

Remove the barrel from the action, and any other attachments such as sight, etc. that may be on the barrel. The barrel is then centered in the lathe using the lath chuck if the spindle is large enough for the barrel will fit inside the Headstock Spindle, If the spindle hole is too small, the barrel will have supported with a steady rest on the chamber end, and the chuck on the muzzle end.

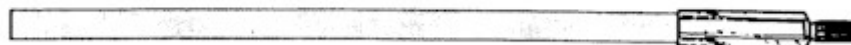


DRILL FOR BORING BARREL

Insert the drill in the tailstock chuck, or in a special holder on the carriage. The carriage is the best way to re bore the barrel, as the drill has to be with drawn about every half inch to clean the drill.



DRILL BIT BARREL DRILL



BARREL BORING DRILL

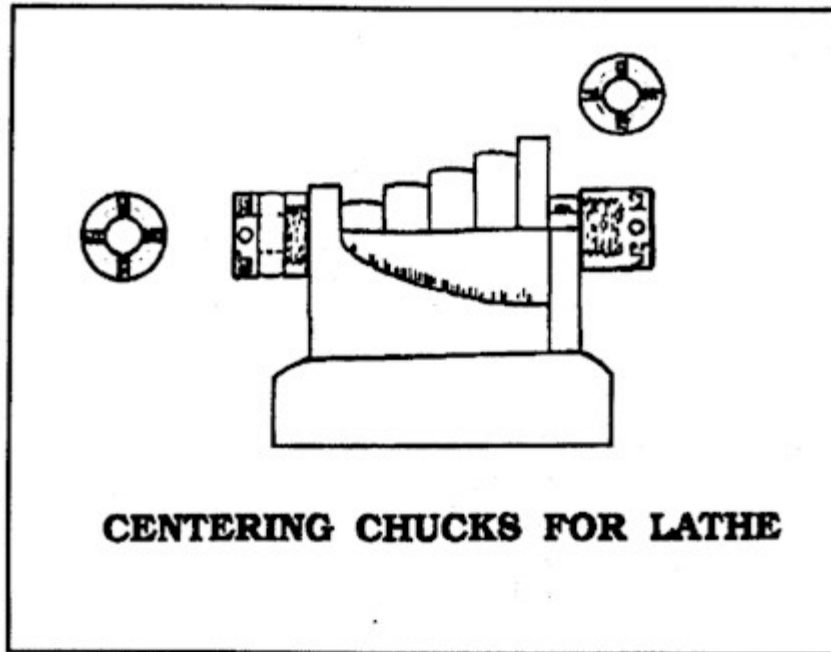
Start the lathe up and lubricate the bore with a good grade of cutting oil, and adjust the speed for the drill size that you are using. Start the pilot in the chamber end, and engage the feed on the carriage. Have the feed set for a slow feed rate, or the drill can be fed in by hand if so desired. Feed the drill in the barrel for about a half-inch, remove from barrel and clean the chips from the drill.

RELINING THE BARREL

The barrel to be relined is placed on a soldering stand, while gas burners heat it along its full length, from each side. When it reaches the proper heat it is tinned all over, inside, with a stiff

BARREL FITTING

wire brush. If the brush is used, it is dipped into and saturated with regular tinnings solder and the barrel is tinned by running the tool or brush through the barrel.



The liner has its ends plugged with wood plugs and is tinned on the outside. It is inserted in the barrel and worked back and forth so excess solder is worked out. The surfaces of both barrel and liner are evenly tinned. The liner is left about three inches longer than the barrel, an even amount being allowed to extend from each end of the barrel.

Stand the barrel upright while cooling. After it is cold, hang it up with a wire and strike it with a small hammer. If you have a good soldering job, it will ring. If it does not ring, the liner will have to be removed and the soldering operation repeated until it does ring.

After the liner is cut off flush with each end of the barrel and fitted to the action, the extractor cut is made, the liner is then chambered for the cartridge and the muzzle end is finished with a countersink equipped with a pilot fitting the bore.

FINISHING THE BARREL

After rifling the liner tube, the rifling cutter sets up a small burr along the edges of the grooves and these must be removed by lapping. Insert a grooved lapping rod in the barrel after wrapping about two of the lower grooves with string, the rod being inserted from the breech end of the barrel and stopped just a little short of the muzzle.

RIFLING THE LINER

In casting a lap in a relined barrel in which an unrifled tube has been inserted and later rifled, the barrel cannot be heated as hot as the one piece barrel, or the solder holding the tube in place will be melted, therefore keep the temperatures to not much over 250 degrees.

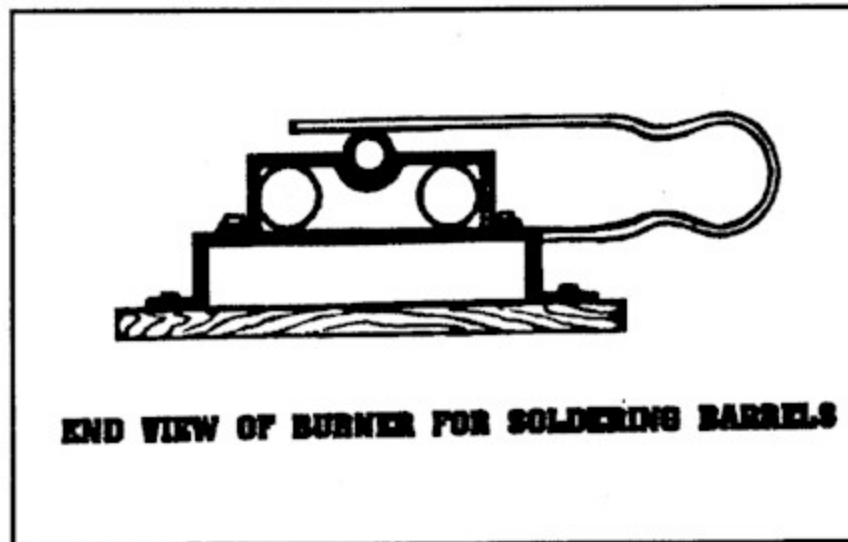
BARREL FITTING

The barrel is held upright in a vise, muzzle-up, and in the case of a new barrel, is heated so that the lead casts a full lap. If it is heated to just a light straw color, it will be sufficient and the heating is done only for the length of the lap at the muzzle, about two inches. The melted lead is poured into the muzzle and is just allowed to come flush with it. As soon as the lead cools, the lap is pushed out of the muzzle just a short distance and examined to see that it is perfect. Any overflow at the muzzle is trimmed off with a sharp knife. Be careful not to allow the lap to come entirely out of the barrel at any time until the lapping work is finished.

LAPPING THE BARREL

Give the lap a coating of light oil and withdraw it to the breech end of the barrel, allowing it to extend slightly from this end, so that the string may be removed and the lap coated with a fine grade of optical emery. While the lap is at the rear end of the barrel, coat the barrel from the muzzle end with oil.

The lapping rod is equipped with a ball bearing cross-handle at the opposite end from the lap so that it may easily be grasped and worked back and forth through the barrel. Stops are placed at both the muzzle and breech end of the barrel so the lap will never be pushed or pulled entirely from the barrel. Lap the barrel for eight or ten minutes, adding fresh emery and oil frequently.

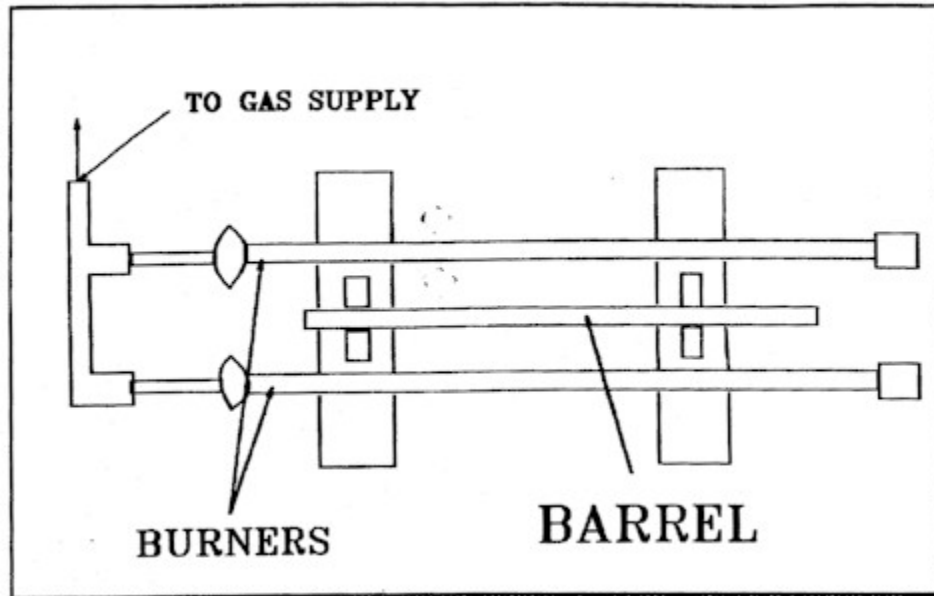


At the end of this time withdraw the lap from the barrel and wash the barrel out with solvent and examine it to see that all burrs have been removed. If further lapping is necessary, an entirely new lap must be cast on the rod in the barrel, as before, for the old lap cannot be reinserted.

If you desire an extra high polish, as is often given to target barrels, the barrel can be polished with rouge and light oil after the lapping operation. Leather washers about two or three thousandths of an inch larger than the groove diameter of the barrel is placed upon a polishing-rod with ball bearing cross handle. The leather washers being separated from each other with small brass washers and retained in place by a brass nut on the end, which is made the bore diameter. During the polishing operation, this nut can be tightened a little from

BARREL FITTING

time to time, to swell the leather washers slightly. The polishing operation may be kept up for an hour or more.



MAKING 22 RF LINERS

I will show you how to make 22 RF liners, for relining shot out 22's. I first came up with the process in 1958, and at that time no one made any liners, except an outfit in England. I sold over 5000 liners in the first 6 months.

Then I realized there was a need for a tow price liner to take care of the thousands of 22 barrels that were shot or rusted out.

I set about trying to come up with a suitable method to mass-produce a liner that would be fast to make. Give the desirable interior finish without any mechanical reaming, and a material and method that would give a superior rifled finish.

LINER TUBING

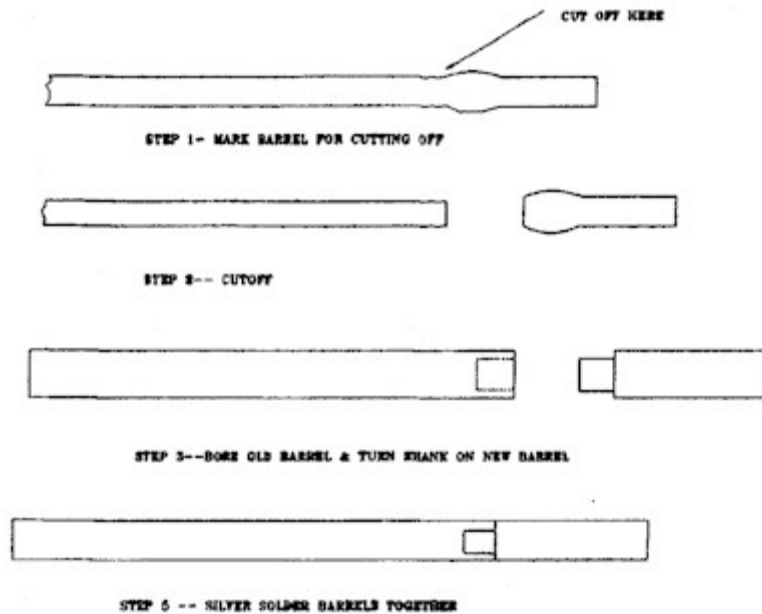
I spent quite a bit of time experimenting with various methods of manufacturing, tried many different types of steels. Right off the start I ran into one big problem with the liners. The interior finish was very bad. It seems the manufacture was unable to hold good I.D. finish or size on the tube. Most often the tubing had chatter marks inside the tube, as well as uneven bore. Some of the tubing ran as much as .015 oversize in the bore (In places). They were supposed to be .187 on the 1.0., but I found that some ran so large that a .219 button would fall through the bore.

I just about gave up on the idea. I even tried to ream the bore to remove the roughness, and did get a very smooth and shiny bore. When I rifled it all the chatter marks reappeared! The inside looked like a washboard. I had been in contact with various tubing makers across the country, and had gotten several samples of tubing to try. One was from Superior Tubing.

BARREL FITTING

When I wiped the grease from the bore it was like a mirror, boy these people had their act together. I cut off a small piece, lubricated it, and rifled it. It was a perfect barrel. Size was perfect, finish was perfect, and was what I was looking for.

If you get tubing, you must specify that you want tubing from Superior Tubing. I know of no other manufacture has their quality. When you order you need 5/16 x .187, plus or minus .003. You should also specify that the tubing should be free from interior chatter.



REPAIRING SHOTGUN BARRELS

The process I will describe here will give you a very high-grade liner as long as you use good tubing. Also this process works fine for making larger calibers. I have turned out several muzzle-loading barrels using this process. The only difference in making muzzle-loading barrels is that you need a longer stroke on the rifling machine. You can buy just about any size tubing that you need. With button reaming you will be able to size most slightly undersize tubing to the correct size the first pass.

22 RF BUTTONS

The first thing that you will need is the sizing button and rifling button. Sizes are given in the chapter on rifling buttons. When you get the tubing you will need to soak them in solvent to get the grease out of the bore. After the tubing is cleaned it will need to be counter bored, and special holders made. After counter boring, wipe the barrel with a light grade of oil for button reaming. The counter bore is $\frac{1}{4}$ inch. You need a $\frac{1}{4}$ inch piece of drill rod that is hardened and about 2 inches long. A sleeve is needed to hold and clamp and clamp to the tubing. Set the liner in the rifling machine, slide the sleeve over the tubing, insert the reaming button, then the drill rod and tighten the set screws, and you are ready to button ream. After button reaming the bore, swab with MolyKote Z and rifle the liner.

22 SIZING BUTTON

BARREL FITTING

The correct size for the swaging button is .216. This is undersize for a 22 barrel but is necessary for liners. When they are rifled the size will be OK. When you are rifling, be sure the liner rod is rotating so there is no binding.

LINER WARPAGE

During the swaging process the liners sometimes warp so these that warp will have to be straighten. The rifling button creates much stress in the swaging process. When you make the rifling button it should be

.223. I find that 6 or 8 groove button works fine on getting good accuracy. You can use these liners for 22 RE or 22 RE mag. The counter bore should be cut off the liner, and it is ready to be chambered and installed. To install, all that is necessary is to measure the liner, get a twist drill the same size or a few thousands larger than the liner, and grind a pilot on it (.218") and resharpened the cutting edge. Then a ¼ inch piece of drill rod is brazed to the end of the drill. The old barrel can be put in the lathe, and rebored to the liner size. Drill from the muzzle to the chamber. You will have to clean the drill every ½ inch or so, and keep it well oiled.

INSTALLING THE LINER

When done, clean the oil from the barrel with solvent. You can the liner in the barrel already chambered. If you want now put the action back in the barrel, remove the extractor from the bolt. Insert a 22 case in the chamber of the liner. Now coat the inside of the barrel and the outside of the liner with epoxy. Slip the liner up to 1/8 inch of the bolt, or where the end of the chamber is. Wipe off all the excess epoxy from the chamber end, close the bolt, and push the liner up flush against the bolt. Once it is flush on the bolt, open the bolt and wipe off any extra epoxy. If you are careful the case can be removed also. All that is left to do is cut the extractor slot, etc.

BARREL STRAIGHTENING

If your drill was properly sharpened, had a good starter bushing, and the starter bushing was in good alignment, the bore should be straight. In most cases the barrel will shoot as good if it is slightly crooked, as it would if it was straight, but if it was very slightly crooked and turned and finish that way, it would tend to walk the bullets as the barrel heated up.

In turning a barrel if you do not get the cutting tools sharp, take too heavy a cut, let the barrel get too hot, let the centers get too hot, or out of round, the stress will warp the barrel. Then you will have to straighten the barrel.

I will try to cover the main points in barrel straightening, and give you a guideline as what to do and what to look for. You must realize that barrel straightening must be self-taught, and to accomplish this, it will be advisable to practice on old barrels.

A shotgun barrel is good to practice on, as it is big, and smooth with no rifling to confuse you. It is next to impossible to straighten a barrel with a rough bore. I will also show you here how

BARREL FITTING

to make barrel-straightening equipment necessary to straighten the barrels,

Let me point out that a large percentage of the factory barrels, to the trained eye are not perfectly straight. Any barrel maker will tell you that a slightly crooked barrel will shoot as well as a straight one, and a rifle used for hunting a crooked barrel makes little difference.

As long as, (1) you let the barrel cool between shots when sighting in, (2) as long as you do not shoot more than 3 consecutive shots, as it will then start to walk.

No amount of bedding will stop this. Now this book was originally written in the late 50's before the coming of computer machines for turning barrels. Before, the barrels when turned were checked at every pass on the turning lathe, but now the complete blank is turned complete on tape machines before being removed from the machine. Then it is checked for straightness, and if crooked will be straightened.

I have gotten several factory barrels in the last few years that when put between centers of the lathe, ran out as much as 1/8 inch on the 00, even though the bore was straight. These barrels would walk as much as 6 inches at 100 yards after 5 shots.

BARREL WALKING

Why does a barrel walk? If the bore is concentric to the outside the thicker side will lengthen more than the thin side and the barrel will bend. In most cases there was enough metal on the barrel to return the barrel to make it even.

PREMIUM BARRELS

Let me point out that premium barrels are barrels that are straight after drilling. They have no loose or tight spots in the bore due to hard or soft spots in the barrel, and they do get a higher price. Even with good steel, some times you find some bars with much stress left in the bar. This steel will warp with every pass of the cutting tool. There is not much you can do to stop it, except straighten it on every pass.

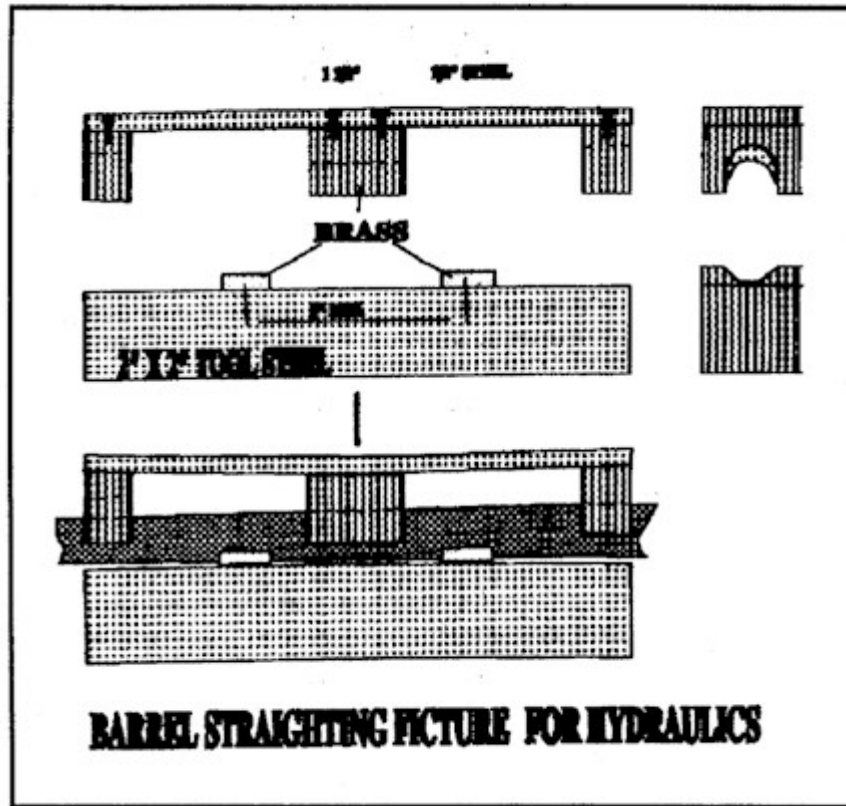
These if you keep them straight up to the final pass will shoot good and will never give any problems later. They are just a pain in you know where. On barrel straightening equipment, the one type that I used is the overhead screw press. You can see the bore as you are making corrections on it.

BARREL PRESSES

This type of press can be made quite easily in the shop with nothing more than a **CUTTING TORCH, ARC WELDER, AND OLD FARMING EQUIPMENT. WHY THE FARMING EQUIPMENT? ALMOST EVERYTHING NEEDED CAN BE FOUND IN USED FARMING MACHINERY. THE PARTS CAN BE CUT OUT AND WELDED IN PLACE. THE WHEEL IS NOTHING MORE THAN A BIG FLY WHEEL WITH HANDLES FASTENED TO IT.**

BARREL FITTING

Another is the type that you use an overhead hydraulic press to make correct it. This is a good type but I have found that it is better if you can feel the pressure when you straighten the barrel. Both will do a good job, but the overhead screw press is faster. If you plan to make quite a few barrels, certainly go to the overhead press.



The oldest method employed by the old time barrel makers was to stretch a fine wire inside the barrel. This is stretch from one end to the other, and touching the sides at each end. One side of the barrel was hammered until the wire touch all the way.

This was used on barrels that was soft, and used lead bullets. The methods that I will describe in this chapter will be the one I have used for years, and have found it easy for others to use. Lets put a barrel to test to see if it is straight.

When you look through a bright finish barrel, the interior surface appears to be spread out in a circular disc as far from the eye to the other end of the barrel. As you look through the center of the disk, is a circular (the bore) orifice, and surrounding it, like the rings of a target, at equal distances, (if ft is straight).

These circles are well-defined circles around the bore. When you do this look at the edge of a door or window, not an open light. If the second, third, or fourth rings are a perfect circle around the bore, the barrel is straight.

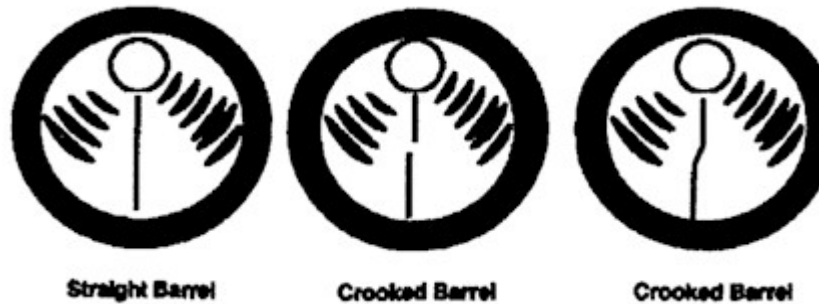
It will be seen that these images are located at a certain point in the bore nearest to the eye. In two thirds of the length of the barrel this is the part of the bore that you must direct your attention to. This is where by using the reflection, which will show you where the bore is

BARREL FITTING

crooked.

LONG BENDS

In long bends you will work from $\frac{1}{2}$ to $\frac{2}{3}$ of the bore, then turn around the barrel and repeat the process. If a distortion of the circle or rings is noticed, revolve the barrel slowly, and you will see what side the bend is on. The hardest part is next.



It will take a little practice and time to tell exactly how far, and at what point the bend is. This is done by looking down the barrel, and at the same time touching on the outside of the barrel. When you think you have the correct spot where the center of the bend is, move the barrel so that the center jaw or hook is in that spot, rotate the barrel so the big part of the rings is at the bottom, and apply a little pressure.

If you are at the correct place in the barrel you will see as you apply the pressure, the barrel goes straight. The rings will form a perfect circle around the bore. If this does not happen, release the pressure. Rotate or move the barrel forward or back and repeat the process.

When you get the perfect circle put more pressure until when you release the pressure the barrel will remain straight. The barrel is like a spring and it will take quite a bit of pressure. When done the interior will look like the above drawings. When you mount the screw press, have it face the window, so you will see half of the edge of the window in the bore.

With a little practice you will be able to straighten the barrel in a few minutes. The second method is with the blocks. This method works fine but is more difficult to get the barrel straight and is time consuming. This method is best used for full size barrel blanks, or if you do not have an overhead press.

A large lead hammer can be used on liners to bend the liner. Find where the bend is, and set the liner on the blocks with the bend to the top. Now give the barrel a good tap with the lead hammer. Check to see if you have made any progress, if not repeat the above with a harder tap.

The secret of this process is to hit the barrel hard enough to straighten it past its elastic limits.

Your skill in using this process lies in your ability to judge by the eye, the exact location where

BARREL FITTING

the bend, its proper location, and then slide the barrel to that location.

To straighten the barrel it must be bent past its elastic limits, so when it springs back it will be straight. If you bend it too far it will be bent the other way, then you will have to rotate the barrel and straighten it back.

With a little practice you will be able to straighten the barrel in a few minutes. The second method is with the blocks. This method works fine but is more difficult to get the barrel straight and is time consuming. This method is best used for full size barrel blanks, due to their large size.

BARREL FITTING

BEGINNING TOOL MAKING

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 make gun barrels, but I am not considering speed, number of operations, or number of barrels produced.

LATHE

Your lathe should have at least a 2-foot bed. If you are planning to make muzzle-loading barrels, a 4 ft. bed is preferable. The hole through the head stock should be at least ½ inch.

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

TOOL POST GRINDER

If you are going to make your tools, such as reamers, rifling buttons, 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 percentage of what it would be if you had to buy them or have them special made.

You will be able to grind your own reamers, deep hole drills, make your rifling buttons, and make your own chambering reamers. In general be able to make any caliber of barrel with any desired chambering.

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.

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 barrels, and ream barrels.

BARREL FITTING

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 that is done on a shaper can be done on a milling machine. However some special shapes can best be 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 1- $\frac{1}{4}$ bar steel with a hacksaw. 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. 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.

I have found that oil hardening tool steel (O1) 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 thousandth of an inch.

BARREL FITTING

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.

GAUGES

Some of the gauges that will be needed are a bore gauge for measuring the finished reamed bore of the rifle barrel. There should be a gauge for each caliber that you make.

Each gauge should have a go and no go gauge on it. They can be turned out on a lathe. The no go gauge should be .015 larger than the go gauge.

ANGLE AND RADIUS GAUGE

Another gauge that you will need is an 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 30 years I have found 01 hard to beat. More on this later.

HEAT TREATING TOOL STEEL

We will now cover as much as possible 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.

That part of this chapter that deals with the heating cycles for typical tool steels, the

BARREL FITTING

quenching or cooling procedure, and the effects of single tempering operations on the structure and properties of the steel.

In most cases, 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, plain carbon steel cannot be hardened adequately, and it is necessary to add allowing elements 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 micro structure 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.

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 gives 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.

BARREL FITTING

TYPICAL TOOL STEELS

The composition of tool steels that we use for tool making is as follows.

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.

MOLYBDENUM HIGH SPEED STEEL, Carbon 0.80, Manganese 0.24, Silicon 0.29, Chromium 4.15, Vanadium 1.89, Tungsten 6.64, Molybdenum 4.94 per cent.

CHROMIUM MOLYBDENUM HOT WORK STEEL Carbon 0.38, Manganese 0.40, Molybdenum 1.35, Nickel 0.30 per cent, Steel of the first analysis listed is usually referred to as plain carbon steel, and is also frequently a water hardening or shallow hardening tool steel.

The next analysis is characteristic of 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, relatively 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.

STEELS OF THE THIRD ANALYSIS

Is referred to as a high carbon, high chromium steel or as an air hardening tool steel.

The last two steels are not used in the small shops, as they require special equipment to process them.

HOW HEATING CHANGES THE STRUCTURE OF TOOL STEEL

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 desired 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

BARREL FITTING

% carbon, the hardness remains practically constant.

For maximum hardness in the steel, therefore, approximately 0.70 % of carbon must be dissolved in the austenite. All the steels being discussed except the chromium molybdenum hot work steels 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.

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

RYERSON TOOL STEEL

Over the years I have used Ryerson Tool Steel in 90% of my tools and reamers with great satisfaction with long life of tools, and overall good results.

BARREL FITTING

WATER HARDENING

AISI-SAE W2-1 .00 Carbon-Vanadium. This is 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.

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

Hardening - AISI SAE 02,

Rockwell C

As Quench	100 deg	200 deg	300 deg	400 deg	500 deg	600 deg
67	67 67	65	62	59	55	

TEMPERING TEMPERATURE

This is one of the tool steels that I used, along with 01. 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 rolls, thread rolling dies, thread chasers, blanking dies and punches, bushings, etc.

Working Temperature Forging, 1800 to 1925 degrees F., never below 1550 degrees F.
Hardening, 1400 to 1475 degrees F. Annealing, 1375 to 1425 degrees F.

Tempering Temperature

As Quenched	300 deg	400 deg	500 deg	600 deg	700 deg
64	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.

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 usually 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

BARREL FITTING

decarbonizing of the surface Brass-brazing flux can be used also, and it works up to 2200 degrees.

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 O1 or O2 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.

QUENCHING MEDIA

I have used quite a variety of liquids to quench the tools in over the years, here are some:

WATER

Water as it comes from the faucet, but I 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 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 re-heating 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, in any temperature range, to put on

BARREL FITTING

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.

BARREL FITTING

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.

BARREL FITTING

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 softening. I then put the part in the oven, then let it cool down over night.

ANNEALING

In some cases 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.

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 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

BARREL FITTING

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.

BARREL FITTING

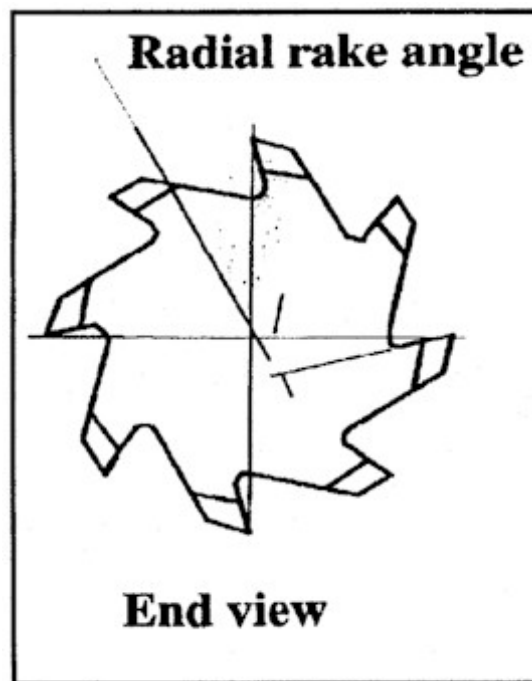
MAKING REAMERS

INFORMATION ON 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.

SHAPE OF FLUTES

Style and shape of the flute determine its ability to carry away chips and 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 reamer is 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.

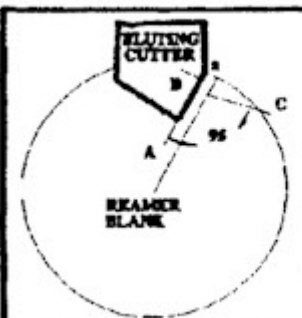
The relief of the cutting edges should be comparatively slight. They can be ground close on the tool post grinder and stoned flat with an Arkansas stone. The flat relief is what I have used

BARREL FITTING

all along, because the reamer has a keener cutting edge.

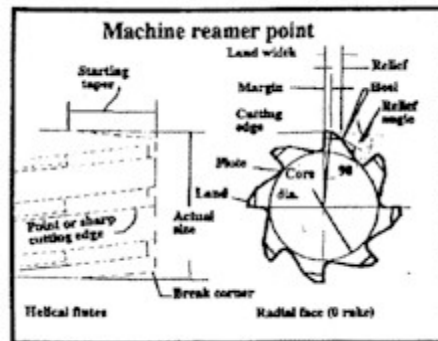
AMOUNT TO SET CUTTER AHEAD OF RADICAL LINE TO OBTAIN NEGATIVE FRONT RAKE

SIZE OF REAMER	DIMENSION IN INCHES	SIZE OF REAMER	DIMENSION IN INCHES	SIZE OF REAMER	DIMENSION IN INCHES
1/4	0.011	7/8	0.018	2	0.027
3/8	0.016	1	0.024	2 1/4	0.036
1/2	0.022	1 1/4	0.032	2 1/2	0.045
5/8	0.027	1 1/2	0.040	3	0.054
3/4	0.033	1 3/4	0.048		



CHATTER

This can be the one thing in barrel making 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. 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.

In most of my reaming I find that a good starting point is about two-thirds the drilling speed for a given material. If you ream too slow, it takes too long to ream a barrel, and very few barrels can be turned out. If too high feed or speed, premature dulling, chatter, and usually 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 oversized, and rough. I also have found that a good starting point is somewhere between .0015 to .004 feed per flute per revolution. The smaller bores such as .175-.224 a

BARREL FITTING

.001 -.003 per revolution is a good start. I can only recommend that you find the highest feed that will produce the required finish and accuracy.

STOCK ALLOWANCES

This is covered in the chapter on barrel reaming and reamers. This is very an important item in barrel making, 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 from the barrel, 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.

LUBRICANTS

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

THE MOST COMMON ARE;

CHATTERING

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 PLUGGING

The surface speed for reaming should be rather slow, on a regular barrel-reaming machine; the reamer turns, but on the lathe the barrel turns and the reamer is stationary. Instead of using an open belt and higher speed, the lathe is set in back gear, and run at the slowest speed.

FEED RATE

The feed can be made faster than that used for drilling. The barrel reamer is pulled through and used for a greater depth, with more cutting edges in contact with the barrel.

For the roughing reamer you can use a feed twice as fast as that for finishing. After the drills

BARREL FITTING

and reamer are made, and kept in good shape, many barrels can be turned out with them.

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 ream the barrel. 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.

I will go through with you on making a reamer. You will need a piece 5/16-inch (.312") drill rod, 6 inch long.

Go to the lathe and chuck up one of the pieces and using a ¼ inch center drill, center the ends just deep enough to get a good center.

Center up the drill rod between centers on the lathe, with a small lathe dog holding the other end. The .312 drill rod is just about the right size for the .308-fluted end, so it will not need to have anything done to it.

The other end will have to be turned. Turn the shank (the end where the square shank or holding is) to about .010 larger than the finish size, and the recessed area between the shank and where the reamer flutes start to about .280.

The shank area will be on most of the reamers about 1-¼ inches long, the recess will be about ¼ inch long.

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

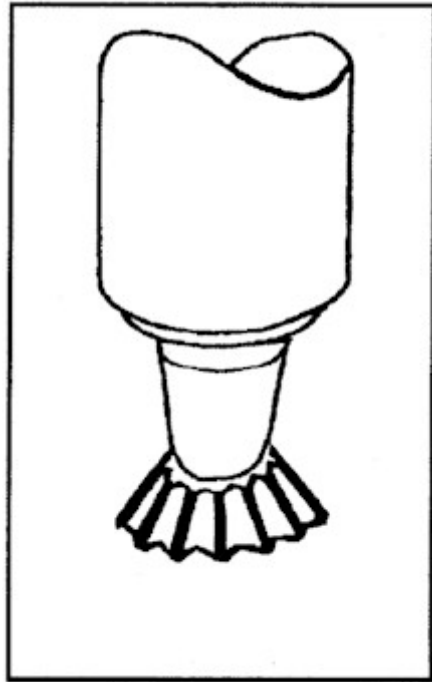
Next chuck up a 60-degree angular cutter in the mill. The diameter of the cutter should be at least 1 ½ inch. Slowdown the mill to about 100 to 150 rpm as the tool steel tends to get fairly hot.

Measure the drill rod on the flute end; in this case it has not been turned so it is .312. 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 .156, 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.

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

BARREL FITTING



THE FLUTE THICKNESS

The flute's wall should have a thickness of about .060 to .080. This would of coarse 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.

You want to use an ample supply of coolant when cutting the flutes. Depending on the reamer size you will go in about $\frac{3}{4}$ 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 heavy a cut, or the cutter is dull, or you may be cutting to fast.

DEPTH OF FLUTES

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, it will probably warp badly. 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.

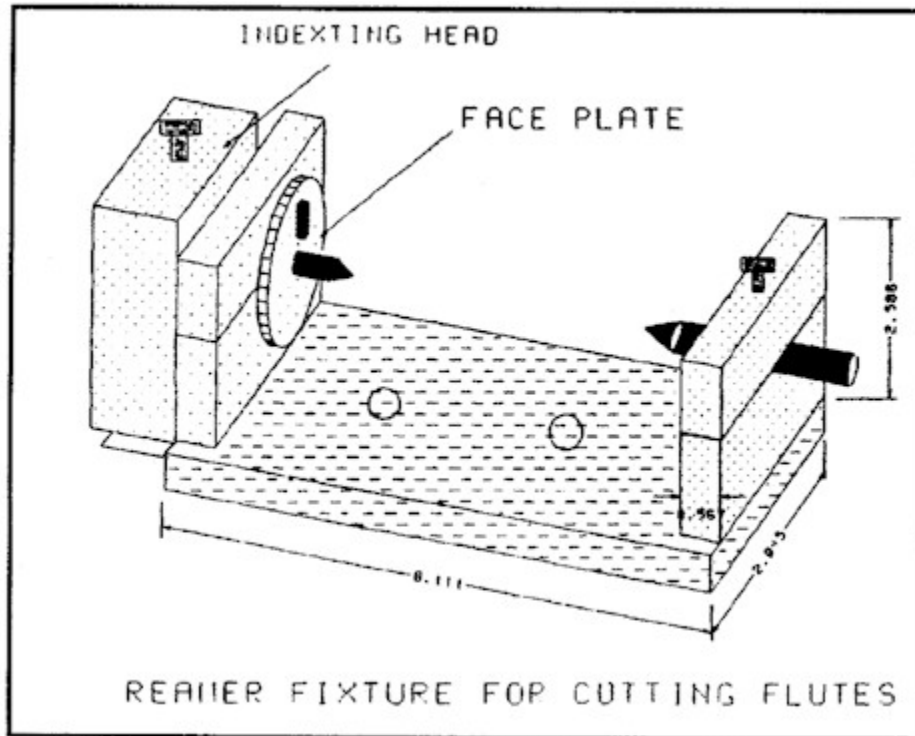
HEAT TREATING THE REAMER

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.

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 lank, or

BARREL FITTING

what other quenching medium the manufacture recommends.



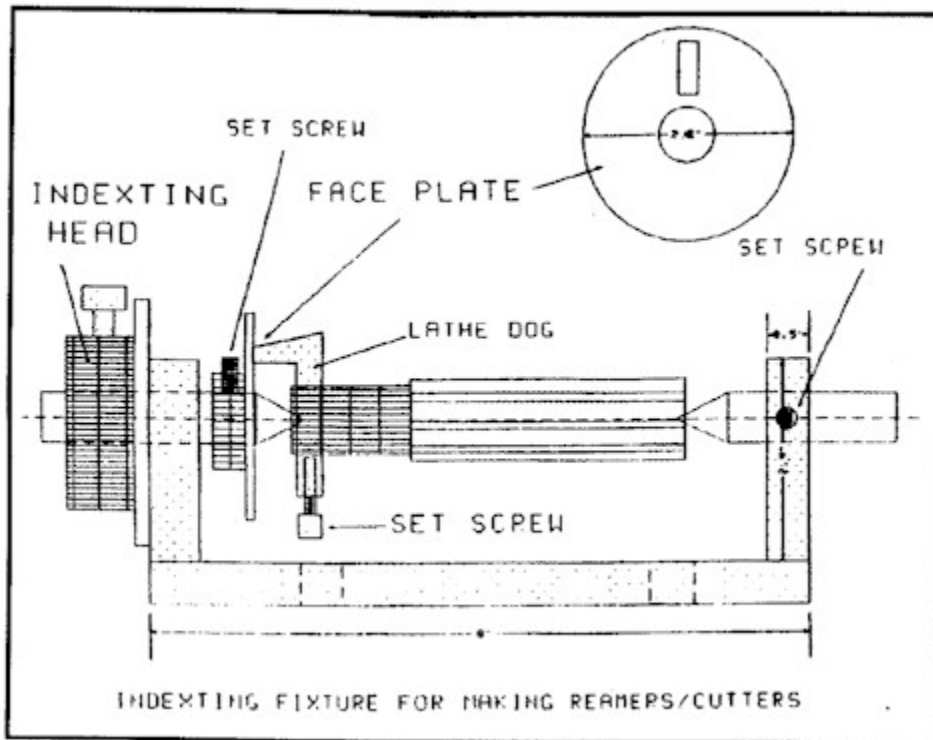
When you quench the blank make sure that you go straight in the quenching tank, if you quench the reamer at an angle, you will warp it. If you do warp a reamer you will have to bring the reamer up to 1500 degrees and let it cool in the oven, then straighten it when cool.

When you have both reamers 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.

BARREL FITTING

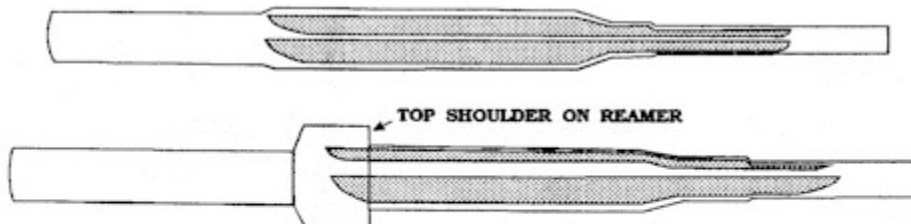
TEMPERING THE REAMERS

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 over night. The reamers will be about 61 to 62 Rc in Hardness.



GRINDING THE REAMER

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.



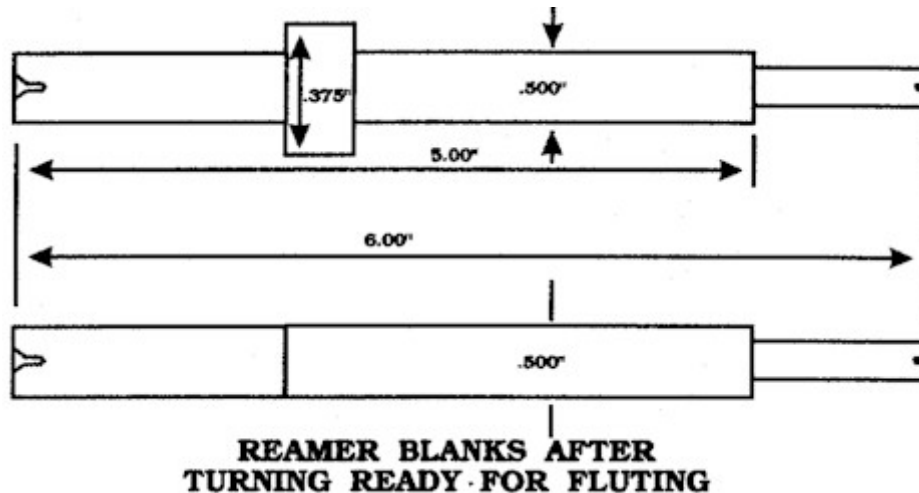
DIFFERENT TYPES OF REAMERS

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. Set the lathe in back gear drives; turn on the lathe

BARREL FITTING

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.

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. There should be about .002 taper from the shank end to the end of the blank.



The tail stock end should be the smaller end. This is very important to have or you will get a rough bore. We are grinding the rough reamer first, so you want to end up with the shank end of the reamer .290. When you get this size remove the reamer blank from the lathe. We will allow the .0005 to hone in to size.

Now that the second reamer blank is ground we will grind the leading edge taper. Set the compound on the lathe to 3 degrees. This will be your cutting edge on the end of the reamer. Turn the lathe on in reverse, and the grinder and start feeding the wheel by hand on the cutting edge of the reamer. It will take several passes to do this. Take light passes not much over .005 so you will get a good finish.

When you have completed this on the reamer, 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.

GRINDING THE SHANK

Square the grinding wheel up; it is time to grind the shanks to the correct size. The roughing reamer, which is .300 diameter, the shank, will be ground to .290. These dimensions are only for instructions and practice, as you can use any size that you need.

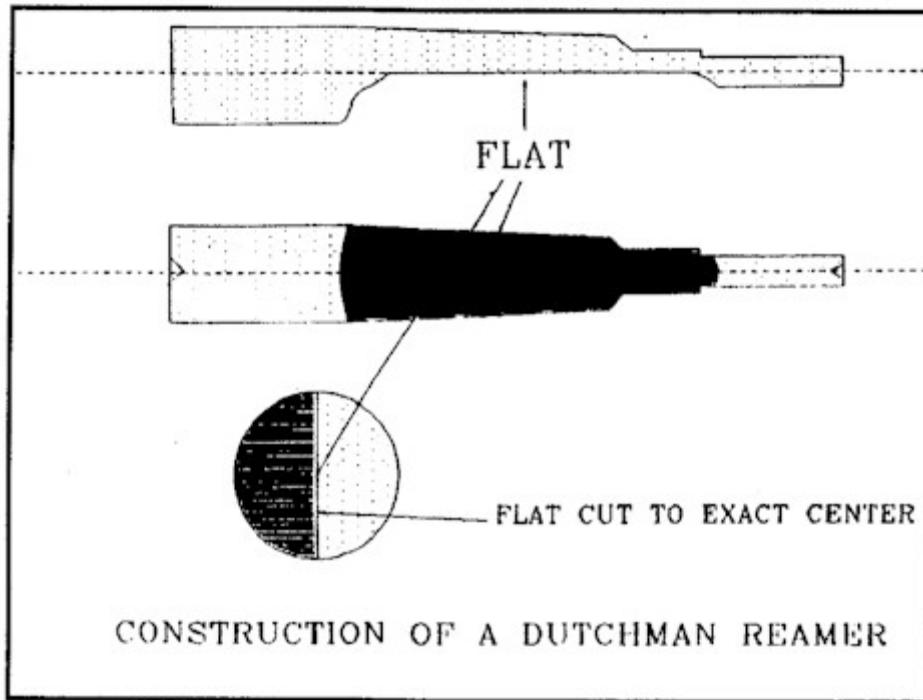
Do not take over .003 per side, as it will heat the metal too much. Go slow and easy.

When completed with the shank there is one more operation to do.

BARREL FITTING

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 will have to 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.

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.

Before using the reamer the first time recoat the flutes and cutting edges with the layout fluid. The reason for this is when you use the reamer the first time. If it gives you trouble the lay out fluid will show up any rub spots, and any place you do not have enough clearance.

The main problem that you may experience will be chatter.

A new reamer has more tendencies to chatter than one that has been used for some time.

BARREL FITTING

Reducing the speed may often reduce chatter; also sometimes increasing the feed will eliminate chatter. If for some reason there should be too little clearance the reamer will not cut freely, as the lands or margin will rub instead of cut against the walls of the hole. In most cases the reamer will either lodge or break off a flute or two, or break off.

BARREL FITTING

MAKING A CHAMBERING REAMER

The chambering reamer is no different in grinding than the barrel reamer, except there are more angles, etc. First look at the following drawings to see how 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.

Leave all diameters about .020 larger than shown. The tool steel blank should be cut 6 ½ 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.

ANGLE CUTTERS FOR MILLING FLUTES

The flutes should be milled with a 60-degree x 1 7/8 inch angle cutter. This should be used on all fluting of reamer blanks. The blank should be set up in the indexing attachment on the mill.

When cutting the flutes on the body of the reamer, be careful that you do not go under .062, as this will weaken the reamer. When cutting the flutes on the neck, you will have to go deeper than the setting on the body. Use plenty of cutting oil, and cut about a ½ to ¾ inch past where the neck of the reamer ends.

When you first start grinding, grind the pilot first. You want to decide if you want to have a throat on the reamer or not. If you want a throat, now is the time to grind it. The pilot should be ground about .001 smaller than the bore diameter.

All of these sizes are shown on chamber 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 the head space is taken from on rimless cases.

This can be set from the taper and angle charts. If you find 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. If you are making a reamer for a belted cartridge, this will have to be ground next.

BARREL FITTING

GRINDING THE CUTTING EDGES

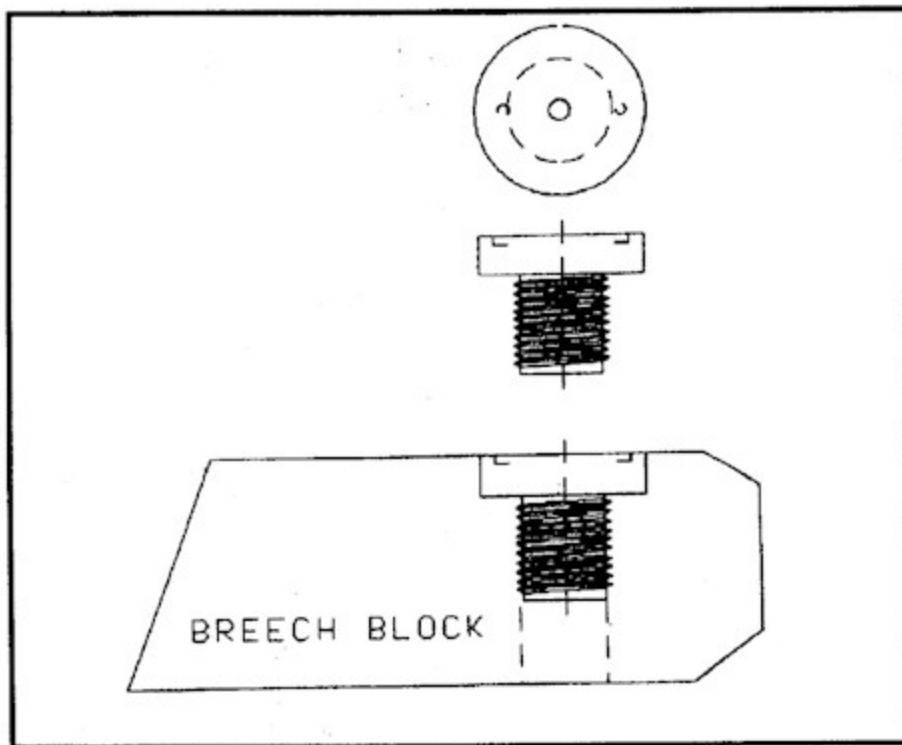
All the edges of the reamer have to be sharpened so the reamer will cut freely. This is the throat (if you use one) the start of the neck, the shoulder, the body of the case, and the start of the belt (if you are making a magnum). It is done the same way as shown in the chapter on barrel reamers. After making one or two reamers, you will understand fully how to do this.

BARREL FITTING

BUSHING FIRING PIN HOLES

There are many old single shot rifles around that with a little conversion would make excellent rifles for modern cartridges. In most cases the barrels are either replaced, or if they are big enough, they are lined with a smaller caliber. If the rifle is to be changed to a higher pressured cartridge the breech block needs to be re-bushed for a smaller firing pin.

This is necessary so that the primer doesn't blow back into the firing pin hole. If this happens the block will stick, and you will be unable to eject the fired cartridge. This is usually the lesser of two problems that can happen. The other problem is that the large firing pin may allow the primer to flow back into the firing pin hole and blow the primer. If this should happen, then high pressure gasses may very blow past the firing pin and tear up internal parts of the rifle.

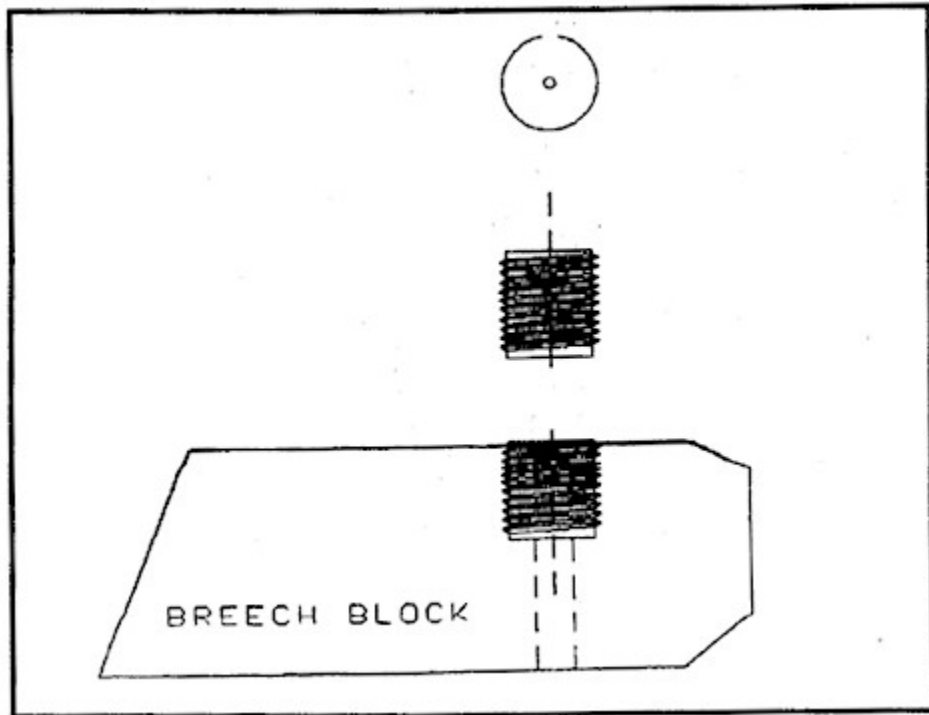


The worst danger is that these high-pressure gases may hit the shooter in the face, and possibly cause eye damage. To eliminate this problem, the firing pin hole needs to be bushed with a hardened block, and the firing pin, and hole reduced in size. This process is the same if you are converting a break-open shotgun to a rifle cartridge. You do need certain equipment to be able to do this properly. You will need some type of lathe, a milling machine or a sturdy drill press with a heavy machine vise to hold the block.

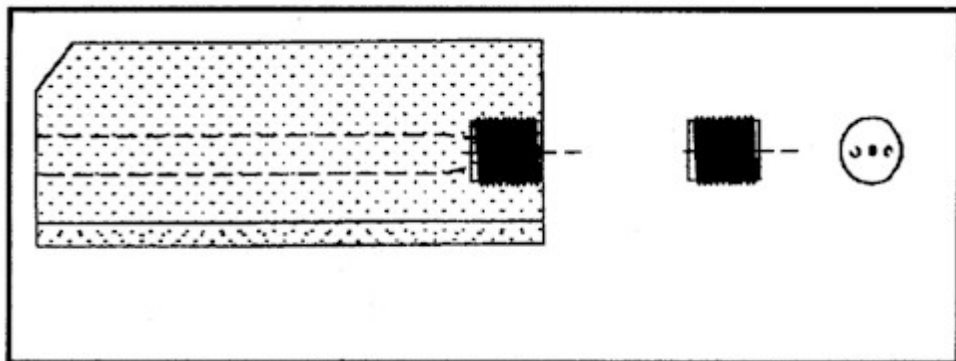
You will need to locate or make a 3/8 inch tap, with as fine a threads as possible to tap the hole with. This tap can be made quite easily. More on making taps in a later issue. You need to have drill rod in 1/2 & 3/8 inch size for bushings, 1/4 or 3/16-inch drill rod, depending on the size of the original firing pin. I will describe the process of converting a sliding breech block to a smaller firing pin.

BARREL FITTING

Remove the block from the rifle, and disassemble the parts from the block. You need to find a drill bit the size that just fits into the firing pin hole (the large size). The breech block can be drilled, on, a lathe after truing it in a 4-jaw chuck; the method that you use is up to you. The firing-pin hole in the bushing is first drilled with a 1/16 inch center-drill and then re-drilled with a #44 wire gauge drill, after which it is reamed from the rear either with a firing-pin reamer or a tapered pin reamer.



I prefer the taper pin reamer for this work, so that the firing pin hole is slightly relieved just back of the face of the bushing. As the firing-pin hole through the bushing may not be centered with the original firing-pin body hole in the breech block, it will be necessary to make a firing pin with an off-center nose if this is the case.



The original firing pin, if it is an odd shape, may be cut off square, after being annealed, replaced in the breech block and the location of the off-center nose located with a drill run through the firing-pin hole in the bushing.

BARREL FITTING

If the firing pin hole is not square to the face of the breech block, but rather on the angle, you will need a pilot ground on the drill bit to keep the bit from walking.

This is ground on the bit that you use to first drill the hole for the 3/8-inch tap clearance drill. The pilot for this drill is ground with the drill bit in the lathe using a tool post grinder.

I have however drilled out many breech blocks without this block, by using short bits, and opening up the hole in steps. Also if you have a assortment of end mills, they can be used to square up the face of the hole by using them as a drill, and going just deep enough to remove the angle. Once this angle is removed the breech block can then be drilled with the correct size drill. Drill the hole to about half the depth of the breech block, or slightly deeper with the first drill, then clean it up with the clearance drill that is the correct size or fine thread tap that you are using.

Once the hole is drilled, do not remove the breech block from the vise. You will need it clamped in the vise so that the hole can be tapped out straight. To do this the tap is centered in a center that is clamped in the chuck of the mill or drill press. The tap will either have a male or female center cut in the end of the tap, which we will use.

Oil the tap, and with slight pressure on the tap from the drill press, turn the tap into the drilled hole of the breech block. Back out every little bit to clean the chips from the cutting edge. The old pin is then drilled and tapped for a piece of tool-steel to be screwed in and turned up for a new nose, after which it is hardened and the temper then drawn to a deep blue in oil. The bushing is removed from the breech block, and 2 small holes drilled for a spanner wrench to make it easier to screw in and out, before hardening, and the temper drawn in oil at a dark purple and the bushing replaced in the block.

Check the surface carefully to see that it is flush with the face of the breech block, if not grind it down flush and polish it. A different method of bushing the breech block is to bore the block out to a square-bottomed bole, to a diameter to thread for 1/3 inch with a fine thread of 32 or more per inch, to a depth of 3/8 inch, A piece of 1/2 inch drill rod is then threaded to fit this bole tightly and screwed in against the bottom of the hole.

To get the right angle the old firing pin hole can be used as a guide. The firing pin hole will need to be drilled all the way through the block. Insert the drill in the back end of the block, and then clamp the other end in the chuck of the drill press. With the machine vise setting loose on the table of the drill press or milling machine, lower the spindle down so that the breech block will be inside of the jaws of vise.

This is done with the drill chucked and the cutting end of the drill in the back end of the breech block. With the breech block in between the jaws of the vise, carefully close the jaws of the vise on the sides of the breech block. Be careful that you do not spring the drill. Slide the vises lightly so that nothing will be in a bind. Once clamped, the firing pin hole can be drill all the way through the breech block at the same angle.

This is very important or when completed, the firing pin may hit the primer off center. You will need to now drill out the breech block for the harden bushing that will be screwed in the block.

BARREL FITTING

Having the 3/8-inch tap and the correct size drill for that tap set the breech block back into the machine vise for drilling. You can use the same procedure for lining up the block as you did before, except that you will drill out the hole this from the front, or cartridge end.

With the first drill chucked in the drill press or the mill, carefully clamp the breech block in the vise using the drill bit to get the correct angle. If the firing pin hole is right angle to the breech block, it is a simple matter to just drill out the block. It is best to drill the bushing hole out with the next size smaller drill, and then clean it out with the correct size drill.

I would, once the breech block is lined up clamp the vise securely to the drill or mill table to keep the vise from moving around while drilling. If you leave it loose, there will be a certain amount of chatter while starting the drill, and once the drill bushing is inserted, these chatter marks will not look good.

A drill fitting the body hole of the firing pin in the breech block is inserted from the rear and the bushing is drilled deeply enough with this to take the body length of the old firing pin. A 1/16 inch drill is then used to drill on through the bushing for the firing-pin nose, which hole is then enlarged with a #44 wire-gauge drill and reamed out as in the previous method.

The bushing is still part of the original 1/2 inch drill rod, several inches long, and is now unscrewed from the breech-block hardened and then the temper is drawn in oil to a dark purple, to blue. The bushing, still on the several inches of rod, is then screwed tightly back into the breech block and, using a high-speed steel hack-saw blade, is cut off just beyond the face of the breech-block, after which it is ground down flush with the breech-block face.

Go to the lathe and with some drill rod, make another firing pin the exact size of the old one, with the exception that the firing pin tip will just be a bit smaller than a #44 drill. Leave the tip .050 to .070 inch longer than the original. The block rises in closing this action, so even if the firing pinhole location is marked centrally with the bore, there is no assurance that the firing pin will always strike a primer centrally.

Firing pins should have a protrusion of .050 inch beyond the face of the breech-block when in the fired position. The firing pins must be made with as little protrusion as possible and still fire the cartridge every time without punching a hole in the primer.

The breech-block moves straight down when the action is opened but it starts to move down before the firing pin begins to retract so that the nose of the firing pin is dragged partly through the primer before being retracted mechanically, which results in a broken firing-pin if the protrusion of the pin is greater than .040 beyond the face of the breech block, and .035 is better yet, if it will fire every time. T

here will be a variation in the thickness of heads of rimmed cases, so this firing-pin protrusion checked with several different makes of cases, to be sure that primers will fire every time. After this is done the single shot rifle will take high-pressure cartridges safely.

Do not exceed the safety limit of the rifle. Some rifles are not made to handle much higher

BARREL FITTING

pressures.

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.

1.1 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.

1.2 DEPTH OF THREAD.

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

1.3 MINOR DIAMETER.

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

1.4 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 are 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.

1.5 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 1 by the number of threads per inch (N).

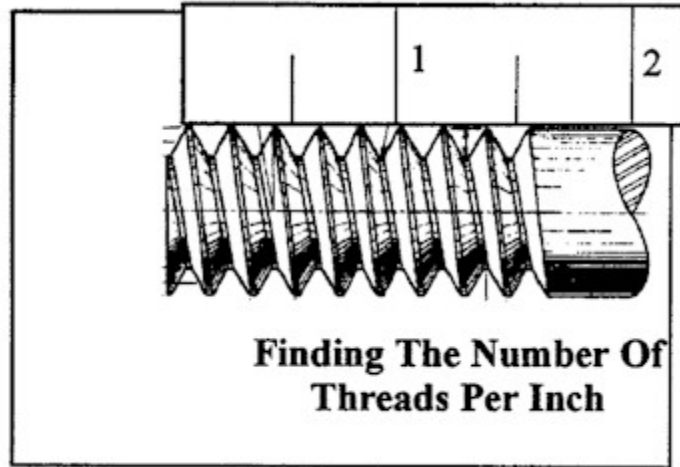
1.6 PITCH DIAMETER.

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.

BARREL FITTING

1.7 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 formula for the lead of a single thread screw, then, is:

1

P (in inches)

N (per inch)

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.

1.8 SINGLE SCREW THREADS.

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

1.9 ANGLE OF THREAD.

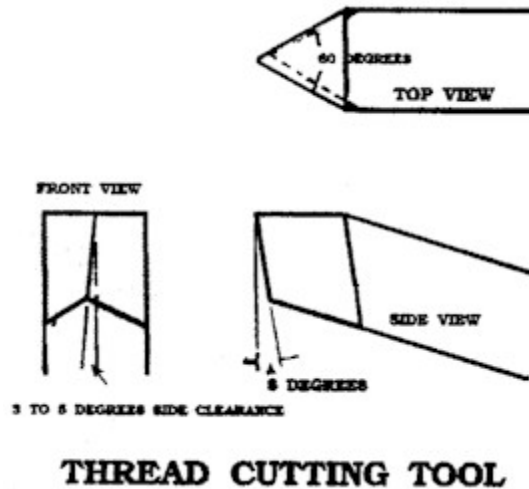
The thread angle is the angle included between the sides of the thread. The thread angle of the Unified National Form is 60 degrees.

1.10 FORMS OF SCREW THREADS

Screw threads are so widely used to connect parts that they are of prime importance when

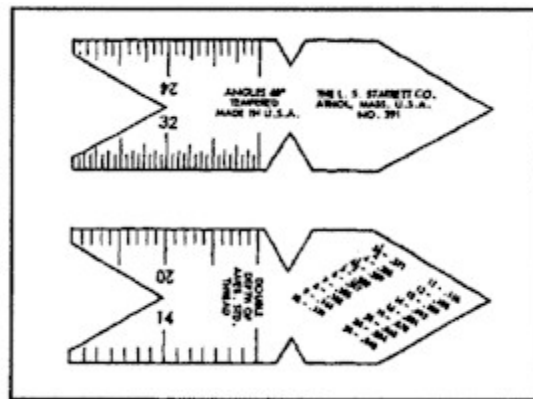
BARREL FITTING

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-Degree Worm threads.

1.11 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 modern 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.

BARREL FITTING

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.

To produce an American Standard thread on the engine lathe, the ratio of tool feed to work piece 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.

1.12 THREAD-CUTTING TOOLS AND THREAD FORMS

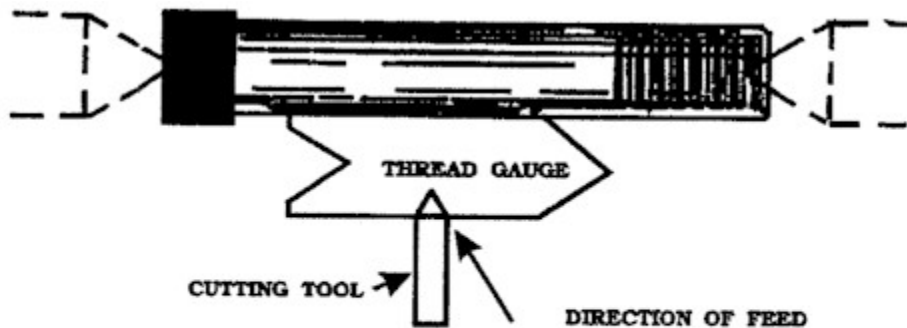
For each of the types of thread a tool bit to form that particular thread must be ground. Since the form of thread is the end product the grinding of the tool bit must take into consideration the clearance angle side rake form and other factors to achieve the result desired.

1.13 CLEARANCE

Because of the rapid advance of the cutting tool across the work piece, 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 degrees of side clearance.

BARREL FITTING



SETTING THE THREAD TOOL

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.

1.14 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 degrees. 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 centerline 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.

1.15 SIZE OF WORK PIECE BEFORE THREADING.

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 gauge the thread after it is cut.

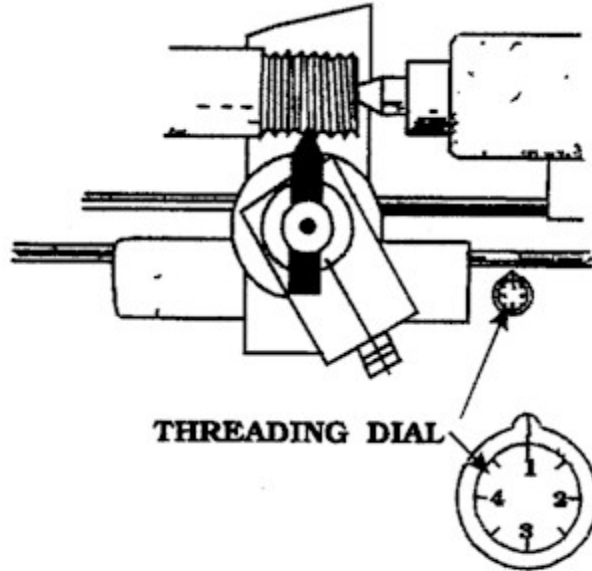
Before the threading operation is begun, the work piece needs to be turned to the maximum or major diameter of the thread specified on the blueprint or other specifications. This size is usually given as O.D. (outside diameter) expressed in inches and decimal parts of an inch. When cutting V type threads, 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 4-5 degrees. For instructions on straight turning, see the section on that operation.

1.16 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 work piece. Threads are often turned on shafts mounted between centers and revolved by a lathe

BARREL FITTING

dog.



THREADING SETUP

After this setup is made and the work is turned to the proper outside diameter, the work piece 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.

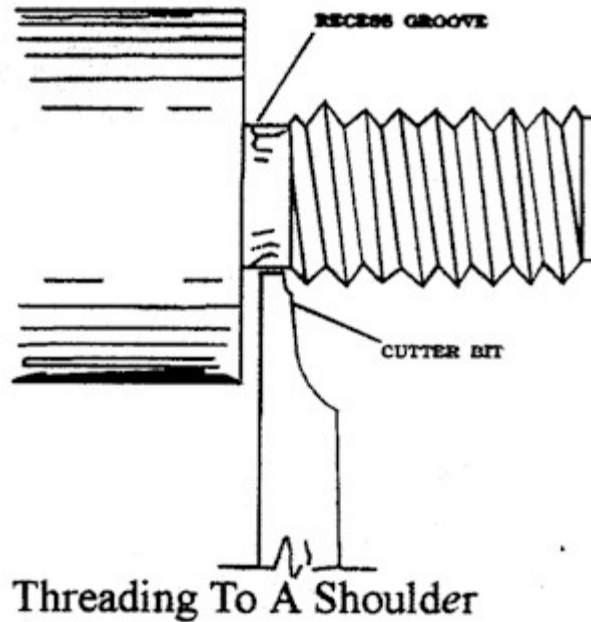
1.17 *SETUP FOR CUTTING AN EXTERNAL (V) THREAD ON TILE LATHE*

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 gearboxes 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 work piece by adjusting the cross-slide and setting its micrometer dial to zero. All adjustments for depth of cut can be made from these settings.

BARREL FITTING



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

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.

1.18 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.

BARREL FITTING

SUGGESTED DRILL SPEEDS FOR VARIOUS MATERIALS ¹	
Material to be Drilled	Cutting Speed (Surface Feet per Minute)
Aluminum and its alloys.....	200-300
Bakelite.....	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 Brinell).....	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.

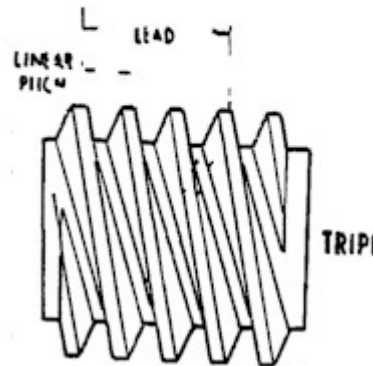
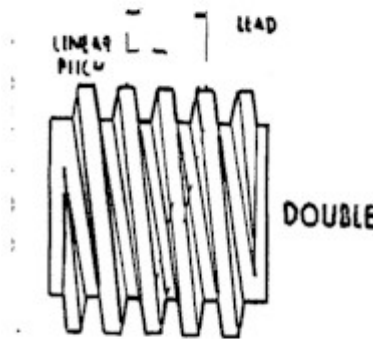
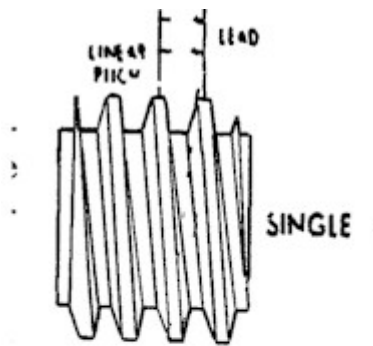
With lathes equipped with quick-change gears, most common threads can be set up by moving the required gears. When it is necessary to cut a thread of an odd pitch, the train of gears between the stud gear and the quick change-gear box will have to be changed. Metric threads can be cut on a lathe having 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 600 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 head stock and set at an angle of 29°.

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

BARREL FITTING



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 centerline of the work piece.

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.

BARREL FITTING

If the tool is fed in square with the work, it will have to 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 at the same time, 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 will have to 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.

1.19 *THREAD CUTTING DIAL*

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.

BARREL FITTING

When it is not necessary that the nut run 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 started 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. 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.

1.20 *THREAD DIAL INDICATOR*

Most modern 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 indicate positions at which the half nuts may be engaged. When the lathe is set up for cutting screw threads, the thread dial indicates 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 re-engaging the half nuts with the lead screw at a point that will assure that the tool follows exactly in the original cut.

BARREL FITTING

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 3 ½ threads per inch, the half nut must be engaged at any odd-numbered line on the dial.
- For quarter-threads, such as 2 ¼ or 3 ¼ 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).

1.21 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 head stock.

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 head stock, and the carriage is made to travel from left to right.

1.22 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 tail stock 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.

BARREL FITTING

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 gearbox on all modern lathes. Directions are shown on the gearbox 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 what 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 determine the total number of thousandths of an inch that the threading tool must be fed in by the compound feed screw, and 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 four 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.

1.23 CUTTING AN EXTERNAL THREADS.

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 degrees 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

BARREL FITTING

away from the work piece. Start the machine and engage the thread-chasing lever as before. Let the work piece 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 work piece slightly and turn on the power. When the tool is aligned properly, proceed as before.

1.24 RESETTING THE TOOL

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 1 B, 2B, and 3B applies to internal threads only. The three classes of fit are:

Class 1. Loose Fit

Class 2. Medium Fit

Class 3. Close Fit

1.25 THREAD FITS

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

1.26 LOOSE 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 be screwed on by hand.

1.27 MEDIUM FIT.

This is used on fasteners where accuracy of fit is highly important, and where no looseness is permitted. A wrench or some other 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.

1.28 CLOSE FIT

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.

BARREL FITTING

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.

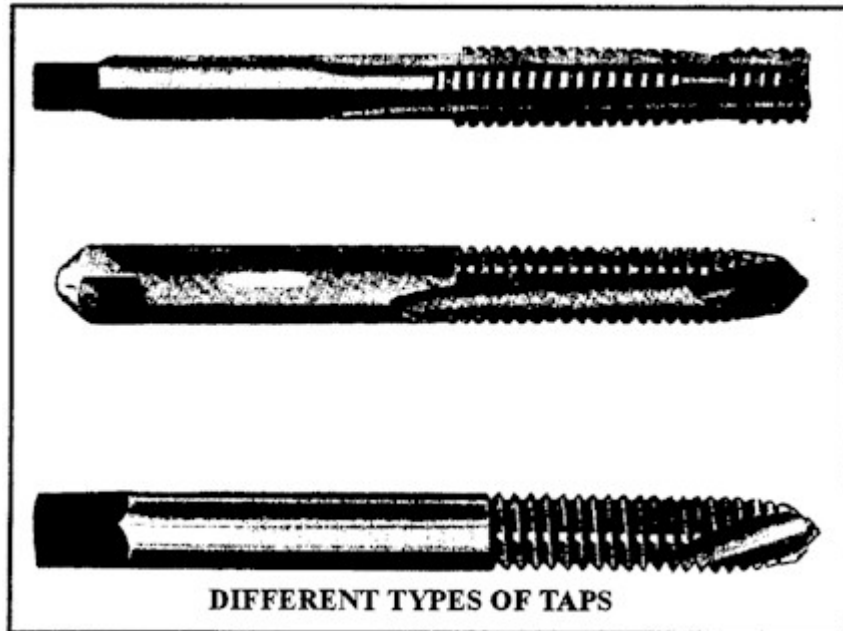
This way, each tap cuts less metal each time and is less liable to be broken by twisting. It takes longer, but is worth the extra time by saving broken taps.

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.

BARREL FITTING

TAPS AND THREADING

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 to 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 keyholes of the chuck to turn it, at the same time 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

BARREL FITTING

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 ½ inch stock turned to a 600 point.

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	.0687	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	.1068	.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
⅜	.3125	18	.2764	.2443	.2524	3°	40'	.0454
½	.3750	16	.3344	.2983	.3073	3°	24'	.0678
⅝	.4375	14	.3911	.3499	.3602	3°	20'	.0933
¾	.5000	13	.4500	.4056	.4167	3°	7'	.1257
⅞	.5625	12	.5084	.4603	.4723	2°	59'	.1620
1	.6250	11	.5660	.5135	.5266	2°	56'	.2018
1 ⅛	.7500	10	.6850	.6273	.6417	2°	40'	.3020
1 ¼	.8750	9	.8028	.7387	.7547	2°	31'	.4193
1 ½	1.0000	8	.9188	.8466	.8647	2°	29'	.5510
1 ⅝	1.1250	7	1.0322	.9497	.9704	2°	31'	.6931
1 ¾	1.2500	7	1.1572	1.0747	1.0954	2°	15'	.8898
1 ⅞	1.3750	6	1.2667	1.1705	1.1946	2°	24'	1.0541
1 ¾	1.5000	6	1.3917	1.2955	1.3196	2°	11'	1.2938
1 ¾	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
2 ¼	2.2500	4½	2.1057	1.9774	2.0094	1°	55'	3.0212
2 ½	2.5000	4	2.3376	2.1933	2.2294	1°	57'	3.7161
2 ¾	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
3 ¼	3.2500	4	3.0876	2.9433	2.9794	1°	29'	6.7205
3 ½	3.5000	4	3.3376	3.1933	3.2294	1°	22'	7.9183
3 ¾	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.

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

BARREL FITTING

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.

UNIFIED AND AMERICAN STANDARD FINE AND EXTRA-FINE THREAD SERIES								
Size	Basic Major Diam.	Tbds. per Inch	Basic Pitch Diam.	Minor Diam. Ext. Tbds.	Minor Diam. Int. Tbds.	Lead Angle, Pitch Diam.		Area, Minor Diam.
	Inches		Inches	Inches	Inches	Deg.	Min.	Sq. In.
FINE THREAD SERIES								
0 (.000)	.0600	80	.0510	.0447	.0465	4°	23'	.0015
1 (.073)	.0730	72	.0640	.0560	.0580	3°	57'	.0024
2 (.086)	.0860	64	.0750	.0668	.0691	3°	45'	.0034
3 (.099)	.0990	56	.0874	.0771	.0797	3°	43'	.0045
4 (.112)	.1120	48	.0985	.0864	.0894	3°	51'	.0057
5 (.125)	.1250	44	.1102	.0971	.1004	3°	45'	.0072
6 (.138)	.1380	40	.1218	.1078	.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
$\frac{1}{8}$ "	.2500	28	.2268	.2062	.2113	2°	52'	.0326
$\frac{3}{16}$ "	.3125	24	.2854	.2614	.2674	2°	40'	.0524
$\frac{1}{4}$ "	.3750	24	.3479	.3239	.3299	2°	11'	.0809
$\frac{5}{16}$ "	.4375	20	.4050	.3762	.3834	2°	15'	.1090
$\frac{3}{8}$ "	.5000	20	.4675	.4367	.4450	1°	57'	.1486
$\frac{7}{16}$ "	.5625	18	.5264	.4943	.5024	1°	55'	.1888
$\frac{1}{2}$ "	.6250	18	.5889	.5568	.5649	1°	43'	.2400
$\frac{9}{16}$ "	.7500	16	.7094	.6733	.6823	1°	36'	.3513
$\frac{5}{8}$ "	.8750	14	.8286	.7874	.7977	1°	34'	.4805
1	1.0000	14	.9538	.9124	.9227	1°	22'	.6464
1 1/8	1.0000	12	.9459	.8978	.9098	1°	36'	.6245
1 1/4	1.1250	12	1.0709	1.0228	1.0348	1°	25'	.8118
1 3/8	1.2500	12	1.1959	1.1478	1.1598	1°	16'	1.0237
1 1/2	1.3750	12	1.3209	1.2728	1.2848	1°	9'	1.2802
1 3/4	1.5000	12	1.4459	1.3978	1.4098	1°	3'	1.5212
EXTRA-FINE THREAD SERIES								
12 (.216)	.2160	32	.1957	.1777	.1822	2°	55'	.0242
$\frac{1}{8}$ "	.2500	32	.2297	.2117	.2162	2°	29'	.0344
$\frac{3}{16}$ "	.3125	32	.2922	.2742	.2787	1°	57'	.0581
$\frac{1}{4}$ "	.3750	32	.3547	.3367	.3412	1°	26'	.0878
$\frac{5}{16}$ "	.4375	28	.4143	.3937	.3988	1°	34'	.1201
$\frac{3}{8}$ "	.5000	28	.4768	.4562	.4613	1°	22'	.1616
$\frac{7}{16}$ "	.5625	24	.5354	.5114	.5174	1°	25'	.2080
$\frac{1}{2}$ "	.6250	24	.5979	.5739	.5799	1°	16'	.2560
$\frac{9}{16}$ "	.6875	24	.6604	.6364	.6424	1°	9'	.3151
$\frac{5}{8}$ "	.7500	20	.7175	.6887	.6959	1°	16'	.3685
$\frac{3}{4}$ "	.8125	20	.7800	.7512	.7584	1°	10'	.4388
$\frac{7}{8}$ "	.8750	20	.8425	.8137	.8209	1°	5'	.5153
1	.9375	20	.9050	.8762	.8834	1°	0'	.5979
$1 \frac{1}{8}$ "	1.0000	20	.9675	.9387	.9459	0	57'	.6866
$1 \frac{1}{4}$ "	1.0625	18	1.0264	.9943	1.0024	0	59'	.7702
$1 \frac{3}{8}$ "	1.1250	18	1.0889	1.0568	1.0649	0	56'	.8705
$1 \frac{1}{2}$ "	1.1875	18	1.1514	1.1193	1.1274	0	53'	.9770
$1 \frac{3}{4}$ "	1.2500	18	1.2139	1.1818	1.1899	0	50'	1.0895
$1 \frac{7}{8}$ "	1.3125	18	1.2764	1.2443	1.2524	0	48'	1.2082
1 5/8	1.3750	18	1.3389	1.3068	1.3149	0	45'	1.3330
1 3/4	1.4375	18	1.4014	1.3693	1.3774	0	43'	1.4640
1 7/8	1.5000	18	1.4639	1.4318	1.4399	0	42'	1.6011
1 5/4	1.5625	18	1.5264	1.4943	1.5024	0	40'	1.7444
1 3/2	1.6250	18	1.5889	1.5568	1.5649	0	38'	1.8937
1 1/2	1.6875	18	1.6514	1.6193	1.6274	0	37'	2.0493
1 1/4	1.7500	16	1.7094	1.6733	1.6823	0	40'	2.1873
2	2.0000	16	1.9594	1.9223	1.9323	0	35'	2.8917

Source: Machinery's Handbook.

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

BARREL FITTING

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 at the same time 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.

These can be set to bear on the work and guide the die square. You should also have a plain diestock 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 many button dies sold at bargain stores that are not true to size. These can cause much 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 it's 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.