

Introduction to 9x20 Lathe Operations

Compiled by: Colin Feaver

Revision Date stamp: 04/04/2005 @ 06:30 pm

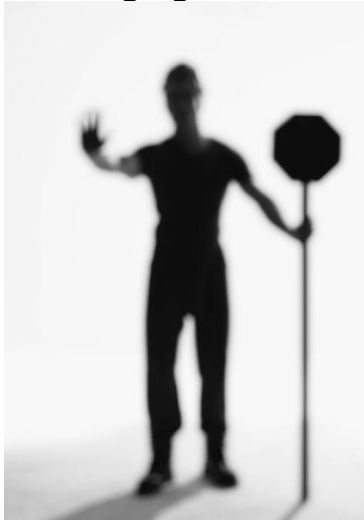
This is a non-profit document produced by and for the members of

<http://groups.yahoo.com/group/9x20Lathe/>

The information contained herein is presented for intellectual enrichment only and may not change hands for monetary gain. The Author, Researchers, Contributors, Manufacturers, Suppliers and Members assume no liability whatsoever from the use of information contained herein. This document is compiled from freely accessible information on the internet, in particular the 7x10 minilathe website, the 9x20 lathe group archives and its members. If you would like to make a comment or addition your input would be welcome.

Foreword

The Asian 920 Lathe offers an inexpensive solution for persons wanting a capable machine without having to dip into “Johnny’s College Fund”. The Asian 920 Lathes are assembled from components originating in various Chinese factories. These components generally converge and are assembled and adjusted at one Main Chinese Factory and the finished product exported and sold by many retail outlets with or without various accessories and under various Brand Names, associated Colors and aftermarket support. There is some consensus that these machines are referred to by hobbyists and metal working enthusiasts as “Kit Lathes” because the fit, finish and assembly may be somewhat incomplete and rough. However, the person willing to invest some “Sweat Equity” may fit and finish one of these machines to a point rivaling equipment costing many times that which was invested. This is not to say that these machines are incapable of running right out of the box; most of them do.



!!!!WARNING!!!!

For your own safety read all instruction and safety manuals before operating any equipment you may use. Understand all safety issues related to your equipment. Read section one, “Safety”, of the Grizzly G4000 instruction manual before you attempt to use any Asian 9x20 lathe.

Table of contents

First steps in turning by the late <i>Edgar T. Westbury</i>	page 4
Lathe Introduction-----	page 5
Terminology-----	page 5
First tool set recommendations-----	page 11
Lathe tool grinding-----	page 12
Facing -----	page 20
Drilling -----	page 28
Turning -----	page 35
Parting -----	page 45
Threading-----	page 53
Knurling -----	page 71
Boring -----	page 75
Accessories-----	page 75
Links-----	page 92
Compare 9x20 lathes and prices-----	page 95
Conclusion-----	page 98

First steps in turning

by “Artificer” (*The late Edgar T. Westbury*)

The answer to queries on toolholders may appear to some beginners to be putting the cart before the horse, so far as they are concerned. What they first want to know is how to use the tools already available, before trying to make special tools. But to the amateur, lathe operation cannot be entirely dissociated from tool making, and the readers who asked direct questions on this subject were given answers which were, I hope, equally direct and practical. So far as elementary problems in the technique of lathe work are encountered, however, the raw beginner sometimes finds the information obtained from handbooks or workshop articles, however explicit, does not cover his particular requirements. There are admittedly some things which cannot be learned “from the book”; one cannot expect to acquire skill in mechanical craft, without practice, any more than one can learn to swim, skate or play the guitar. First steps, in the literal sense, must necessarily be halting and inefficient, with more failures than successes, and this is one of the facts of life which must be accepted by anyone who wishes to achieve anything worthwhile.

I well remember the very first time I tried to do a job on a lathe, without the aid of a book or any supervisory guidance. I attempted to take a cut on a slender shaft projecting a long way out of the chuck, with no back centre support. Needless to say, the work rode up over the tool, and finished up in the semblance of a dog’s hind leg, rather than a straight shaft ! This and other experiences, such as jobs which flew out of the chuck, and tools that dug in and broke off, were all part of my early self-training; they might nearly all have been avoided if I had had an instructor at my elbow, but one advantage of learning things the hard way is that one is unlikely to forget the lessons learnt. One makes many mistakes, but learns to avoid repeating them and to use the knowledge gained for the general improvement of workmanship. This applies to all workshop crafts, besides lathe work. As the proverb says, “the burnt child fears the fire,” and the novice who breaks off a tap in a component to which hours of work have been devoted will do his best to see that it does not occur again. So may even our mistakes and mishaps be put to good account, if we have the wit to apply what they teach us. Perhaps this may be regarded as just a string of smug platitudes, but the extent to which the most elementary questions should be explained in detail is not always easy to estimate. Some beginners demand a “blow by blow” description of simple operations, which others may regard as kindergarten stuff. I hope I shall not be accused of writing “down” to readers, by dealing with some of the queries from readers who appear to have had no previous workshop experience. However naive the question, it does not imply ignorance or lack of intelligence on the part of the querist and it is never beneath serious consideration. Although it is often possible to refer the querists to books or articles dealing with matters relevant to particular problems, they may not provide all the detailed information required.

Introduction

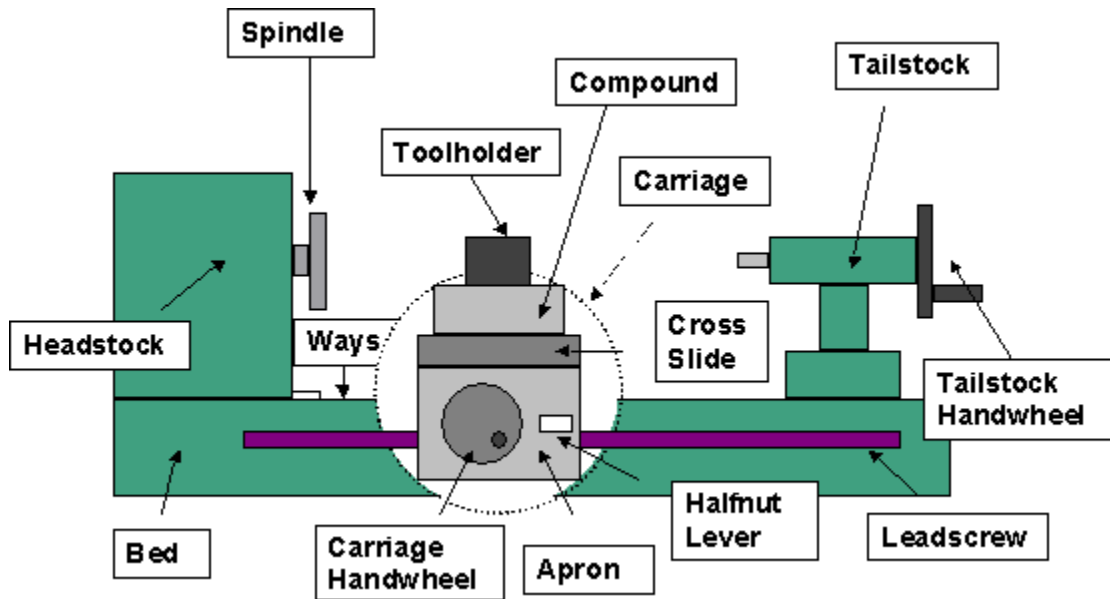
If you are new to metalworking lathes and lathe work, this document will help you understand some of the basic concepts, terminology and capabilities. In essence, a lathe rotates a cylindrical workpiece along its axis and removes material from the workpiece to form it into a specific shape.

On a woodworking lathe, the cutting tools are usually hand-held against a support and are moved in and out and back and forth along the surface of the work by hand to form a shape such as a table leg.

On metalworking lathes, the cutting tools are held rigidly in a tool holder that is mounted on a movable platform called the carriage. The tool is moved in and out by means of hand cranks and back and forth either by hand cranking or under power from the lathe. The result is that material can be removed from the workpiece under very precise control to produce shapes that are truly precision made. Dimensional accuracies of one-one-thousandth of an inch (.001") are typical. Because of the inherent rotational nature of a lathe, the vast majority of the work produced on it is basically cylindrical in form. In spite of this, the lathe is an extremely versatile machine capable of producing a surprising variety of objects.

Terminology

To gain a good understanding of the lathe, you will need to know the names of the various components, as illustrated in the diagram. The carriage (in the circled area) consists of the apron, the vertical casting on which the carriage handwheel is mounted; and the saddle (not shown), the H-shaped casting that rides on the ways to which the apron is attached.



Lathe Dimensions

When comparing the size and working capacities of metal lathes there are several key dimensions to consider:

Swing over bed	The diameter of the largest workpiece that can be rotated on the spindle without hitting the bed. This is the first of the two numbers used to describe the size of a metal lathe. In the case of the 9x20 lathe it is 9".
Distance between centers	The longest piece of work that can be held between a center in the headstock and a center in the tailstock. (See glossary below for more information). This is the second of the two numbers used to describe the lathe size.
Swing over the carriage	The diameter of the largest workpiece that can rotate over the carriage without hitting it. On the 9x lathes this is about 5"
Diameter of spindle through-hole	The diameter of the hole that passes through the spindle. On the 9x lathes (or any lathe having a #3 Morse Taper spindle) it is about 3/4". When facing relatively long stock, the free end of the stock can pass through the spindle if it is no larger than the through-hole diameter.

Here's a table summarizing some of the dimensions for a 7x12 and 9x20 lathe:

	7x12	9x20
Swing over bed	7"	9"
Distance between centers	12"	20"
Swing over carriage	4"	5"
Spindle Taper	#3MT	#3MT
Spindle through-hole diameter	3/4"	3/4"
Tailstock taper	#2MT	#2MT

Glossary of Lathe Terms

Apron	Front part of the carriage assembly on which the carriage hand wheel is mounted
Bed	Main supporting casting running the length of the lathe
Between Centers	1. A dimension representing the maximum length of a work piece that can be turned between centers. A 9x20 lathe is 19" between centers; a 7x12 lathe is 12" between centers. Lathe vendors sometimes inaccurately represent this number. 2. A method of holding a work piece by mounting it between a center in the headstock spindle and a center in the tailstock spindle (see Center).
Carriage	Assembly that moves the tool post and cutting tool along the ways
Carriage Handwheel	A wheel with a handle used to move the carriage by hand by means of a rack and pinion drive
Center	A precision ground tapered cylinder with a 60° pointed tip and a Morse Taper shaft. Used in the tailstock to support the end of a long work piece. May also be used in the headstock spindle to support work between centers at both ends.
Center_Drill	A short, stubby drill used to form a pilot hole for drilling and a shallow countersunk hole for mounting the end of a workpiece on a center.
Centerline	An imaginary line extending from the center of the spindle through the center of the tailstock ram, representing the central axis of the lathe around which the work rotates.
Chuck	A clamping device for holding work in the lathe or for holding drills in the tailstock.

Compound	Movable platform on which the toolpost is mounted; can be set at an angle to the workpiece. Also known as the compound slide and compound rest.
Compound Handwheel	A wheel with a handle used to move the compound slide in and out. Also known as the compound feed.
Cross slide	Platform that moves perpendicular to the lathe axis under control of the cross-slide handwheel
Cross-slide Handwheel	A wheel with a handle used to move the cross-slide in and out. Also known as the cross feed.
Faceplate	A metal plate with a flat face that is mounted on the lathe spindle to hold irregularly shaped work.
Facing	A lathe operation in which metal is removed from the end of workpiece to create a smooth perpendicular surface, or face.
Gib	A length of steel or brass with a diamond-shaped cross-section that engages with one side of dovetail and can be adjusted by means of screws to take up any slack in the dovetail slide. Used to adjust the dovetail for optimum tightness and to compensate for wear.
Halfnut	A nut formed from two halves which clamp around the leadscrew under control of the halfnut lever to move the carriage under power driven from the leadscrew.
Halfnut Lever	Lever to engage the carriage with the leadscrew to move the carriage under power
Headstock	The main casting mounted on the left end of the bed, in which the spindle is mounted. Houses the spindle speed change gears.
Leadscrew	Precision screw that runs the length of the bed. Used to drive the carriage under power for turning and thread cutting operations. Smaller leadscrews are used within the cross-slide and compound to move those parts by precise amounts.
Morse Taper	A taper of specific dimensions used to mate matching male and female parts such that they lock together tightly and concentrically. Tapers are of various sizes such as #0, #1, #2, #3, etc. with larger numbers representing larger sizes. The spindle of the mini-lathe has a #3 Morse Taper and the tailstock ram has a #2 Morse Taper.
Saddle	A casting, shaped like an "H" when viewed from above, which rides along the ways. Along with the apron, it is one of the two main components that make up the carriage.
Spindle	Main rotating shaft on which the chuck or other work holding device is mounted. It is mounted in precision bearings and passes through the headstock.

Spindle Through-hole	A dimension indicating the minimum diameter of the hole that passes through the spindle. A workpiece with a diameter smaller than this can pass through the spindle to facilitate working on long pieces of work. On the lathe it is 3/4" but can safely be reamed out to 13/16"
Swing	A dimension representing the largest diameter workpiece that a lathe can rotate. The 9x20 lathe has a 9" swing, meaning that the maximum size workpiece that can rotate without hitting the bed is 9" in diameter.
Tailstock	Cast iron assembly that can slide along the ways and be locked in place. Used to hold long work in place or to mount a drill chuck for drilling into the end of the work.
Tailstock Handwheel	A wheel with a handle used to move the tailstock ram in and out of the tailstock casting.
Tailstock Ram	A piston-type shaft that can be moved in and out of the tailstock by turning the tailstock handwheel. Has a tapered internal bore to accept a #2 Morse Taper shank
Tool	A cutting tool used to remove metal from a workpiece; usually made of High Speed Steel or carbide.
Tool Blank	A piece of High Speed Steel from which a cutting tool is ground on a bench grinder. Typically 5/16" square by 2 1/2" long for mini-lathe use.
Toolpost	A holding device mounted on the compound into which the cutting tool is clamped
Turning	A lathe operation in which metal is removed from the outside diameter of the workpiece, thus reducing its diameter to a desired size.
Ways	Precision ground surfaces along the top of the bed on which the saddle rides. The ways are precisely aligned with the centerline of the lathe.

First Tool Set recommendations

Posted in Yahoo Groups 9x20lathe group by cba_melbourne

3/8" shank bits are the biggest I would ever consider putting in such a small lathe. In fact, 5/16" has plenty of rigidity reserves for the deepest cuts you can take, even if you upgraded your spindle drive & motor to three times the HP it now has. 80% of my tools are 5/16" (=8mm), the remainder are 1/4" (=6mm).

The toolpost may take up to 1/2" shanks, but it's not intended for 1/2" tools; rather it allows you to clamp, say, a sleeve holder for round boring tools, or a clamping holder for a parting blade. Both holders may well require up to a 1/2" toolpost slot, but the tools they clamp are much smaller.

With a smaller tool you can reach far better into confined spaces. Not all jobs are just plain facing or turning, there are many more complex workpiece shapes. And even if only plain turning/facing, imagine a rod of say 1/2" dia between centers, how do you avoid hitting the center turning it with a 1/2" tool?

On this lathe, there is nothing to be gained by a tool larger than 5/16". The deepest cuts you will ever take, in soft material, are 1/8" (or 3mm). Now look at the insert of a 5/16" tool: you are going to utilize only a fraction of the cutting edge. The remainder will just be in your way when doing more complex jobs.

If using HSS blanks, you will find that grinding a 5/16" tool is 3x faster and easier than a 1/2" tool and a 1/4" tool is 2-3x faster and easier than a 5/16" tool to grind. Why grind a cutting edge 12x longer than you will ever need, just because a set of 1/2" tools was on special? You would regret it many times over.

Rigidity? 5/16" tools are used in high speed production CNC lathes. If you ever have seen at what speeds and depths of cut, you will never again blame tool rigidity on your 9" lathe.

Tool shimming? In a 4-way toolpost you need to shim every tool, regardless of size, to correct center height. If you use indexable carbide insert tools, one shim will do for all. Their center height is very accurate and does not change in use because you do not resharpen inserts (they are one way, wear and toss).

If you are out shopping for a first tool set, I recommend 5/16" as the ideal standard to start with. As you gain experience and tackle smaller jobs you may get some 1/4" or smaller tools later. But I guarantee you will never need any bigger unless you get a bigger lathe first. If you get carbide insert tools, you can choose between CCMT and TCMT style inserts (diamond shaped or triangular). Its personal preference, both are good. Settle for one style for all tools, as spares come in boxes of 10 and you do not want to stock too many shapes. Choose uncoated inserts, they are sharper and can take finer cuts, and this is very important until you learn to take deep cuts to a precise dimension. Choose an intermediate tip radius (.4mm is a good compromise). Only go for tools with POSITIVE rake, your lathe has NOT the rigidity to handle negative rake. Avoid the old style reversible inserts with rectangular cross section, however cheap, they are no good on a hobby lathe. 5/16" tools may cost a little more than 1/2" tools, because they are not sold by low-cost tool discounters. Dealers specialized for us hobby users have them. The Indian made Glanze sets are excellent at 1/3 the price of brand name holders sold by the big expensive industrial tool shops.

If you decide to start with HSS blanks, go for some 1/4" and 5/16". You will need a grinder too, and you need to practice some grinding. The best blanks are HSS-Co and well worth the little extra cost.

Chris

Lathe Tool Grinding

Tool grinding is part science and part art, but can be an enjoyable side activity to working with the lathe. The goal here is to teach beginners enough to get them started with a few basic tools. You can, of course, buy ready-made carbide-tipped cutting tools. If good quality, these work well and last a long time. The real advantage to grinding your own tools is not cost savings, but having the ability to make a tool for whatever purpose you may run across in your work. For example, I have made some very small boring tools from 3/16" tool blanks which are quite handy for boring out a small hole, say .373 in diameter, to press-fit a 3/8" shaft.

The essence of lathe tool grinding, as I think of it, is to undercut the tip of the tool to provide 'relief' so that the metal just below the cutting tip does not contact the work. This concentrates enough cutting force on the tip to cut into the metal of the workpiece.

Most of the regular cutting tools I make are undercut on the front and left edge of the tool. Since most tools are designed to cut while moving from right to left (towards the headstock) it is not necessary to provide relief on the right side of the tool. Additionally, I usually grind a similar relief, or rake, on the top surface of the tool.

When you order your lathe, be sure to order about 10 5/16" x 2 1/2" High Speed Steel (HSS) tool blanks. (Note: the older, non-Sieg, Homier/Speedway lathes use 3/8" x 2 1/2" tools) I usually get mine from Enco. They are normally about \$1 each but often go on sale for about \$0.80.

The four sides of the blank are ground to a smooth, shiny finish. The ends are a coarse finish with a preformed angle of about 15 degrees.

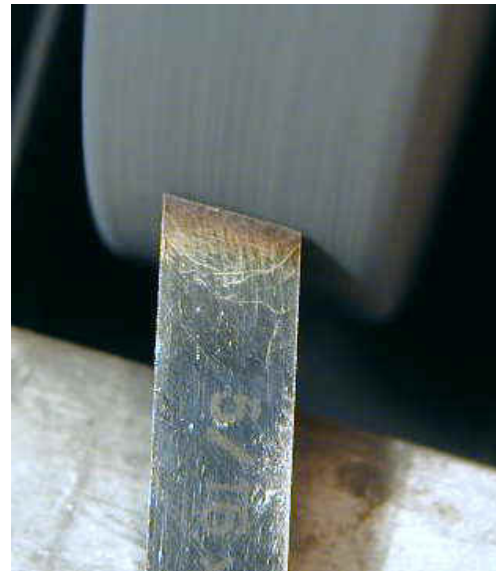
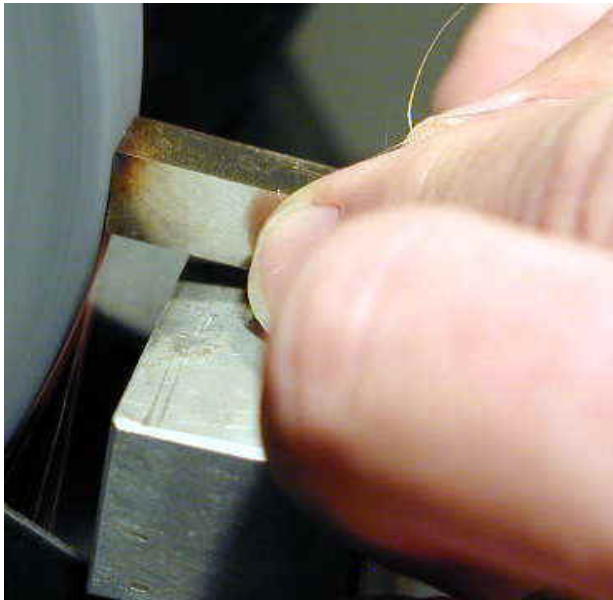


We will use a simple four-step procedure to make our cutting tool

1. Grind the end relief
2. Grind the left side relief
3. Grind the top rake
4. Round the tip

Grinding the End Relief

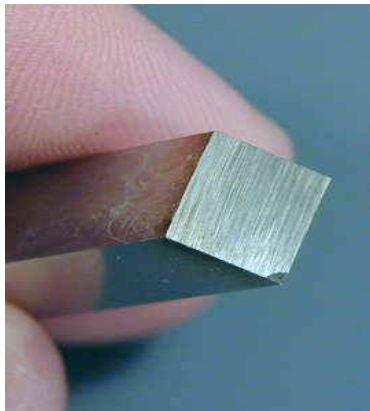
First we will grind the end of the tool blank. Use the coarse wheel of your bench grinder and hold the tool blank angled downwards from the tip to the rear and with the tip pointing to the left about 10-15 degrees.. The tip of the tool blank should be a little below the center line of the wheel.



Grinding causes the tool blank to get quite hot so you will need to dip the end of the tool into a water bath every 15 seconds or so during the grinding operation. When you see the tip of the tool start to discolor from the heat it's a good time to make a cooling dip. Fortunately, HSS does not conduct the heat to your fingers very fast, but you can get burned if you go too long between cooling dips.



Here's a picture of the tool after grinding the end:



Grinding the Left Side Relief

Now we'll grind the left side of the tool. The procedure is essentially the same except that we hold the tool with the side at about a 10 degree angle to the grinding wheel.



Grinding the Top Rake

Now we grind the top surface to form the rake. Be careful during this operation not to grind down the cutting edge or you will end up with a tool whose tip is below the center line of the lathe. If this happens, the tool will leave a little nub at the center of the workpiece when you make a facing cut. The usual remedy is to use a thin piece of shim stock under the tool to bring it back up to the center line. A much nicer solution is an adjustable-height tool holder.



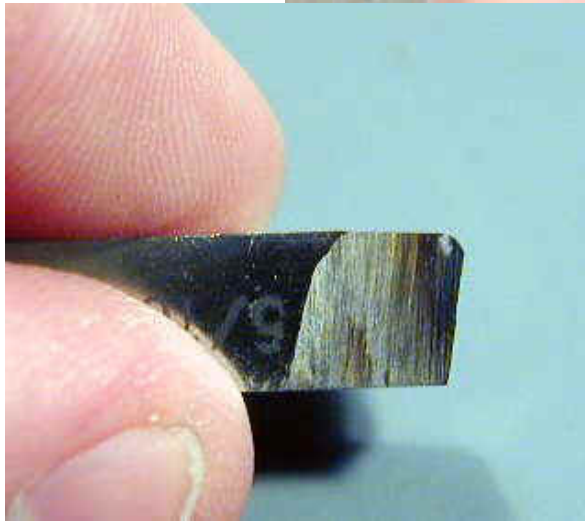
After this operation we have a working tool with a very sharp tip. This tool is useful as-is for operations that need a sharp tip to turn down to an interior edge such as a shoulder.



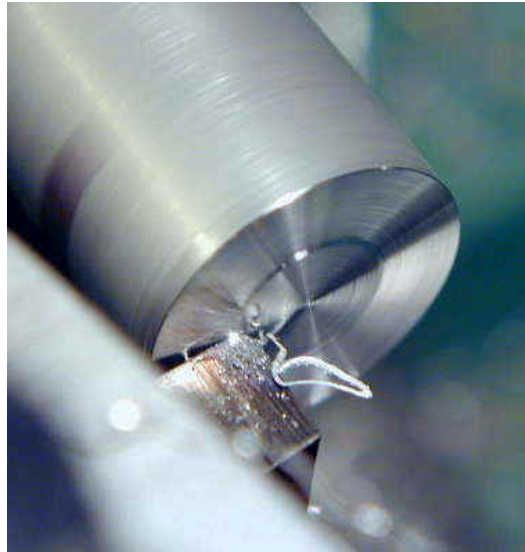
Rounding the Tip

We will round the tip to form a tool that is useful for facing and turning.

Hold the tool so the tip touches the wheel and with the tool tilted downward. Rotate the tool against the wheel to round the tip to about a $\frac{1}{32}$ " radius.



Here's the finished tool in action making a finishing cut on a facing operation:



As a final step, you may wish to smooth the cutting tip on a fine diamond hone or oilstone. I've found that the tip tends to get smoothed pretty quickly after a few cuts, so I usually skip this step, but it does make a difference if you need a fine finish.

So now you know how to grind the most basic of cutting tools. There are many other types of tools that you can grind including shaping tools, cutoff tools and boring tools. Here's some additional info posted by Brian Pitt:

You usually want to set the tool on center to about .003-.005" above center but almost never below (the workpiece will try to roll over the top and chatter). Cutoff, threading and most carbides should be on center while HSS for general turning can be a hair above to make up for the deflection of the workpiece and machine which will bring it back on center. It can also add a little burnishing action as the work rubs the front face of the bit and will give smoother finishes. The side and top rake reduce the amount of cutting force and heat generated and helps to control the chip by giving it a different angle to curl on depending on the material being cut. Softer and stringier materials like aluminum and 1018 steel need more rake to give a knife like edge while materials that break up easily like brass and cast iron use less rake to keep the tool from digging in and grabbing.

One of the most important facts often overlooked by beginners is that the cutting face of the grindstone wheel quickly becomes dulled and clogged with metal particles. To maintain an aggressive cutting face it is essential to refurbish the wheel face frequently with a dressing tool. I use a dressing tool with a single industrial diamond point for this purpose. I got it from Enco for around \$15. I use this tool to refresh the grinder wheel after about every 10 minutes of grinding time. It makes a big difference.

Here's some great information on various types of tool rake, posted by Dub Thornton:

No; rake angles and front relief are NOT the same. Front relief is the angle ground into the front of your tool which allows ONLY the front cutting edge to contact the workpiece. If the tool contacts the workpiece below the cutting edge, you will get a "rubbing" action, and the tool cannot bite into the workpiece. Side relief (or clearance) is the angle ground into the side of the tool which allows only the side cutting edge of the tool to contact the workpiece.

There are two rake angles, both on the top of the tool. Back Rake is the angle from the tip of the cutting tool toward the back of the tool. It may be positive, neutral, or negative. If it slopes down from the tip of the tool toward the back of tool, it is positive rake. An upward slope would be negative rake, and neutral is self-explanatory.

Rake angles, particularly back rake, may be built into the tool holders. The old lantern type holders I grew up on usually had a positive back rake angle built in. This was a help when grinding your tools, as you did not have to grind any back rake into the tool itself. When grinding threads though, a neutral back rake angle is desirable, thus one had to grind a negative rake angle (point of tool pointed downward) to compensate for the positive back rake angle built into the tool holders.

Side rake is the angle from the side cutting edge of the tool toward the opposite side of tool (across the top of the tool). It can also be ground for negative, neutral, or positive side rake. A negative rake angle is usually used for brittle materials, such as brass, which are notorious for "hogging in" as you cut. A positive rake angle would increase this "hogging in" action, while a negative rake angle will push the tool away from the work, eliminating the tendency to "hog in". "Hogging in" is an old term for the material grabbing the tool, pulling it into the material for a deeper cut than you are set up for. Backlash in your machine increases the possibility for such "hogging in". Many projects are spoiled, as well as tools broken, etc, by this action.

A very general rule of thumb, for heavy, roughing cuts, use less clearance and rake angles. This leaves more material in the cutting tool to withstand the pressures of heavy cutting, plus more "beef" in the tool means more ability to carry heat away from the cutting edges. For finish cuts, and turning such "soft" materials as aluminum, use more clearance and more rake angles for a better finish. I have seen many references on this reflector to Top Rake, which is confusing to me, as it does not define whether side rake or back rake is being referred to.

Lathe Operation: Facing

Facing Operations

Facing is the process of removing metal from the end of a workpiece to produce a flat surface. Most often, the workpiece is cylindrical, but using a 4-jaw chuck you can face rectangular or odd-shaped work to form cubes and other non-cylindrical shapes.

When a lathe cutting tool removes metal it applies considerable tangential force to the workpiece. To safely perform a facing operation the end of the workpiece must be as close as possible to the jaws of the chuck. The workpiece should not extend more than 2-3 times its diameter from the chuck jaws unless a steady rest is used to support the free end.

Cutting Speeds

If you read many books on machining, you will find a lot of information about the correct cutting speed for the movement of the cutting tool in relation to the workpiece. You must consider the rotational speed of the workpiece and the movement of the tool relative to the workpiece. Basically, the softer the metal, the faster the cutting. Don't worry too much about determining the correct cutting speed; working with the 9x20 lathe for hobby purposes, you will quickly develop a feel for how fast you should go.

Until you get a feel for the proper speeds, start with relatively low speeds and work up to faster speeds. Higher speeds, and particularly the HI range, are used for operations such as polishing, not cutting.

Preparing for the Facing Cut

First, make sure the tumbler gear lever is in the neutral position so that the leadscrew does not rotate. This is very important since you will clamp the half nut on the leadscrew during the facing operation to keep the saddle from being forced back away from the end of the workpiece by the force of the cutting operation.

Clamp the workpiece tightly in the 3-jaw chuck. To get the work properly centered, close the jaws until they just touch the surface of the work, then twist the workpiece in the jaws to seat it; then tighten the jaws. It's good practice to tighten the jaws from all 3 chuck key positions to ensure even gripping by the jaws.



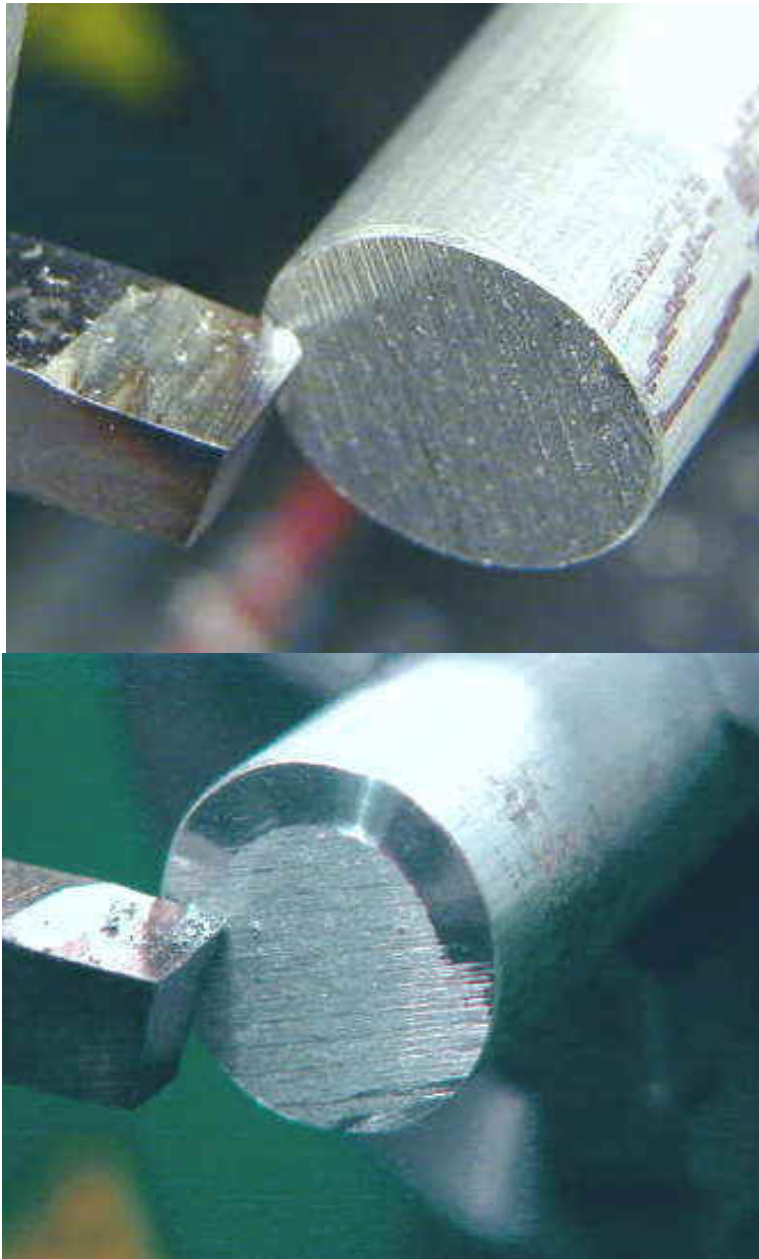
Choose a cutting tool with a slightly rounded tip. A tool with a sharp pointed tip will cut little grooves across the face of the work and prevent you from getting a nice smooth surface. Clamp the cutting tool in the tool post and turn the toolpost so that the tip of the cutting tool will meet the end of the workpiece at a slight angle. It is important that the tip of the cutting tool be right at the centerline of the lathe; if it is too high or tool low you will be left with a little bump at the center of the face.

Clamp the toolpost in place and advance the carriage until the tool is about even with the end of the workpiece. Make sure that the compound is not all the way at the end of its travel towards the chuck.

Set the lathe to its lowest speed and turn it on. Make sure the leadscrew is not turning. Turn the lathe off and press the Halfnut lever down to engage the half-nut with the leadscrew. You may have to work the carriage back and forth a little to get good engagement. Locking the half-nut to the leadscrew will prevent the carriage from moving back away from the workpiece during the facing operation. If this were to happen, the end of workpiece would be a slight cone shape instead of perfectly flat - or the tool might stop cutting entirely. A much better way to lock the carriage in place is to add a carriage lock to your lathe as described on Varmint Al's site and on my mini lathe mods page.

Beginning the Facing Cut

Use the compound crank to advance the tip of the tool until it just touches the end of the workpiece. Use the cross feed crank to back off the tool until it is beyond the diameter of the workpiece. Turn the lathe on and adjust the speed to a few hundred RPM. Now slowly advance the cross feed crank to move the tool towards the workpiece. When the tool touches the workpiece it should start to remove metal from the end. Continue advancing the tool until it reaches the center of the workpiece and then crank the tool back in the opposite direction (towards you) until it is back past the edge of the workpiece.



Since we started with the tool just touching the end of the workpiece, you probably removed very little metal on this pass. This is a good idea until you get used to how aggressively you can remove metal without stalling the lathe. Also, since the end of the workpiece can be very uneven, you want to avoid gouging the tool into the end of the workpiece if it hits a high spot.

The Roughing Cut

Use the compound crank to advance the tool towards the chuck about .010" (ten one-thousandths of an inch, or one one-hundredth of an inch). If the compound is set at a 90 degrees to the cross slide (which is how I usually set mine) then each division you turn the crank will advance the tool .001" (one one-thousandth of an inch) toward the chuck.

If the compound is set at some other angle, say 30 degrees to the cross slide, then it will advance the tool less than .001" for each division. The exact amount is determined by the trigonometric sine of the angle. Since the sine of 30 degrees is .5 the tool would advance .0005" (five ten-thousandths or 1/2 of one one-thousandth of an inch) for each division in this example. Always wondered why you needed trig? Now you know ;-)

Here's a picture of the first pass of a facing operation. I am removing about .010" of metal in this pass.



Cutting on the Return Pass

If you crank the tool back towards you after it reaches the center of the workpiece you will notice that it removes a small amount of metal on the return pass. This is because the surface is not perfectly smooth and it is removing metal from the high spots. If you need to remove a lot of metal, to reduce the workpiece to a specific length, for example, you can take advantage of this return cut to remove more metal on each pass by advancing the tool a small ways into the workpiece on the return pass. Since the tool must plunge into the face of the workpiece, this works best with a fairly sharp pointed tool.

The Finishing Cut

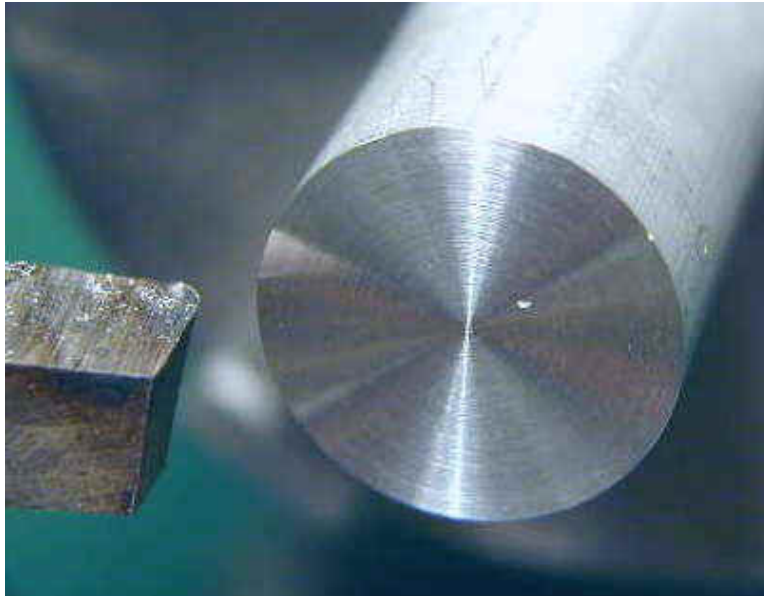
Depending on how rough the end of the workpiece was to begin with and how large the diameter is, you may need to make 3 or more passes to get a nice smooth finish across the face. These initial passes are called roughing passes and remove a relatively large amount of metal.

When you get the face pretty smooth you can make a final finishing cut to remove just .001" to .003" of metal and get a nice smooth surface. The finishing cut can also be made at higher RPM (say 1500 RPM) to get a smoother finish.

Here's a picture of a finishing cut in progress, removing about .002" of metal at around 1000 RPM.



Here's a picture of the finished face of the workpiece.



The picture above shows what happens if the tip of your cutting tool is below the center line of the lathe - a little nub is left at the center of the workpiece. The same thing happens if the tool is too high but the nub will have more of a cone shape in that case. If the tool is too low, place a suitable thickness of shim stock underneath the tool in the tool holder. If it's too high, grind the top down a few thou.

Filing the Edge

Facing operations leave a rather sharp edge on the end of the workpiece. It's a good idea to smooth this edge down with a file to give it a nice chamfer and to avoid cutting yourself on it. With the lathe running at fairly low speed, bring a smooth cut file up to the end of the workpiece at a 45 degree angle and apply a little pressure to the file.



Right (above) - left hand holding tang end of file

Wrong (below) - left hand reaching over spinning chuck!

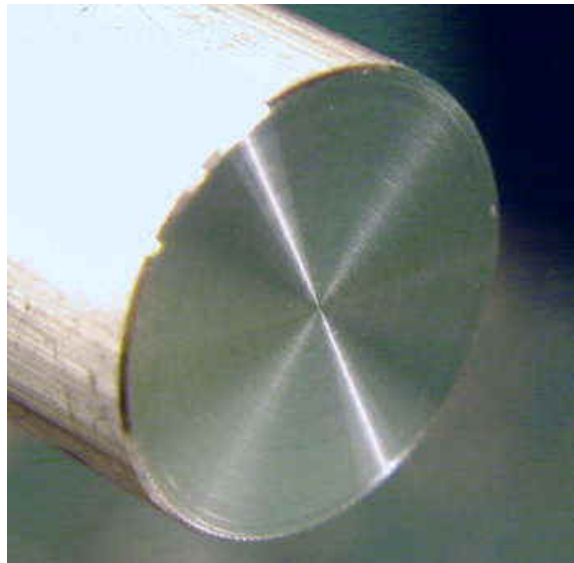


Note that I am holding the tang end of the file in my *left* hand and the tip in my right hand to avoid having my left hand reaching over the spinning chuck. Thanks to John Chatham of the 7x10 interest group for this important safety tip.

Be sure the metal tang of the file is protected by a wooden handle or other material. Without this precaution there is a danger that the file tang could be forced back into your palm or wrist with serious consequences. Mine is covered by several layers of heat shrink tubing and tape. I use the colored tape in various combinations to help me quickly find the tool I need from my working pile of tools. Some guys replace each tool to a specific holder after they use it. Another good idea is to drill a hole in an old golf ball and just jam the file tang into the golf ball. Instant handle!



This picture shows the finished surface and beveled edge. This is what a good facing cut should look like; smooth even surface with no raised bump in the center. Lay an accurate straight edge across the surface of the face and you should not be able to see any light under the edge. If you detect a slight convex shape, the carriage may be moving back away from the cut.



Lathe Operation: Drilling

Drilling Operations

The alignment between the headstock and tailstock of the lathe enables you to drill holes that are precisely centered in a cylindrical piece of stock. I tried doing this once with my drill press and vise before I had the lathe; it did not turn out too well.

Before you drill into the end of a workpiece you should first face the end as described in the facing operations section. The next step is to start the drill hole using a center drill - a stiff, stubby drill with a short tip. If you try to drill a hole without first center drilling, the drill will almost certainly wander off center, producing a hole that is oversized and misaligned. We hate that!

Center drills come in various sizes such as #00, #0, #1 - #5, etc. You can purchase sets of #1-#5 for under \$5.00 on sale from several suppliers.



Preparing to Drill

Before drilling you need to make sure that the drill chuck is firmly seated in the tailstock. With the chuck arbor loosely inserted in the tailstock bore, crank the tailstock bore out about 1/2". Lock the tailstock to the ways, then thrust the chuck firmly back towards the tailstock to firmly seat the arbor in the Morse taper of the tailstock. (The chuck is removed from the tailstock by cranking the tailstock ram back until the arbor is forced out).

Choose a center drill with a diameter similar to that of the hole that you intend to drill. Insert the center drill in the jaws of the tailstock chuck and tighten the chuck until the jaws just start to grip the drill. Since the goal is to make the drill as stiff as possible, you don't want it to extend very far from the tip of the jaws. Twist the drill to seat it and dislodge any metal chips or other crud that might keep the drill from seating properly. Now tighten the chuck. It's good practice to use 2 or 3 of the chuck key holes to ensure even tightening (but all three may be impossible to reach given the tight confines of the 9x20).

Slide the tailstock along the ways until the tip of the center drill is about 1/4" from the end of the workpiece and tighten the tailstock clamp nut. The locking lever for the tailstock ram should be just snug - not enough to impede the movement of the ram, but enough to ensure that the ram is as rigid as possible.



Cutting Fluid

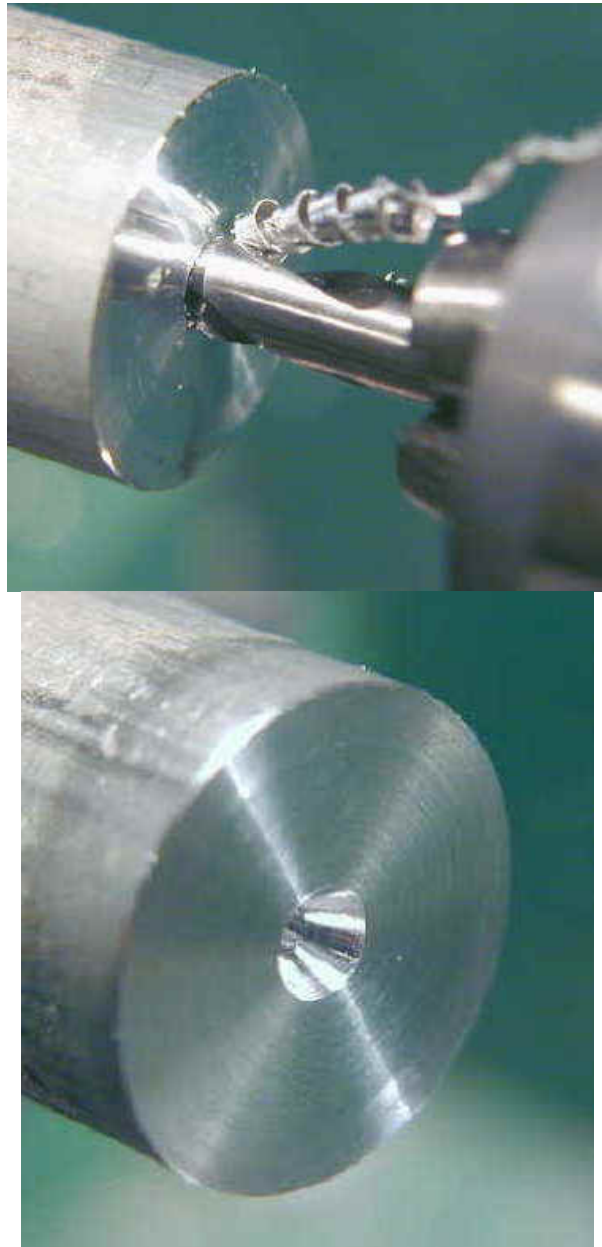
Unless I'm working with brass, I nearly always use a cutting fluid when drilling. Particularly with aluminum which helps to ensure a smooth and accurate hole. I use Tap Magic brand cutting fluid but there are several other excellent brands available.



You only need a few drops at a time, so a small can should last for a long time. I use a small needle tipped bottle to apply fluid to the work. The bottle originally contained light oil & was obtained at Home Depot.

Center Drilling

Turn on the lathe and set the speed to around 600 RPM. Use the tailstock crank to advance the drill slowly into the end of the workpiece and continue until the conical section of the center drill is about 3/4ths of the way into the workpiece. This is as far as you need to go with the center drill since its purpose is just to make a starter hole for the regular drill. Back the center drill out and stop the lathe.



Drilling the Hole

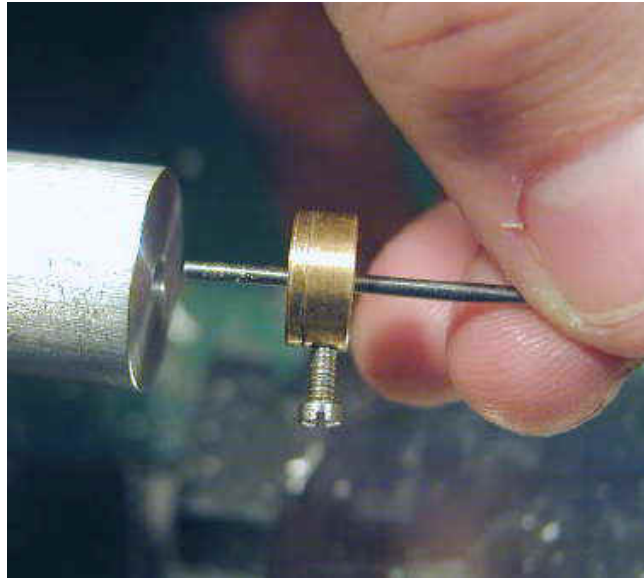
Loosen the tailstock clamp nut and slide the tailstock back to the end of the ways. Remove the center drill from the chuck and insert a regular drill and tighten it down in the chuck. Slide the tailstock until the tip of the drill is about 1/4" from the workpiece and then lock the tailstock in place. Place a few drops of cutting fluid on the tip of the drill, then start the lathe and drill into the workpiece as before, at 400 to 600 RPM.



After advancing the drill about twice its diameter, back it out of the hole and use a brush to remove the metal chips from the tip of the drill. Add a few more drops of cutting fluid if necessary, then continue drilling, backing the drill out to remove chips about every 2 diameters of depth.

Measuring Drilling Depth

Unless you are drilling completely through a fairly short workpiece you will generally need a way to measure the depth of the hole so that you can stop at the desired depth. One of the first accessories I made on the lathe is a simple depth gauge - just a small cylinder of brass with a locking screw which slides on a piece of 1/16" drill rod about 3" long. It's quite handy for checking the depth of holes. You can use a shop rule to set the brass slider to the desired depth and then lock it in place with the little set screw.

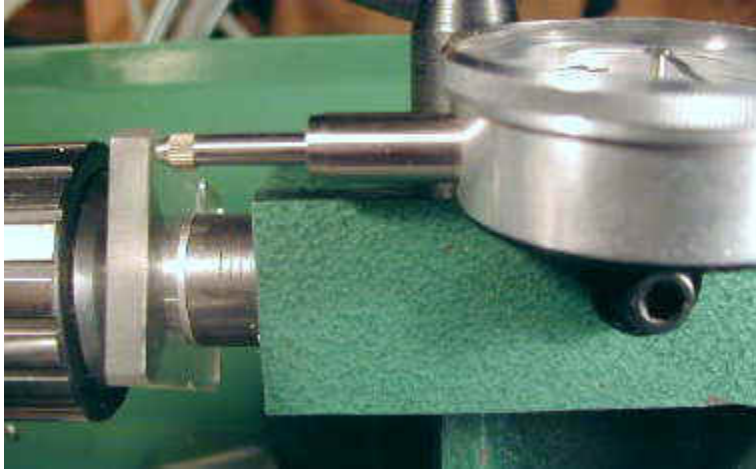


Another way to measure the depth is to use the graduated markings on the barrel of the tailstock. These are not easy to see, though.



If you need real accuracy, Varmint Al came up with a nifty idea to mount a 1" dial indicator on the tailstock. The tip of the DI touches a plastic plate

that is mounted on the tailstock ram. The DI is bolted into a 1/4-20 hole drilled and tapped in the side of the tailstock. If you make this mod to your lathe, remove the ram from the tailstock before drilling the mounting hole for the DI to avoid drilling into the ram.



Drilling Deep Holes, Blind Holes and Large Holes

In the world of metalwork, a "deep" hole is any hole more than about 3 times the drill diameter. A blind hole is one in which you are not drilling all the way through the workpiece; i.e. the bottom end is closed. The critical thing when drilling such holes is to frequently back the drill completely out of the hole to allow the chips to escape from the hole. You need to do this repeatedly each time you advance the drill by about twice its diameter. Failure to follow this procedure will cause the chips to bind in the hole, weld to the drill, and create a hole with an uneven and rough diameter. Cutting fluid will also help to keep the chips from binding to the drill or the sides of the hole.

Large holes are relative to the size of the machine and for the mini-lathe, I consider a hole larger than 3/8" to be "large". If you try to drill a large hole, say 1/2", starting with a 1/2" drill, you may not get a nice clean hole because too much material is being removed at one time. It is better to drill the hole in stages, starting, say, with a 5/16" drill, then a 3/8" and so forth, until you work up to the 1/2" drill for the final pass. This way, the large drill is removing only a small amount of material around the perimeter of the hole and will have a much easier job to do.

Lathe Operation: Turning

Turning Operations

Turning is the removal of metal from the outer diameter of a rotating cylindrical workpiece. Turning is used to reduce the diameter of the workpiece, usually to a specified dimension, and to produce a smooth finish on the metal. Often the workpiece will be turned so that adjacent sections have different diameters.

Chucking the Workpiece

We will be working with a piece of 3/4" diameter 6061 aluminum about 2 inches long. A workpiece such as this which is relatively short compared to its diameter is stiff enough that we can safely turn it in the three jaw chuck without supporting the free end of the work.



For longer workpieces we would need to face and center drill the free end and use a dead or live center in the tailstock to support the workpiece. Without such support, the force of the tool on the workpiece would cause it to bend away from the tool, producing a strangely shaped result. Additionally, there is danger that the work could be forced to loosen in the chuck jaws and fly out as a dangerous projectile.

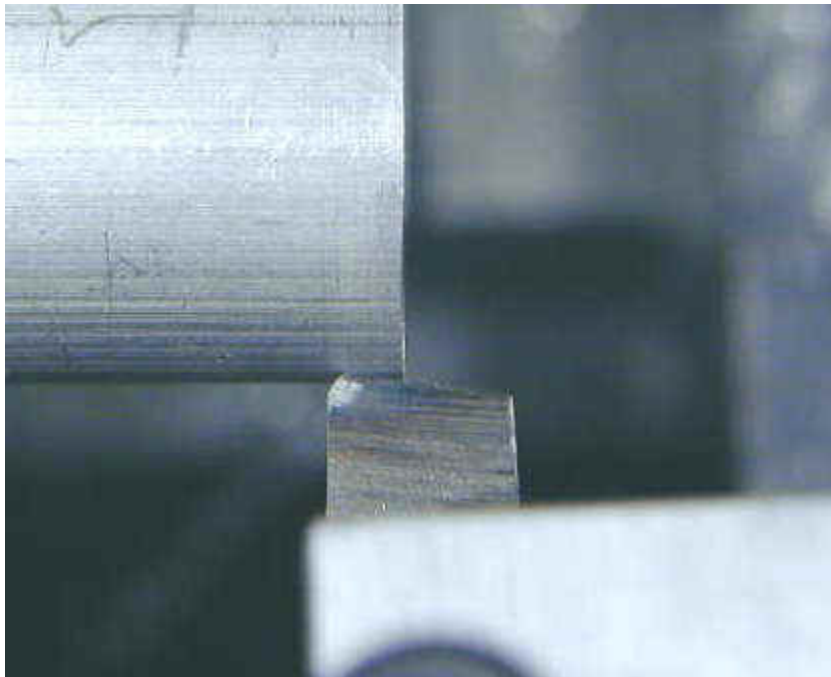
Insert the workpiece in the 3-jaw chuck and tighten down the jaws until they just start to grip the workpiece. Rotate the workpiece to ensure that it is seated evenly and to dislodge any chips or grit on the surface that might keep it from seating evenly. You want the workpiece to be as parallel as possible with the center line of the lathe. Imagine an exaggerated example

where the workpiece is skewed at an angle in the chuck and you can easily visualize why this is important. Tighten the chuck using each of the three chuck key positions to ensure a tight and even grip.

Adjusting the Tool Bit

Choose a tool bit with a slightly rounded tip. This type of tool should produce a nice smooth finish. For more aggressive cutting, if you need to remove a lot of metal, you might choose a tool with a sharper tip. Make sure that the tool is tightly clamped in the tool holder.

Adjust the angle of the toolholder so the tool is approximately perpendicular to the side of the workpiece. Because the front edge of the tool is ground at an angle, the left side of the tip should engage the work, but not the entire front edge of the tool. The angle of the compound is not critical; I usually keep mine at 90 degrees so that the compound dial advances the work .001" per division towards the chuck.



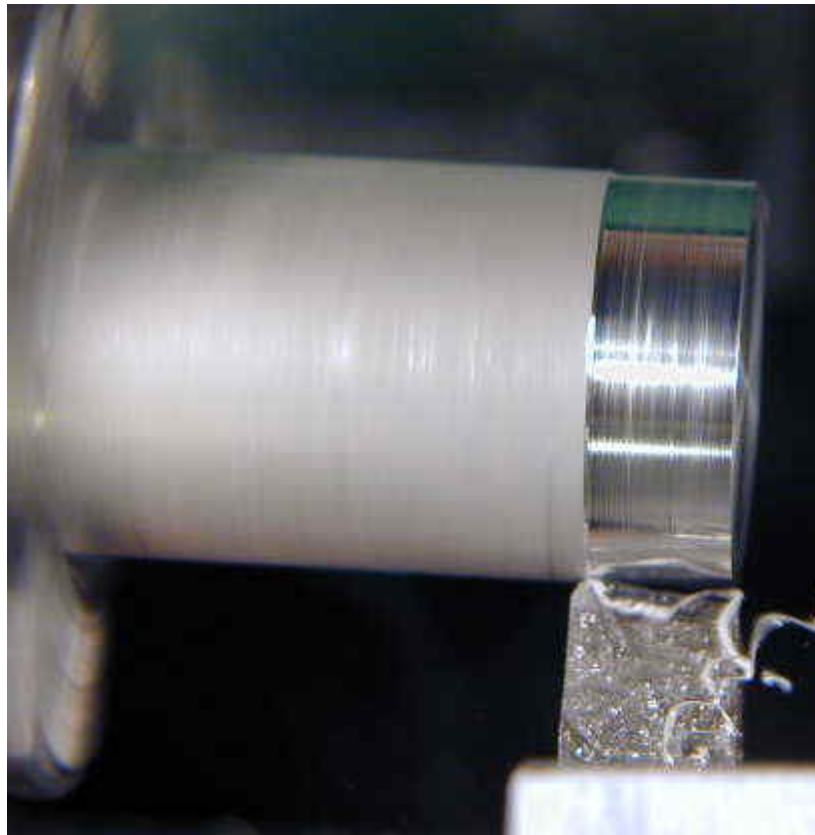
Make sure the half nut and feed levers are disengaged. If necessary, back off the cross slide until the tip of the tool is back beyond the diameter of the work. Move the carriage until the tip of the tool is near the free end of the workpiece, then advance the cross slide until the tip of the tool just touches the side of the work. Move the carriage to the right until the tip of the tool is just beyond the free end of the work.

Cutting Speeds

If you read many books on machining you will find a lot of information about the correct cutting speed for the movement of the cutting tool in relation to the workpiece. You must consider the rotational speed of the workpiece and the movement of the tool relative to the workpiece. Basically, the softer the metal the faster the cutting. Don't worry too much about determining the correct cutting speed; working with the 9x20 for hobby purposes, you will quickly develop a feel for how fast you should go. Until you get a feel for the proper speeds, start with relatively low speeds and work up to faster speeds.

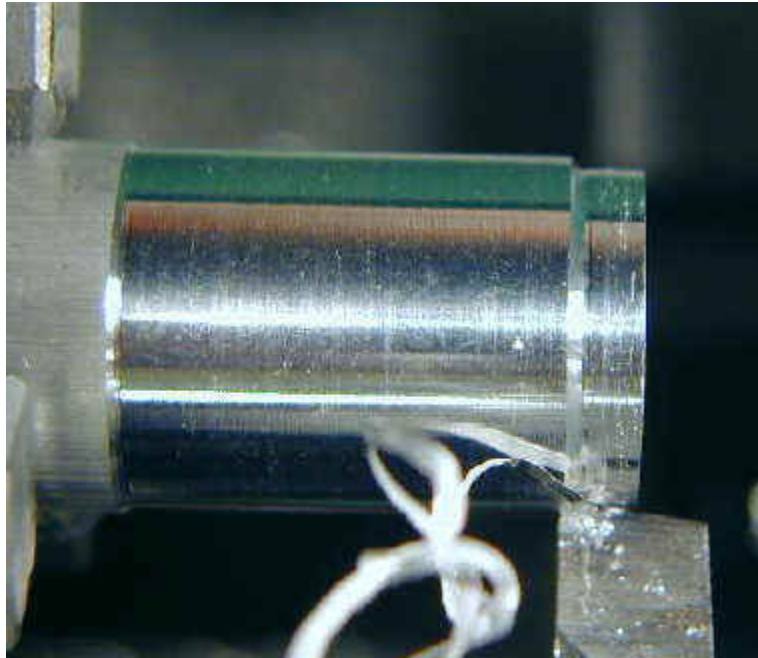
Turning with Hand Feed

As always, wear safety glasses and keep your face well away from the work since this operation will throw off hot chips and/or sharp spirals of metal.



Now advance the cross slide crank about 10 divisions or .010" (ten one-thousandths or one one-hundredth of an inch). Turn the carriage handwheel counterclockwise to slowly move the carriage towards the headstock. As the tool starts to cut into the metal, maintain a steady cranking motion to get a nice even cut. It's difficult to get a smooth and even cut turning by hand.

Continue advancing the tool towards the headstock until it is about 1/4" away from the chuck jaws. Obviously you want to be careful not to let the tool touch the chuck jaws!



Without moving the cross slide or compound, rotate the carriage handwheel clockwise to move the tool back towards the free end of the work. You will notice that the tool removes a small amount of metal on the return pass. Advance the crossslide another .010 and repeat this procedure until you have a good feel for it. Try advancing the cross slide by .020 on one pass. You will feel that it takes more force on the carriage hand wheel when you take a deeper cut.

Turning with Power Feed

One of the great features of the 9x20 is that it has a power leadscrew driven by an adjustable gear train. The leadscrew can be engaged to move the carriage under power for turning and threading operations. Turning with power feed will produce a much smoother and more even finish than is generally achievable by hand feeding. Power feed is also a lot more convenient than hand cranking when you are making multiple passes along a relatively long workpiece.



Make sure the half-nut lever is in the disengaged (up) position. Turn the motor on. The leadscrew should now be rotating counterclockwise. When the leadscrew is engaged the gear train makes kind of an annoying noise, but you'll get used to it. Lubricating the gear train with white lithium grease will cut down some on the noise.

With the tool positioned just beyond the end of the workpiece and advanced to make a cut of .010, engage the half-nut lever. The carriage should move slowly to the left under power from the leadscrew. When the tool gets to within about 1/4" of the chuck, disengage the half-nut to stop the carriage motion.

Now you can use the carriage handwheel to crank the carriage back to the starting point by hand. If you do so without first retracting the cutting tool, you will see that the tool cuts a shallow spiral groove along the workpiece. To avoid this, especially during finishing cuts, note the setting on the cross-slide dial, and then turn the cross feed crank a half turn or so counterclockwise to retract the tool. Now crank the carriage back to the starting point by hand, advance the cross-slide back to the original dial setting plus an additional .010 and repeat the process. You should get a nice, shiny, smooth finish.

Just as in facing, you normally will make one or more relatively deep (.010"-.030") *roughing* cuts followed by one or more shallow (.001"-.002") *finishing* cuts. Of course you have to plan these cuts so that the final finishing cut brings the workpiece to exactly the desired diameter.

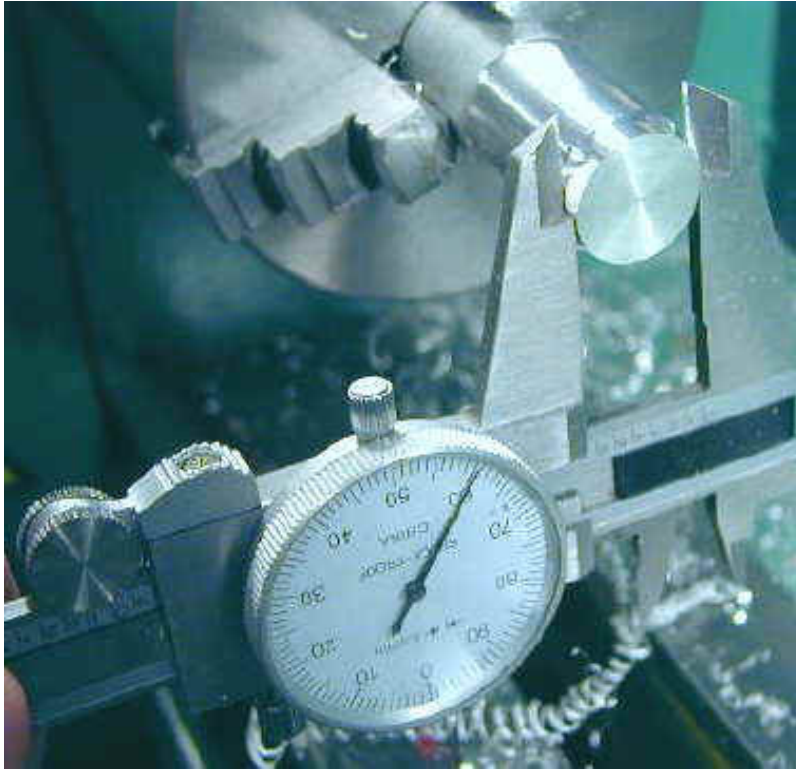


When cutting under power, you must be very careful not to run the tool into the chuck. This seems to happen to everyone at one time or another, but it can shatter the tool and damage the chuck and will probably ruin the workpiece. There is also potential to damage the half-nut, leadscrew or other parts of the power train, so pay close attention and keep your hand ready on the half nut lever.

Measuring the Diameter

Most of time, a turning operation is used to reduce the workpiece to a specified diameter. It is important to recognize that, in a turning operation, each cutting pass removes *twice* the amount of metal indicated by the cross slide feed divisions. This is because you are reducing the radius of the workpiece by the indicated amount, which reduces the diameter by twice that amount. Therefore, when advancing the cross slide by .010", the diameter is reduced by .020".

The diameter of the workpiece is determined by a caliper or micrometer. Micrometers are more accurate, but less versatile. You will need a machinist's caliper capable of measuring down to .001". Vernier calipers do not have a dial and require you to interpolate on an engraved scale. I prefer a dial caliper which gives a direct easy-to-read and hard-to-misinterpret measurement. Fortunately, good quality Chinese 6" dial calipers are now available for under \$20 from suppliers such as Enco or J&L.



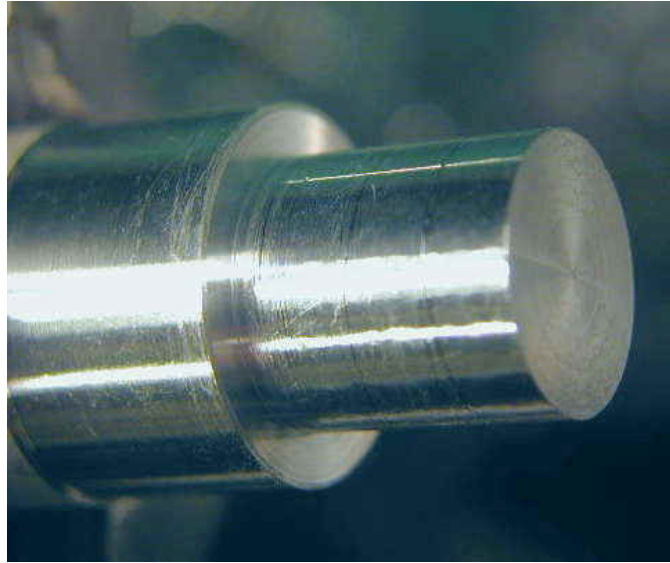
It should be self-evident that you should *never* attempt to measure the work while it is in motion. With the lathe stopped, bring the dial caliper up to the end and use the roller knob to close the caliper jaws down on the workpiece. I try to use the tips of the caliper since they are thinner. Gripping the work in the thicker portion of the caliper jaws can force the jaws apart a few thou if you twist the caliper even a small amount.

I like to take an initial reading of the dial while it is still gripping the work since it is easy to inadvertently twist the caliper when removing it, thus changing the reading. You can use the locking screw on the caliper to help prevent this. Slide the jaws straight off the workpiece being careful not to twist the caliper.

It's a good idea to take at least two separate measurements just to make sure you got it right. As it turns out (no pun intended) it's much easier to *remove* metal than it is to put it back ;-)

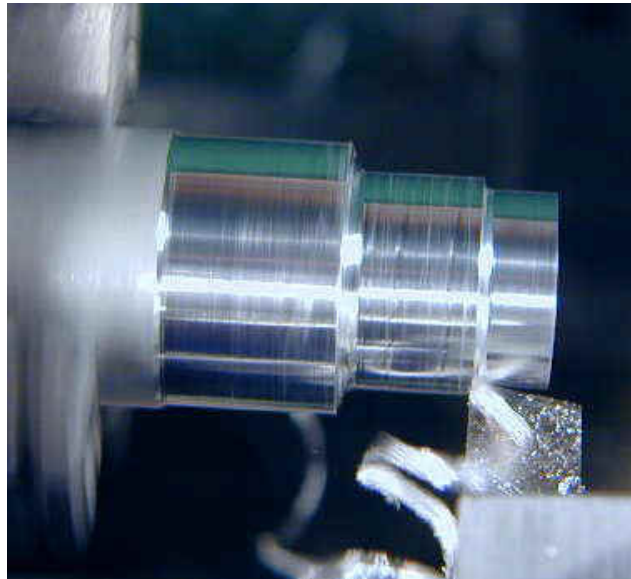
Turning a Shoulder

A shoulder is a point at which the diameter of the workpiece changes with no taper from one diameter to the other. In other words, there is a 90 degree face moving from one diameter to the other as you can see in the next photograph.



We will make a shoulder on our workpiece by reducing the diameter of the end of the workpiece for a distance of about 1/2".

Advance the cross slide about .020" and use power feed to turn down about a 1/2" length on the end of the workpiece. Repeat this a few more times until you have reduced the diameter of the end section to about 1/2".



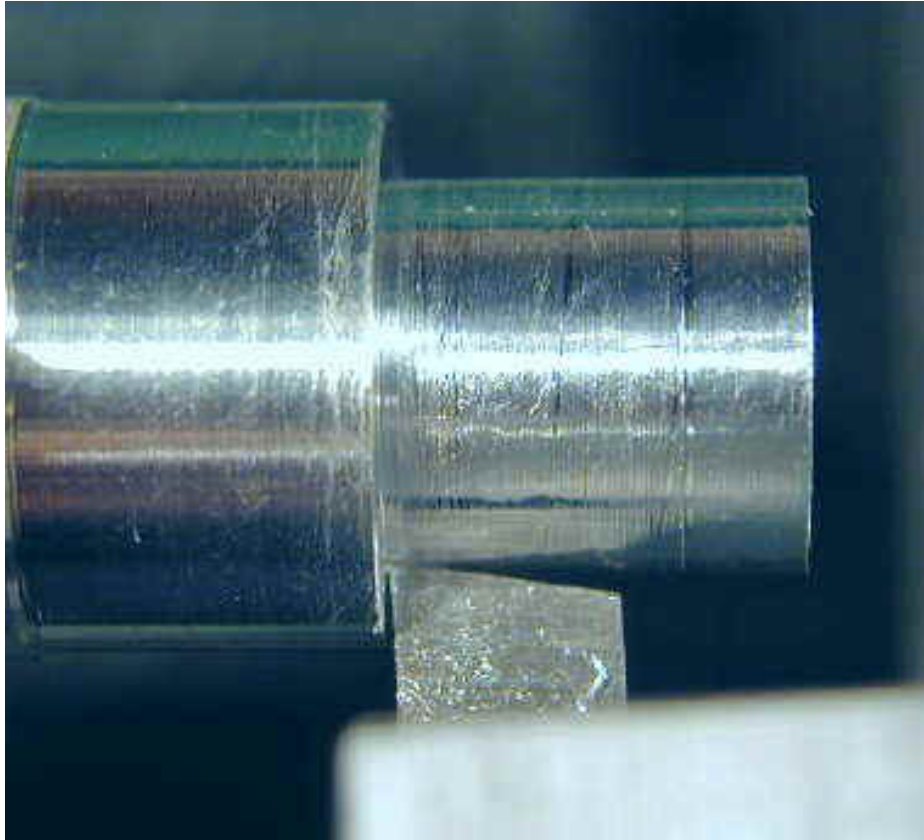
Since the tip of the tool is rounded, the inner edge of the shoulder takes on a rounded profile.



To get a nice square edge we must switch to a tool with a sharp point ground to an angle of less than 90 degrees so that it can work right down into the corner of the shoulder.



Now we will use this pointed tool to make a square finishing cut into the corner of the shoulder. Since this is such a short distance, we will use hand feed, not power feed. You can use hand feed with the leadscrew turning - just don't engage the half-nut or feed lever.



To get a nice square face on the shoulder you will need to make a facing cut. This works best if you have the carriage locked on your lathe. Lock the carriage and clean up the face of the shoulder until it is square. If you use the sharp pointed tool you will need to use fairly high RPM, say 1500, and advance the tool slowly or you will get little grooves from the pointed tip instead of a nice smooth finish.

If you haven't made yourself a carriage lock you will need to use the half-nut to lock the carriage in place for the facing cut. Of course you must first disengage the lead screw before you do this!

Finally, you may want to use a file as described in the facing section to make a nice beveled edge on outside edge of the shoulder and on the end of the workpiece.

Lathe Operations: Parting

Parting Operations

Parting uses a blade-like cutting tool plunged directly into the workpiece to cut off the workpiece at a specific length. It is normally used to remove the finished end of a workpiece from the bar stock that is clamped in the chuck. Other uses include things such as cutting the head off a bolt.

Commercial Parting Tools

There is a wide variety of commercial parting tools available from tool suppliers.



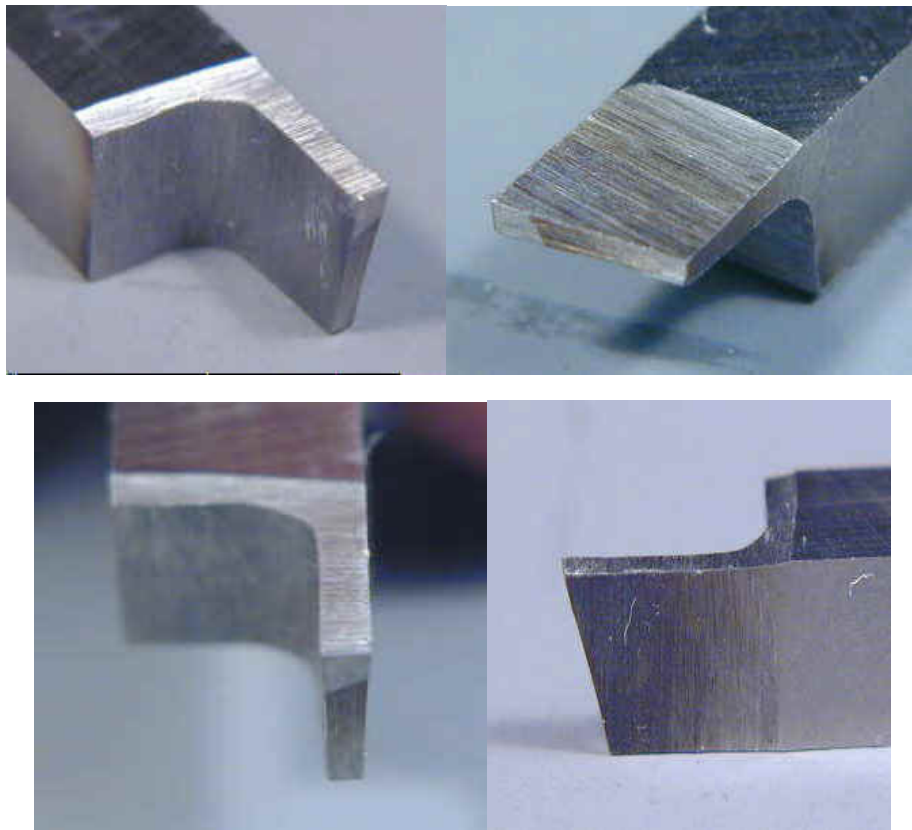
It is important for the top of a parting tool to be right on center.

Here's another 1/2" commercial parting tool. I plan to make a custom toolholder for it someday.



Custom Ground Parting Tools

Grinding your own parting tool is not real difficult but it takes a long time and generates a lot of metal and grinder dust due to the relatively large amount of metal that you must remove from the blank. Here are some pictures of a typical home-ground tool. Note that the tool is tapered from top to bottom (like a narrow keystone) and from front to back to provide relief for the cutting tip. The top of the tool has been ground down by a few thousandths of an inch to align the top edge of the tool with the lathe centerline. If you have a toolholder with adjustable tool height, this would not be necessary. Forming the parting blade near the edge of the tool allows the tool to work up close to the chuck jaws.

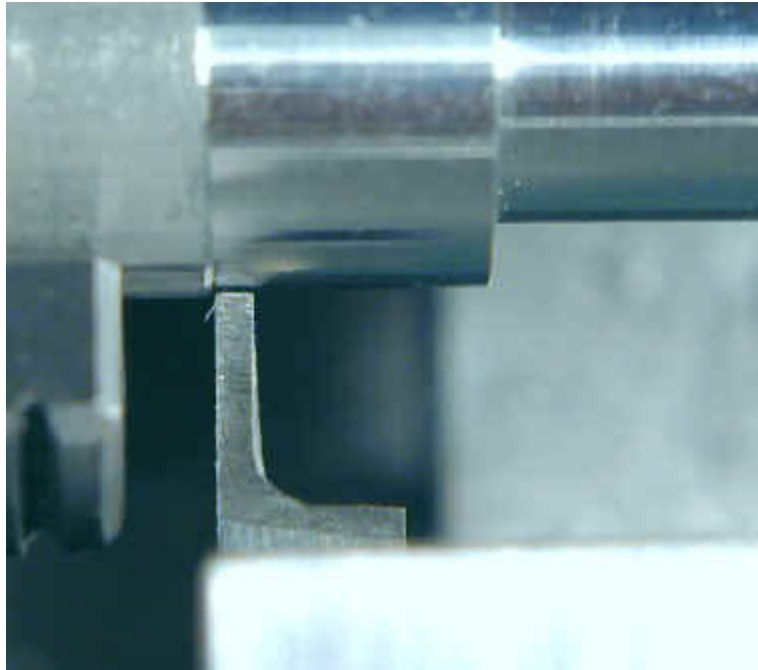


Chucking the Workpiece

Parting is *always* done close to the chuck jaws - no more than 1/2" out, and, preferably, no more than 1/4" out. (Note: this is also relative to the diameter of the workpiece; 1/4" may be right for a 3/4" diameter workpiece, but would be too far out for a 1/8" dia. piece.) Parting cuts impose great tangential force on the workpiece that could cause the workpiece to be forced out of the chuck if you cut too far from the chuck jaws.

Adjusting the Tool Bit

For a parting cut the top of the tool should be exactly on the center line of the lathe, or no more than .005" above the center line. If the tool is a little high it will have a tendency to 'climb' the work; a little low will cause a tendency to dig in. The tip of the tool should be exactly perpendicular to the workpiece.



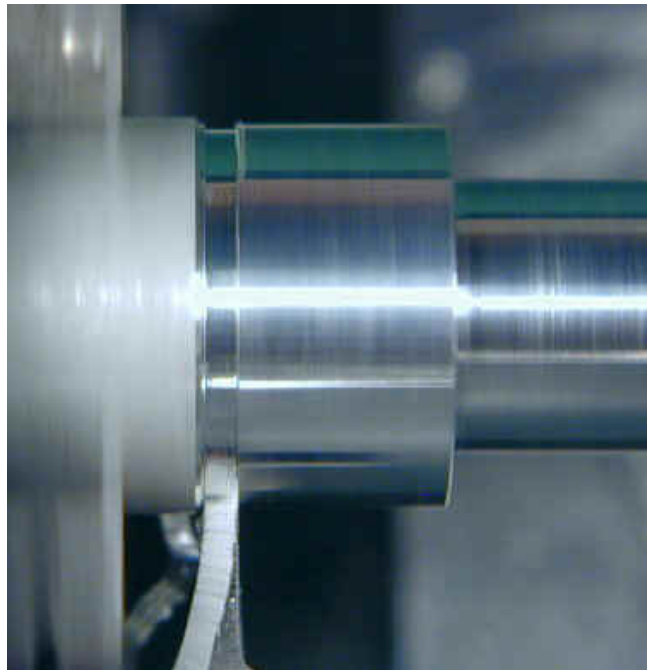
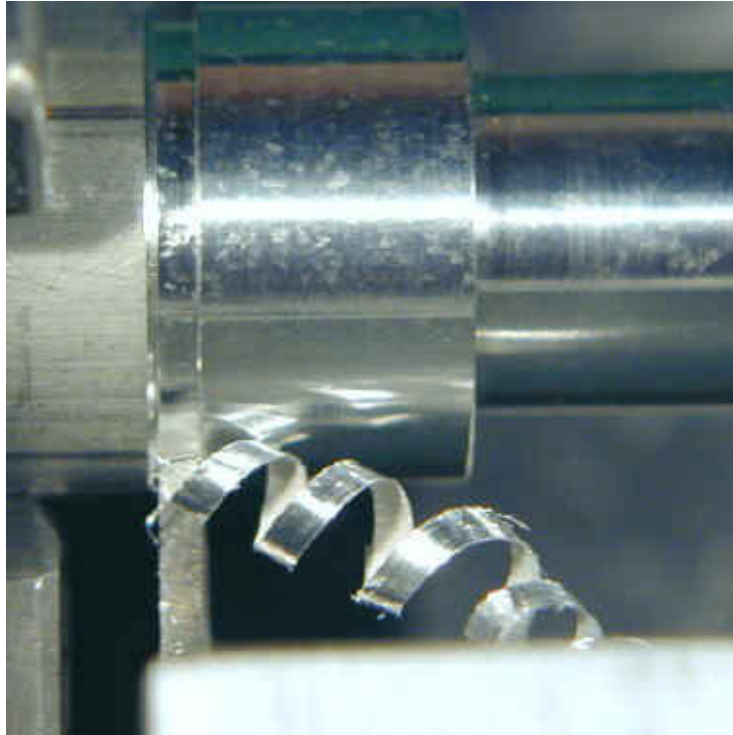
Speed and Feed

Make sure the leadscrew is in the neutral position so that the leadscrew is not moving. Now lock the carriage to keep the carriage from moving during the parting cut.

Parting cuts should be made at low speed; say 200-300 RPM or even slower.

Making the Cut

With the tip of the tool just beyond the surface of the workpiece, turn on the lathe. Slowly advance the cross-slide crank until the tool starts cutting into the metal. Keep advancing the tool until you get a steady chip curling off the workpiece and then try to maintain this cutting speed.



It's a good idea to use cutting oil for a parting cut and you will find that the heat generated will most likely cause a fair amount of smoke as the cutting oil burns off. Avoid breathing this smoke - I haven't heard of any ill effects, but I'm sure it's not good for your lungs. A small fan to disperse it may be a good addition to your workbench.

Chatter

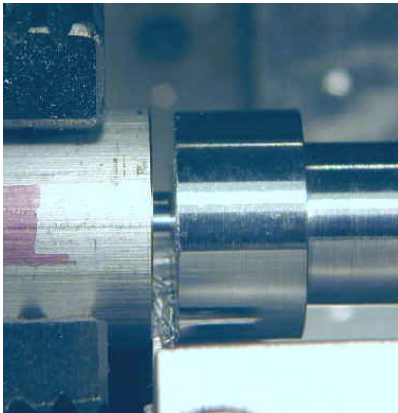
Parting often causes 'chatter'. If you have never heard this sound, you will easily recognize it when you first do. It is a pulsing, whining vibration that can shake the whole lathe and even cause it to move around on the workbench if it is not bolted down. You can stop chatter quickly by backing off the pressure on the tool. The trick is to find the right speed at which to advance the tool with minimal chatter.

Here are some tips to minimize chatter:

- Tool tip should be quite sharp
- Top of tool should be right on the lathe centerline
- Tool should be perpendicular to the workpiece
- Gibs on cross-slide and compound should be snug
- Saddle should be snug to the ways
- Use carriage lock to lock saddle to ways
- Use cutting fluid
- Maintain steady advance of cross-slide

Finishing the Parting Cut

Keep advancing the tool until it reaches the center of the workpiece. As you get close, the workpiece is suspended by a thin stalk of metal.



Be careful: if the workpiece extends from the chuck more than a few times its diameter, the end of the workpiece can start to swing in a dangerous arc. As you get near the center, you may need to slow down the chuck speed to keep things safe. If you notice the workpiece starting to wobble, stop the lathe and move the workpiece back and forth by hand to break it free.

The end of the workpiece that you cut off will generally have a pretty rough finish and a little stalk of metal protruding from the end.



One limitation of parting tools is the diameter of the work that can be parted. The tool illustrated here is a little less than 3/8" long and can part off work up to 3/4" in diameter. In the previous picture you can see that the edge of the work is rounded because it was rubbing up against the shoulder of the cutting tool. If you make the tip of the tool much longer than about 1/2" it starts to get too limber and will easily break off. So on a small lathe like this, the largest diameter work that you can part off is probably around 1". To cut off bigger work, you can use a small hacksaw while turning the work at low speed in the lathe. Even better, if you have a metal-cutting band saw use it to cut off the work. I nearly always use the band saw for work larger than 1/2" diameter.

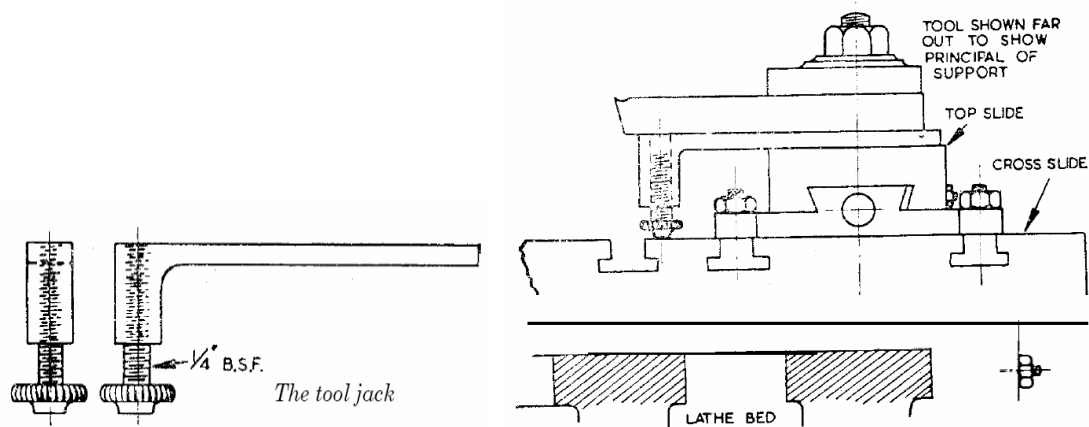
The final step is to mount this piece in the chuck and make a facing cut to clean up the end. One problem with this step is that the chuck jaws can mar the finished workpiece. If you look carefully at the next picture you can actually see the imprint of the chuck jaws. To avoid this, you could wrap the workpiece in a thin strip of emery paper, or similar protective material, before clamping it.



The Parting tool Support Jack- a post by Tim

Without a doubt, every turner has a certain amount of trouble parting-off metal in the chuck. We amateurs seem to suffer most, principally on account of the size of machine we use; but even the professional turner with the best of equipment still seems to have a fair amount of bother. There have been many tool holders and gadgets described in various forms which give added rigidity to the tool, but all I have seen have never bothered to get over, to my mind, the principal cause of the trouble ; that is deflection of the top-slide when used in its normal position. We know, of course, that it is preferable to turn the top-slide parallel with the chuck, but with many small lathes this is tedious to perform owing to having to shift the clamp-bolts. Also, it is advised to use a rear toolpost, turning the parting tool upside down. This is all very well if we have one, but on many jobs they are apt to get in the way. Having a fair amount of parting-off to do, I decided that something would have to be done about this chatter, so I made the little tool illustrated, which has completely banished all my troubles. I use ordinary " Eclipse" or "Darwin Toledo" parting-off blade type cutters, just clamped under the toolpost with my tool jack placed underneath, which is so made as to give correct height to the tool. I have had no bother with the tools tipping sideways and for speed of changing, find it perfectly satisfactory. Of course, to take the matter to its logical conclusion, one could make a holder on the Armstrong or Jones and Shipman principal, and fit it with the elevating screw to give the support under the tip of the tool, which is the main object of the whole principle. It still not as good as the method described, for the simple reason that the tool has to project from the holder and would not be supported under its tip. I now use this support for nearly all screw cutting, especially square and Acme threads, and find that it gives excellent results. When turning cranks, it really comes into its own; using slow speed, there is very little fear of a dig in. Its main points are ease of adjustment, and, secondly, which is the most important of all, complete rigidity of the tool, taking the load immediately under the cutting point. One presumes, of course, that the tool is correctly ground, and that the lathe bearings are correctly adjusted; if not, well, one cannot expect to turn true. I have a shrewd suspicion, however, that this jack would help a lot by eliminating one of the sources of trouble, i.e., tool deflection. I am not giving any dimensions, as they are fixed by the size and make of the lathe, but anyone can see how it works and how best to adapt it for their own use. My advice to all who use lathes is to try it; I am sure that they will find it most useful and well worth the small effort needed to make it.

TIM.



Lathe Operations:

Thread Cutting

Introduction

A versatile feature of the mini-lathes not found on many other low-cost lathes is the ability to cut threads. Thread cutting takes a fair amount of skill, so is best left until you have mastered the basic lathe operations such as facing, turning and drilling. This document assumes that you are familiar with these basic operations and with lathe components and terminology.

Threading will open up a whole new set of possibilities for your projects. In addition to the standard threads that can be cut with a die, on the lathe you can cut matching internal and external threads on any size item up to 3 or 4 inches in diameter and in materials including aluminum, brass, steel and plastics. Here are some examples of some threaded items made on the mini-lathe:



In essence, thread cutting consists of making multiple passes along the length of cylindrical workpiece using a cutting tool accurately shaped to a 60° point. On each successive pass, the tool is advanced into the workpiece by a few thousandths of an inch. The cutting tool is driven under power by the leadscrew which is coupled to the spindle by a train of gears. The gear ratio, together with the pitch of the leadscrew, determines the pitch of thread that is cut.

While this sounds pretty simple, in practice it is a little more difficult, and beginners often have a frustrating time trying to cut their first threads. Hopefully, the information presented here will help you to quickly master this useful skill.

Thread Properties and Standards

There are many different standards for defining the shape and properties of threads. We will focus here on the Unified National Thread Form (UNTF), commonly known as "inch" or SAE (Society of Automotive Engineers) threads. These will accommodate the great majority of your needs, as they are the standard thread types for joining components together. Other thread types, such as ACME threads, are used in more specialized applications such as vise screws and lead screws.

UNTF threads use a numbering system for the smaller sizes and an inch system for the larger sizes. As an amateur machinist you are no doubt already familiar with these common screw threads:

2-56, 4-40, 6-32, 8-32, 10-32, 1/4-20, 3/8-18, 1/2-16

The first number before the hyphen designates the nominal outside diameter of the screw and the number following the hyphen designates the number of threads per inch, or TPI. Sometimes this number is referred to as the pitch of the thread, but correctly speaking, pitch is the reciprocal of the threads per inch. For example, a 20 TPI thread has a pitch of 1/20 or .05. Although all of the 'number' size threads are even-numbered, there are, in fact, odd numbered sizes - they are just much less common since an assortment of the even-numbered sizes is adequate for most practical applications.

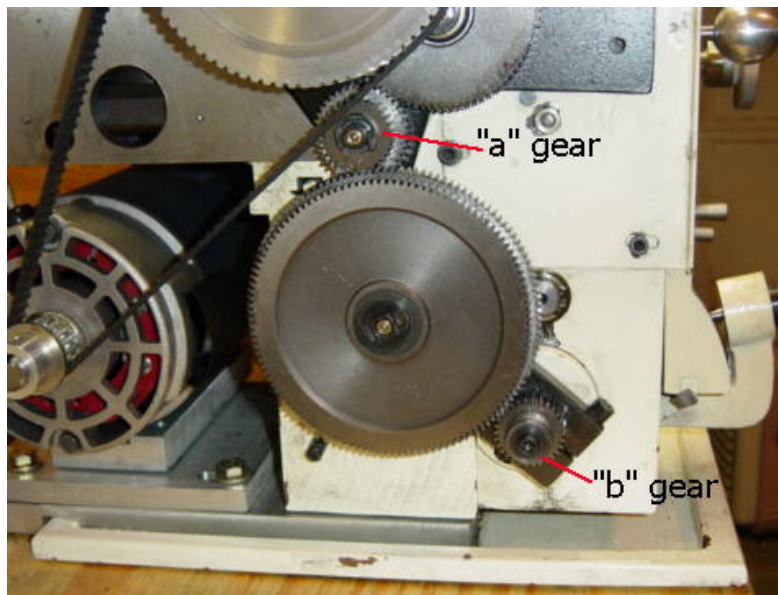
The UNTF system includes two major subsets of threads: Unified National Coarse (UNC) and Unified National Fine (UNF). Under the earlier but very similar American National Thread Form, these were designated as NC and NF, so you may see those designations in older texts. For a given diameter, the UNC thread has fewer threads per inch than the UNF thread. The geometry of the thread shape dictates that finer threads will also be shallower than coarse threads. All of the threads in the above list of common screw sizes are a UNC thread, with the exception of 10-32 which is UNF.

Since a screw is a form of lever, the finer pitch of the UNF screw imposes more force when it is tightened, resulting in a tighter fit. Nuts and bolts also come in a variety of strengths (the strength and hardness of the metal from which they are made) and tolerance ratings (how precisely the thread matches the nominal dimensions), but we will not address those factors here.

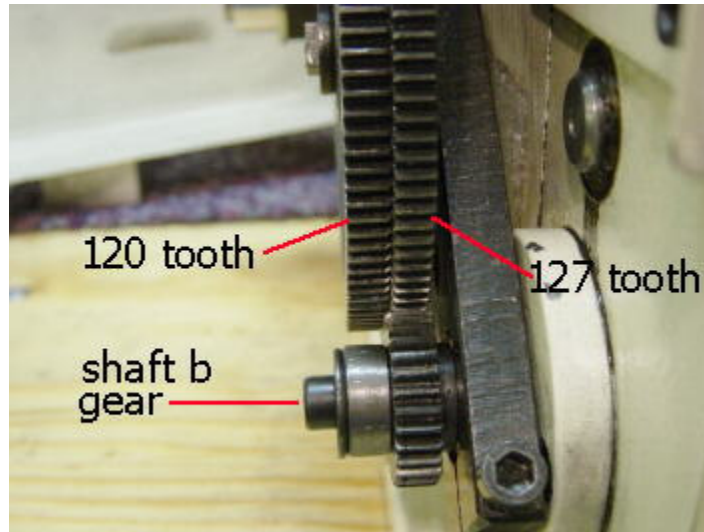
Here's a more complete table:

Size	Outside Diameter	UNC TPI	UNF TPI
2	.086	56	64
4	.112	40	48
6	.138	32	40
8	.164	32	36
10	.190	24	32
1/4	.250	20	28
3/8	.375	16	24
1/2	.500	13	20

Threading Example
9x20 Lathe with quick change gearbox



This picture shows the location of the a & b gear locations. This lathe looks a little different because of the tumble reverse and the DC motor but the gear locations are the same. On the "a" gear shaft there are two gears. The gear to the back is a 40 tooth gear and it remains in the back position at all times. In this pic the "a" position has a 30 tooth gear installed. The "b" gear shown in the above pic is a 30 tooth gear.



Pay close attention to the "b" gear position. The pic above shows the "b" gear meshing with the 127 tooth gear. The 127 tooth gear is used for inch threads. If you are going to cut metric threads the "b" gear has to mesh with the 120 tooth outer gear. The spacer shown above on the outside of the "b" gear would be moved to the inside of the gear for metric threads.

After the correct gears have been installed you will need to select the correct setting on the quick change gearbox. On the threading chart on the lathe this is referred to as the Lever selection. For a 16 tpi thread the lever position would be hole # 1.

Lead Screw Operation

Thread cutting lathes always have the following components:

- **Lead screw**
- **Half nut lever**
- **Change gears**

On the 9x20 lathes, the lead screw is used both to feed the carriage for turning operations as well as for thread cutting operations. The expensive lathes also provide a Quick Change Gear Box to quickly select the desired thread pitch by engaging a pair of levers. On the 9x20 lathes you must manually install the correct set of gears for the desired thread pitch.

The change gears couple the lead screw rotation to the spindle rotation in a fixed ratio. For example, if the leadscrew and spindle are coupled with two gears which each have the same number of teeth, it is apparent that the leadscrew will make one rotation for each rotation of the spindle. If the gear on the spindle has twice as many teeth as the gear on the leadscrew, then the leadscrew will make two rotations for each rotation of the spindle. Other combinations of three and four gears make it possible to adjust this ratio to obtain a wide range of thread pitches.

Ultimately, the pitch of the thread being cut is determined by the pitch of the leadscrew itself. On the 9x20 lathes the leadscrew is a true 16 threads per inch (TPI). This may seem a little surprising when you consider that all of the other dimensions of the lathe, including the cross-feed and compound lead screws, are metric pitch. However, it works to the advantage of those owners who need to cut primarily SAE rather than metric threads.

When the half nut engages the leadscrew, the carriage is driven along by the leadscrew. The thread cutting tool moves along the surface of the work at this same speed. Its pretty easy to see that if the change gear ratio is 1:1, then the thread pitch that is cut will be the same as that of the leadscrew: 16 TPI. If the ratio is 2:1 then the cutting tool will move twice as fast relative to the spindle rotation and will cut an 8 TPI thread as faster motion of the cutting tool results in a coarser thread.

The Thread Dial Indicator

Attached to the right side of the apron is a threading dial or thread indicator. It is pretty simple, comprising a pinion gear that engages the leadscrew thread and a dial driven directly by the pinion. If you move the carriage by means of the carriage handwheel, you will see that the thread indicator dial rotates as the carriage moves relative to the leadscrew. (Note: I sometimes remove the thread indicator to gain an extra inch of carriage movement.)



If the leadscrew is turning with the half nut open, the carriage does not move, but the threading dial rotates because it is engaged with the rotating lead screw. If you now close the half nut on the rotating lead screw, you will see that the thread indicator stops turning. The thread indicator shows only the relative movement between the carriage and lead screw, so it does not move when the carriage is being moved by the leadscrew.

Since threads are cut in multiple passes, the function of the thread indicator is to ensure that the carriage engages with the leadscrew at the same relative position for each pass, so that the threads on each pass are cut in the same groove. Without the help of the thread indicator, it would be difficult or impossible to engage the half nut at the right point. (There is, however, another way to accomplish this, as will be explained later.)

The markings on the dial of the thread indicator are reference points for various thread pitches. Depending on the pitch being cut, and its relationship to the 16TPI thread of the leadscrew, the half nut can be engaged at different marks on the thread indicator. Posted on the front of the change gear cover is a small chart that shows which marks may be used for various thread pitches.

INDICATOR TABLE			
16T			
T.P.I.	SCALE	T.P.I.	SCALE
12	1.3.5.7	26	1.5
13	1	28	1.3.5.7
14	1.5	32	1~8
16	1~8	36	1.3.5.7
18	1.5	38	1.5
19	1	40	1~8
20	1.3.5.7	44	1.3.5.7
22	1.5	48	1~8
24	1~8	52	1.3.5.7

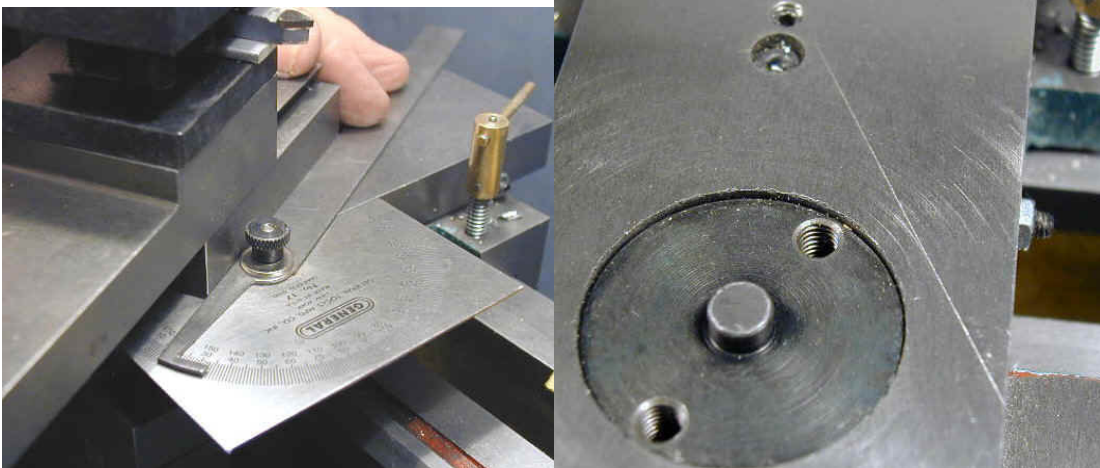
As you can see, for each thread pitch in the 'T.P.I.' column, there are one or more numbers in the 'SCALE' column. For example, in the SCALE column for 12 TPI you see 1.3.5.7. The meaning of this is that you can engage the half-nut when any of the numbers 1, 3, 5 or 7 is aligned with the index mark on the thread indicator as it is rotating. For 13 TPI, you can see that the half-nut can be engaged only when the number 1 passes the thread indicator index mark. For 24 TPI, the notation 1~8 means that the half-nut may be engaged when any of the numbers 1-8 is aligned with the index mark.

Setting the Compound Slide

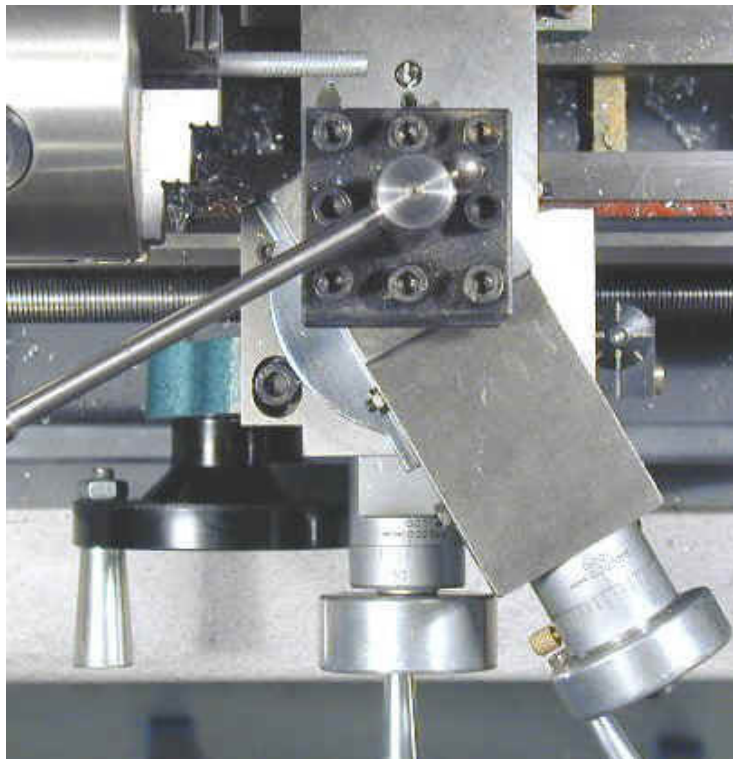
Confusion often arises in setting the compound to the proper angle for thread cutting. This is because some books specify the angle as 30°, and the lathe owner therefore sets the compound the 30° as measured by the compound protractor. This is incorrect, as the protractor measures the angle relative to the lathe axis or centerline, while the 30° angle specified in the textbooks is relative to a line tangent to the centerline, or 60° angle relative to the lathe axis.

Since the correct angle is "off the scale" of the compound protractor, you will need a separate machinist's protractor, or other suitable tool, to

properly set the compound. Once you have set the proper angle, you can scribe a line along the edge of the compound onto the top of the cross slide and use this line to set the compound in the future.



Here's a top view of the compound, showing the correct angle. Note that the tip of the cutting tool is perpendicular with the side of the workpiece.



Some textbooks and old timers specify the compound angle as 29.5° rather than 30° . The reason for this, I believe, is to cause all of the cutting to occur on the leading edge of the cutting tool, resulting in a slightly cleaner cut. Either angle will work, or you can experiment with both and determine which setting you prefer.

Thread Cutting Tools

Thread cutting tools may be purchased commercially or you can grind your own from standard high speed steel blanks. Since the tip of the tool must form a pretty accurate 60° angle grinding your own tools takes a fair amount of skill unless you use some sort of jig to hold the tool at the proper angle to the face of the grinding wheel. On the other hand, once you learn to grind your own tools, you can grind them to some special shapes that let you cut threads closer to the edge of a shoulder.

Shown below are several thread cutting tools:



Left to right they are as follows:

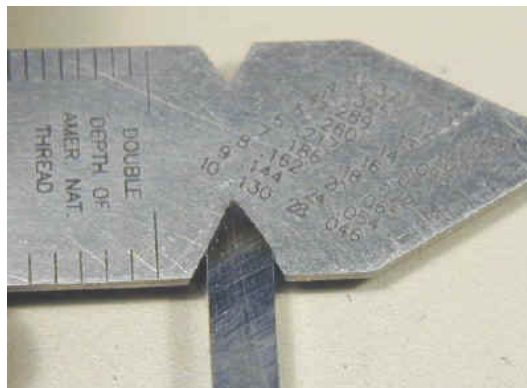
- commercial cemented carbide tool with a 1/4" shank
- hand ground from a 5/16" HSS blank
- hand ground, offset angle, 3/16" blank
- hand ground and offset to left edge, 3/16" blank
- hand ground, internal threading tool, 5/16" blank

The offset tools are designed to be able to cut threads up close to the edge of a shoulder. Since a shoulder is often used as a stop or aligning face for a threaded workpiece, this is a handy capability.

To grind your own threading tool, hold the blank at an angle to the face of the grinding wheel and with the blank tilted downwards from the tip to the end. The top surface of the blank should also be a tilted a little bit downwards so that the tip is undercut, providing relief - much like the bow of a ship. It is not necessary to provide any top rake, so the top surface of the blank is left in its original form. A small India stone can be used to clean up the edges to provide a cleaner cutting surface.



As you grind the tip, check the angle using the notch in the thread gage. The tip of the tool should fit pretty cleanly, with little or no light showing between the edges of the tool and the sides of the notch.



Preparing the Workpiece

I recommend 1/4" diameter 6061 aluminum rod for practicing thread cutting. It cuts easily, is inexpensive and easy to obtain. You can purchase a 6-foot length from Enco for \$2.74 (P/N 505-3644) at the time of this writing, so \$10 worth will give you plenty of material to practice on.

Chuck a length of 1/4" 6061 aluminum stock in the 3-jaw chuck with about 1" extending from the chuck jaws - any more than this and the work will flex away from the cutting tool and result in an incorrectly formed thread. Now, with the lathe turning about 1500 RPM, use a rounded turning tool and advance the carriage slowly by hand to turn the workpiece down to

about .245" diameter and put a smooth finish on the workpiece. This will ensure that the workpiece is not oversize and is concentric. Although the nominal outside diameter for a 1/4" thread is .250, the actual diameter for commercial bolts is typically around .244 to .246.

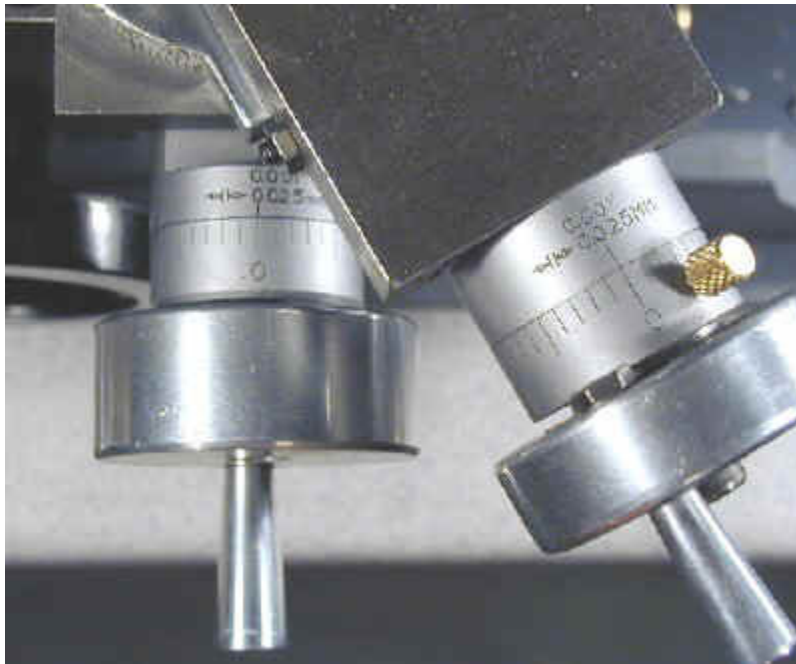
Aligning the Cutting Tool

Insert your thread cutting tool in the toolpost and position it about 1" away from the surface of the workpiece. Next you want to adjust the angle of the tool so that it is exactly perpendicular to the side of the work - an imaginary line perpendicular to the side of the work should bisect the 60° angle of the tool. A handy aid for lining up the tool is a thread gage. It has two different sized 60° notches cut in its sides. Position the thread gage so that one side is lined up with the side of the workpiece. Then adjust the angle of toolpost so that the tip of the tool aligns exactly in the notch. With some practice you may find that you can line the tool up without using the thread gage, and this is sometimes necessary if the workpiece is short and there is not enough room to use the gage.



Zeroing the Cutting Tool

The next step is to zero the tip of the cutting tool to the side of the workpiece. Set aside the thread gage and advance the tool using the cross-slide handle until the tip of the tool is just about touching the side of the workpiece - but stop with the handle in the 'down' position. This 'bottomed out' position of the handle will serve as a convenient reference point during the thread cutting in which you will advance and retract the tool quite a few times. Now set the dial on the cross-slide to zero. Next, advance the tool using the compound handle until the tip of the tool just touches the side of the work; then set the compound dial to zero.



It is very important that the cross-feed and compound dials do not slip or get moved during the thread cutting process - they are your reference point for the thread depth, which must be accurately determined. I found it necessary to add a locking screw to the compound dial on my lathe to prevent it from slipping.

Back off the cross-slide by one full turn, stopping when the handle reaches to bottom point. This moves the tool back about .040" from the workpiece. Now crank the carriage towards the tailstock until the tool is about 1" beyond the end of the workpiece. Advance the cross-feed one full turn, being careful to stop at the zero setting on the dial. You will be repeating this process many times during the cutting operation and it is important to stop exactly on the zero mark. If you overshoot, you must crank the cross-feed out one full turn and then advance it a full turn to the zero mark. This procedure ensures that backlash in the lead screw has been removed.

Making a Test Cut

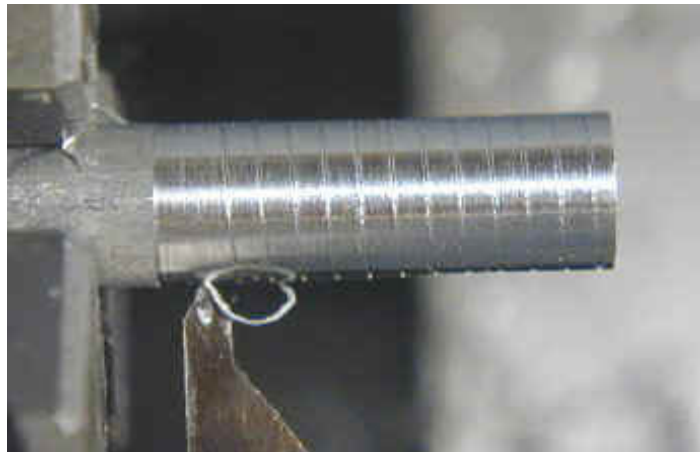
Safety warning:

Keep in mind that when cutting threads, the carriage moves toward the headstock very rapidly as compared to a turning operation - the coarser the thread pitch, the faster the motion. You must watch carefully and be prepared to disengage the half-nut quickly to avoid crashing the tool into the chuck.

- Use the slowest speed available on your lathe
- Keep your full attention on the lathe (no kids or other distractions in the shop)
- Keep your right hand on the half-nut lever.

To get a feeling for how fast the carriage will move, crank the carriage well away from the headstock and engage the half-nut lever. Keep your hand on the half-nut lever so that you can quickly disengage it as the carriage approaches the headstock.

At this point, we want to make a very shallow test cut to make sure that we are actually cutting the intended pitch. Turn on the lathe and set the speed as low as it will go without stalling during the cut. Make sure the leadscrew is turning and observe that the threading dial is rotating. Engage the half-nut lever just a hair before the threading dial passes one of the numbers: 1, 3, 5 or 7. Keep your hand on the half-nut lever and disengage it quickly as the tool gets close to the chuck. This operation should produce a very shallow spiral groove along the surface of the work.





Another handy tool (tools: gotta have 'em!) to have on hand at this point is a screw pitch gage. Available from the usual suppliers for around \$5-15, this tool has multiple blades which rotate out from a housing, with each blade having a template for a specific TPI thread.

Use the 20 TPI gage to verify that the teeth of the gage line up with the grooves of the spiral. If they do not, double check to make sure you have set up the gear train properly.



Repeat the procedure for retracting the tool: Back off the cross-slide by one full turn, stopping when the handle reaches to bottom point. This moves the tool back about .040" from the workpiece. Now crank the carriage towards the tailstock until the tool is about 1/2" beyond the end of the workpiece. Advance the cross-feed one full turn, being careful to stop at the zero setting on the dial. You will be repeating this process many times during the cutting operation and it is important to stop exactly on the zero mark. If you overshoot, you must crank the cross-feed out one full turn and then advance it a full turn to the zero mark to overcome the backlash in the lead screw nut.

Determining the Thread Depth

Because the compound is set at an angle to the lathe axis, when you advance the cutting tool using the compound dial the tool advances towards the work less than the amount indicated by the dial. The 60° angle of the compound relative to the lathe axis or centerline dictates that the tool is advanced by $\sin(60)$ or .866 times the dial reading. Thus, if the dial is advanced by .010", the tool only advances by about .0087". To simplify your life, I have compiled the following table which shows the thread depth as measured by directly reading the compound dial. For example, for a 1/4-20 thread, you can see in the column labeled "Dial Depth" that the compound dial will read 17.6 thousandths (.0176) when the thread has been cut to the proper depth. (To print the table, highlight it, then copy and paste it to Excel)

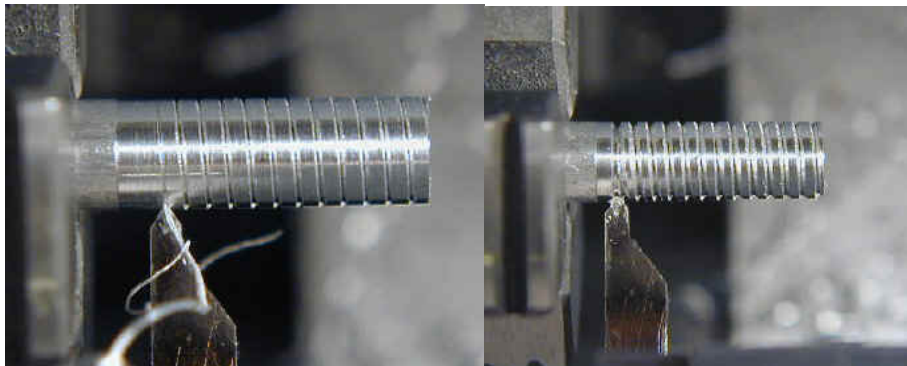
Size	TPI	UNTF	Major Dia.	Minor Dia.	Depth	Dial Depth
2	56	UNC	0.0860	0.0641	0.0110	0.0063
2	64	UNF	0.0860	0.0668	0.0096	0.0055
4	40	UNC	0.1120	0.0813	0.0154	0.0088
4	48	UNF	0.1120	0.0864	0.0128	0.0074
6	32	UNC	0.1380	0.0997	0.0192	0.0110
6	40	UNF	0.1380	0.1073	0.0154	0.0088
8	32	UNC	0.1640	0.1257	0.0192	0.0110
8	36	UNF	0.1640	0.1299	0.0171	0.0098
10	24	UNC	0.1900	0.1389	0.0256	0.0147
10	32	UNF	0.1900	0.1517	0.0192	0.0110
12	24	UNC	0.2160	0.1649	0.0256	0.0147
12	28	UNF	0.2160	0.1722	0.0219	0.0126
1/4	20	UNC	0.2500	0.1887	0.0307	0.0176
1/4	28	UNF	0.2500	0.2062	0.0219	0.0126
5/16	18	UNC	0.3125	0.2443	0.0341	0.0196
5/16	24	UNF	0.3125	0.2614	0.0256	0.0147
3/8	16	UNC	0.3750	0.2983	0.0384	0.0220
3/8	24	UNF	0.3750	0.3239	0.0256	0.0147
7/16	14	UNC	0.4375	0.3499	0.0438	0.0252
7/16	20	UNF	0.4375	0.3762	0.0307	0.0176
1/2	13	UNC	0.5000	0.4056	0.0472	0.0271
1/2	20	UNF	0.5000	0.4387	0.0307	0.0176

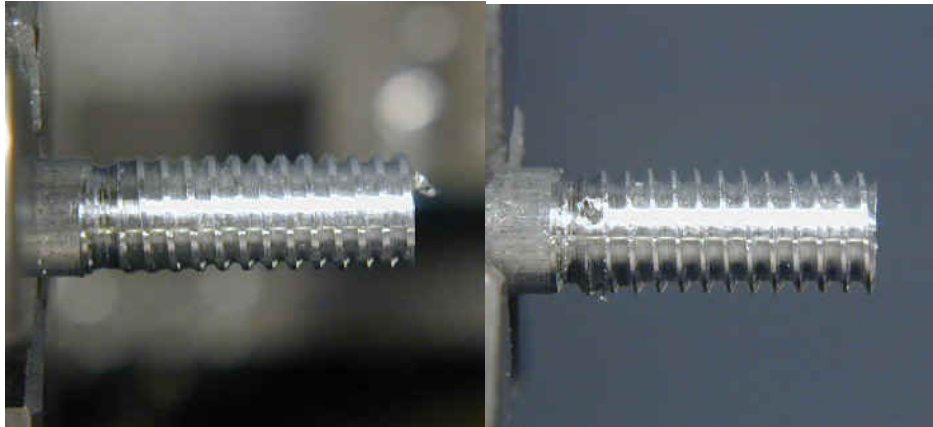
In the terminology of screw threads, the outside or maximum diameter of the thread is called the Major Diameter and the minimum diameter across the base of the screw thread is called the Minor Diameter. The distance from the top of the V to the bottom of the V is the thread depth. The Major Diameter minus the Minor Diameter is twice the thread depth. If you examine the table you will observe that threads of the same TPI have the same depth.

Cutting the Thread

During the thread cutting operation, the cross-feed is used only to withdraw the tool from the work so that the carriage may be cranked back towards the tailstock without the tool hitting the work. The cutting depth of the tool is controlled entirely by the compound feed. With the tool now at the original zero point, advance the cross feed by .006". This will set the depth for the first cut. On successive cuts we will reduce the cutting depth on each pass since the tool will be removing more metal.

On the first few passes I usually set the depth at about .006", then decreasing to .005, .004, .003, .002 and so forth on subsequent passes on down to .001 or .0005 on the final passes. For coarser threads, such as 10 TPI, you can take somewhat deeper cuts in the early passes. With a little practice you will get a feel for how aggressive a cut you can take on each pass. When the desired depth is reached, it is a good idea to make a few extra passes without advancing the tool to clean up any roughness in the thread. Here are a few pictures showing the thread at various stages of the process:

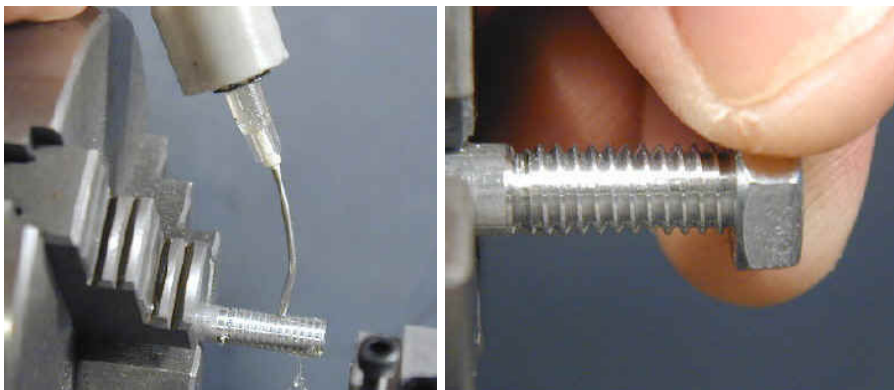




Cutting the thread to the proper depth is merely a repetition of the steps we have already performed. Start with the cutting tool positioned near the end of the workpiece, then follow these steps:

- **Set the cross-feed to the original zero position**
- **Advance the compound feed by a few thousandths - smaller amounts on each pass**
- **Engage the half-nut lever when your number comes up**
- **Disengage the half-nut lever when you reach the end of the thread**
- **Back off the cross-feed one full turn**
- **Crank the carriage back to the starting point**
- **Repeat above steps until depth has been cut to specified Dial Depth from the chart**

A few drops of cutting fluid will help to keep the chips flowing and provide a better finish. I like Tap Magic Aluminum when I'm working with aluminum.



Test the thread with a standard commercial nut of a matching size. You may need to clean up the end of the thread with a file to get the nut to start onto the thread. As a final step, you may wish to make a few additional passes without advancing the compound. This will generally remove a small amount of additional metal, resulting in a finer finished thread.

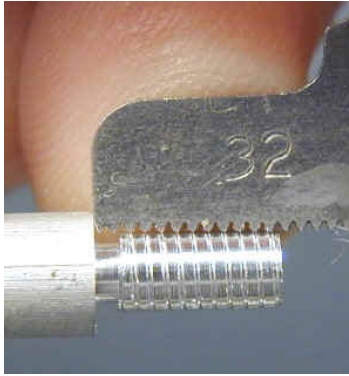
Cutting an 8-32 Thread

This time we will cut an 8-32 thread up to a shoulder on a 1/4" diameter aluminum rod.

Chuck a piece of 1/4" aluminum stock with about 3/4" extending from the chuck. Consulting the thread chart, we determine that the nominal major diameter for an 8-32 thread is 0.164". To allow for clearance, turn the workpiece down to 0.161" for about 1/2", leaving a smooth surface. Now make a test thread and check it with the 32-tooth gage. Use a cutoff tool to form a shoulder with a shallow groove where the thread will end. This is better way to form a nicely finished thread than in the previous example where we merely withdrew the tool at the end of the cut. The shoulder provides a seating surface for a nut or other surface with which the thread will engage.



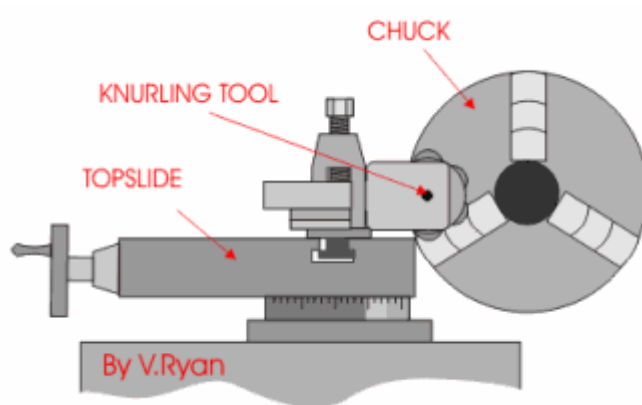
Now make a shallow test thread and check it with the 32-TPI gage. If the threads match the gage, then you are ready to cut the thread; if not, double-check the gear chart and gear train to make sure you have set up the gears properly.



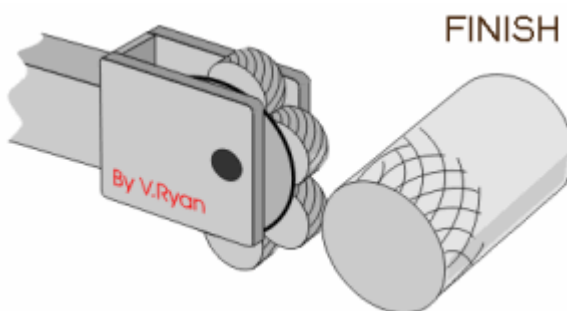
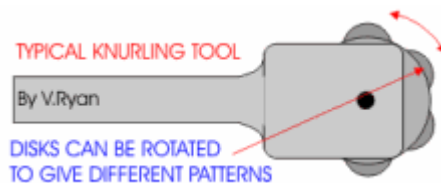
Now it's just a matter of following the standard procedure until you reach the proper Dial Depth, .022" in this case. Refer to the thread dial chart on the front of the gear cover and you will see that, for a 32 TPI thread, you can engage the half-nut lever at any of the marks (1~8) on the dial. Again, here's the procedure:

- Set the cross-feed to the original zero position
 - Advance the compound feed by a few thousandths - smaller amounts on each pass
 - Engage the half-nut lever when your number comes up
 - Disengage the half-nut lever when you reach the end of the thread
 - Back off the cross-feed one full turn
 - Crank the carriage back to the starting point
 - Repeat above steps until depth has been cut to specified Dial Depth from the chart
-

HOW TO USE A KNURLING TOOL



A knurling tool is used to press a pattern onto a round section. The pattern is normally used as a grip for a handle. Apprentice engineers often manufacture screwdrivers. These have patterned handles, to provide a grip and this achieved through the technique called knurling. The pattern produced is called a 'knurled pattern'.



This diagram shows the knurling tool pressed against a piece of round section steel. The lathe is set so that the chuck revolves at a low speed. The knurling tool is then pressed against the rotating steel and pressure is slowly increased until the tool produces a pattern on the steel.

The automatic control lever is engaged which starts the automatic traverse of the saddle. As the saddle moves along the bed of the lathe the knurled pattern is pressed into the steel along its length. If the traverse of the lathe is

stopped and then reversed a diamond pattern is produced.



Depending on the knurling tool selected, a variety of knurled patterns can be produced. Three typical patterns are seen opposite

Using a Boring Bar

A boring bar is used in the cutting of an inner surface. It can make a hole, which has a large accurate diameter, and smooth surface finish.



Boring Bars

Setting of a Boring Bar

Why is the boring processing difficult?

The boring processing is said to be difficult for some of the following reasons:

- (1) Its hard to see the processing surface.
- (2) The scraps are kept in the hole. When the scraps are kept, the surface cannot be finished to high roughness.
- (3) Because we cannot see the bottom of a deep hole, it is hard to stop the boring bar at the bottom location. We must depend on the scale of the lathe and the sound.
- (4) Especially the case of a small hole, the backside of the boring bar touches the material. We must set the height of the edge suitably.
- (5) Because a boring bar is long, the tool has vibrations easily. It is dependent on the rigidity of the boring bar.

Actual Boring Process

The following photographs introduce the actual boring processing of a mechanical part of a fish robot. The material is aluminum alloy (JIS A2017). The size is 118 mm of outer diameter and 191 mm of length. The hole is 103 mm of inner diameter and 186 mm of depth. The part is waterproofed by an O-ring and the inner surface needs to have a smooth surface finish.



(1) An undersized hole is made with a drill.



(2) The hole is spread with a boring bar.



(3) The edge must not attack to the bottom surface.



(4) It is hard to see the inner surface because of swarf.



(5) As the diameter is enlarged, cutting of the thin material becomes harder.



(6) The boring processing is finished. It has a beautiful surface finish.

Lathe Accessories

Aluminum Round Stock



Aluminum is an excellent material to practice on since it is inexpensive, cuts easily and takes a nice finish. I use it for most of my projects unless there is some reason to use another material, such as the need for extra strength.

In practice, there are many varieties of aluminum. During the refining process aluminum is mixed with specific proportions of other metals to produce desired characteristics of strength, weight, corrosion resistance and machinability in the combined metal, which is called an alloy. Many different alloys are available to meet specific needs, but '6061' alloy is a good choice for working on the lathe.

You are unlikely to find 6061 or similar alloys at your local hardware store - the kind you find there is typically very soft and gummy and does not machine well. To get the good stuff you will need to order from an industrial supplier such as Online Metals, Metal Mart or Enco. A good starter supply would be 3' lengths of 1/8", 1/4", 1/2" and 3/4". Some of the online metal suppliers have kits that include a wide range of stock for about \$100.

From some suppliers you must specify the length that you want, and they may charge a 'cutting fee' to cut the material to size. Other suppliers sell

stock in fixed length of 2', 3' or 6'. Typically I buy aluminum in 6' lengths for diameters up to 1". Since the amount of metal in a rod increases with the square of the diameter, the price goes up steeply for larger diameters. To give you an idea, here are some prices from the 2003 Enco catalog for 6 foot lengths of 6061 round stock:

Diameter	Price
1/4"	2.74
3/8"	4.26
1/2"	6.90
1"	22.20
2"	80.68

Fortunately, when working with larger diameters, the workpieces are usually relatively short, so you don't need as much.

Bench Grinder



You will need a 6" bench grinder for grinding tool bits..

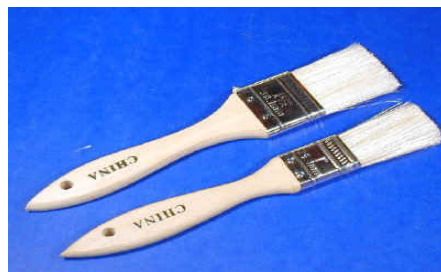
Brass Round Stock



Brass is a nice material to work with, though somewhat expensive compared with aluminum or steel. It can add a nice touch of contrasting color to a project that will be displayed. The alloy most often used for home shop work is 360. You can obtain it from the typical suppliers such as Online Metals, Metal Mart or Enco.

Center drills come in many sizes but you will definitely need sizes #1, #2 and #3 (good idea to get 2 of each). You can buy sets of #1-#5 on sale for under \$5.00

Chip Brushes

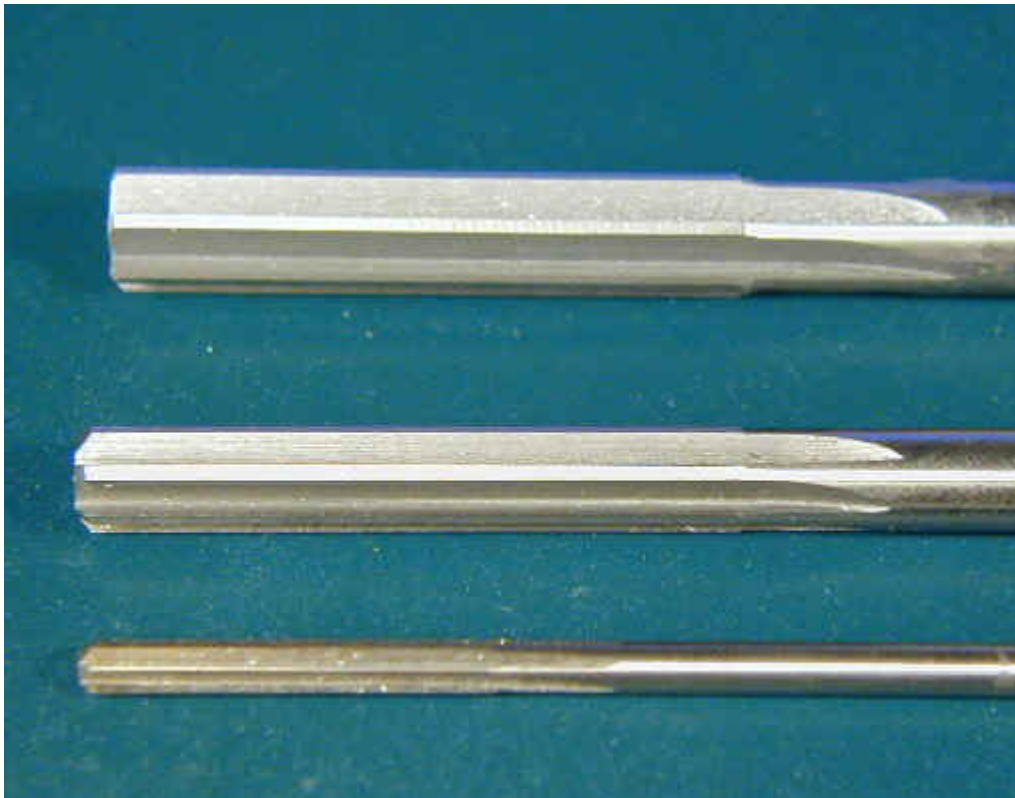


Chip brushes are inexpensive paint brushes that are handy for all kinds of uses around the shop. I'm not sure where the name originated, but I don't think it had anything to do with removing chips; nevertheless this is one of the uses these brushes excel at. They are also ideal for use with Kerosene or WD-40 to clean the packing grease off a new lathe or mill. Incidentally, experienced machinists will tell you always to use a brush, rather than

compressed air, to clean the chips from machine tools as compressed air will drive the chips deep into the recesses of the machine. Chip brushes and a shop vac are the preferred way to clean up chips.

You can get them from the usual industrial suppliers such as MSC, J&L and Enco and can usually find them at the big-box hardware stores in the paint section. At about \$.30 to \$.60 each depending on the size, they are cheap enough that you won't feel too bad about tossing them out if they get too gunked up, but they hold up for years in most applications. Get several in each size such as 1", 2" and 3" and keep them around the shop. You can't have too many!

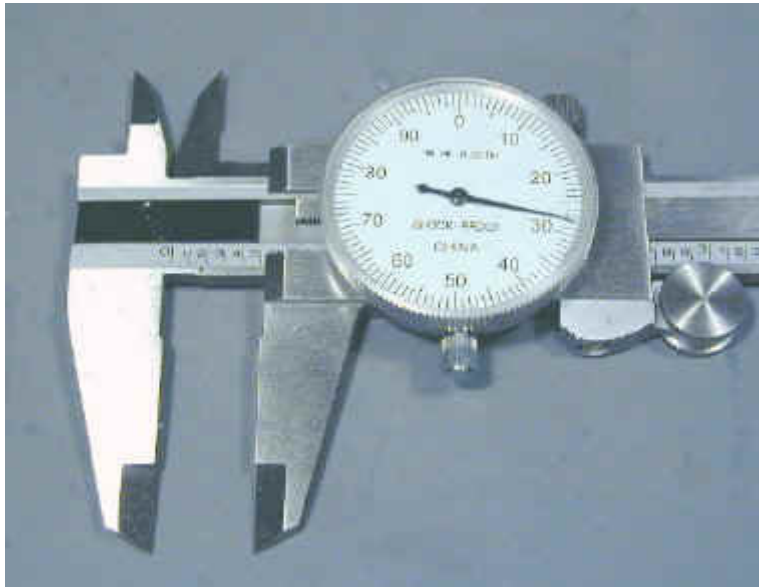
Chucking Reamers



Chucking reamers are used to make the final cut on relatively small diameter holes (typically 1/2" or less). Unlike a standard twist drill bit, a reamer will form a hole that is exactly round in cross-section and with a diameter accurate to .001". In use, they are clamped in the tailstock chuck or a drill press chuck and advanced into the work as in a drilling operation. They are relatively expensive so it makes sense to buy them one or a few at a time as you need them.

They are available in nominal sizes such as 1/8", 3/16", 1/4", etc., and also sized .001 under the nominal size or .001 over the nominal size, for example, .249 and .251. The under size reamers are used when you want a press-fit or tight fit and the oversize reamers when you want a free or relatively loose fit, as for an axle or rotating shaft. You can buy sets that contain one undersize and one oversize reamer for each nominal size, in increments of 1/16". These are very useful to have on hand when you need them.

Dial Caliper



I find a dial caliper to be indispensable in my shop - it is one of the tools I use most frequently. In fact, I keep 2 or 3 around so that if one gets misplaced or broken work does not come to a stop. With it you can measure diameters, depths of holes and inside diameters of holes to an accuracy of .001". Good quality ones are available from J&L, Harbor Freight, Grizzly, Enco, etc. for around \$15 for a 6" stainless steel version.

Watch out for (1) vernier calipers - these do not have a dial, and are a pain to read since you have to interpolate the reading (2) cheap plastic calipers. In the catalogs they look like the real thing, and often cost as much, but they are junk. Make sure the catalog specifies stainless steel.

Digital Caliper



[Harbor Freight SKU 47257](#)

Digital calipers are used just like dial calipers for making inside and outside measurements accurate to one, one thousandth of an inch but have a direct LCD digital readout. On a dial caliper you first read the major dimension to the nearest tenth of an inch from the slide and mentally add to that the minor dimension from the dial to the nearest thousandth. This becomes second nature after a while, but still introduces opportunities to make a mistake. A digital caliper reads out the full dimension on the display, so is pretty foolproof, as long as it is properly zeroed. You can also switch between metric and inch modes as needed.

Drill Rod



Drill rod is a steel alloy with a shiny silvery color (thus it is known in the U.K. as silver steel) and good machining properties. Unlike other raw materials, which may vary from the nominal diameter by + .010" or more,

drill rod is surface ground to within about .001" of the nominal diameter. For many applications, this means that the outside surface requires no additional finishing. While not classified as a stainless steel, drill rod is moderately resistant to rust - more, at least, than ordinary carbon steels. It is great for applications such as shafts and axles. It is available from the usual industrial suppliers typically in 3-foot lengths. Don't confuse this material with Drill Blanks which are short lengths of hardened high-speed steel used for making drills.

A useful property of drill rod is that it can easily be hardened by heating to a red hot state and then quenching in oil or water. Thus treated, the metal is hard enough to use for tools such as punches. Depending on the quenching method, drill rod comes in three types: air hardening, water hardening and oil hardening.

Drill Sets

Drilling is one of the most commonly performed operations on the lathe, so you will need a good collection of decent-quality drills. When you buy your lathe, don't forget to order a tailstock, chuck and arbor to hold the drills.

Poor quality drills are easy to find, but they are truly a waste of money. That's not to say that you need to buy top quality industrial drills as there are good quality import drills available from the usual suppliers such as MSC, J&L, Harbor Freight, Grizzly and Enco. U.S. made drills from reputable industrial suppliers are nearly always of excellent quality, but some people have purchased some U.S. made drills from hardware stores that were real junk.

It's not always easy to tell a good drill bit from a poor one just by looking and, of course, it's even harder if all you have is a picture on a web site or in a catalog. Generally, though, the lowest priced drill sets are the ones to stay away from.

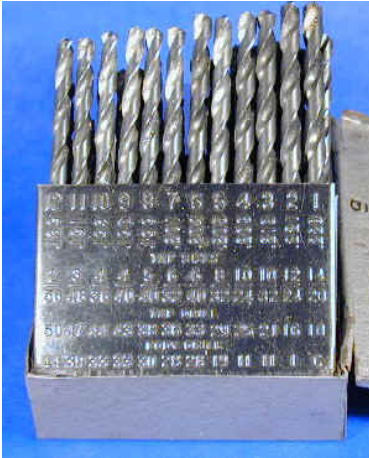
Drill bits come in several standard size ranges:

- **Fractional drills; often sold in sets of 29 from 1/16" (.0625) to 1/2" (.500) in increments of 1/64"**
- **Number drills; sold in sets of 60, numbered #1 (.228") to #60 (.040")**
- **Tiny wire gauge number drills; sold in sets of 20 from #61 (.039) to #80 (.0135")**
- **Letter drills - less commonly used; sold in sets of 26 from A (.234") through Z (.413")**
- **Metric drills - sold in 19 pc. sets from 1mm to 10mm in increments of .5mm**
- **Silver & Demming drills - large size (17/32" to 1") drills with a 1/2" diameter shank**

If you are just starting out, a good choice is a set of 29 fractional drills and a set of 60 number drills. Unless you do a lot of metric work, these will meet most of your needs on the lathe.



- **Set of 29 TiN (Titanium Nitride) coated fractional inch drills**



Set of 60 black oxide coated number drills



Set of 119 fractional, number and letter drills.

There are several other factors that you need to be aware of when selecting drills. The drills commonly used are called "jobber" drills. These are the type that we are all familiar with that you would find in your local hardware or automotive store. They have a straight shank, the same diameter as the drill and a 119° tip. Another type that can be handy for the lathe, are screw machine drills. These look much like jobber drills but are shorter - which can be an advantage within the short confines of the lathe.

Another factor to consider is the coating of the drill bits. Typically you will find three coating options:

- Uncoated bright steel
- Black Oxide
- TiN (Titanium Nitride)

Coated drill bits, and especially the newer TiN coated ones, shed chips more easily and are thus less likely to bind or get chips welded to the drill bit surface. Aluminum is especially subject to chip welding, and if it

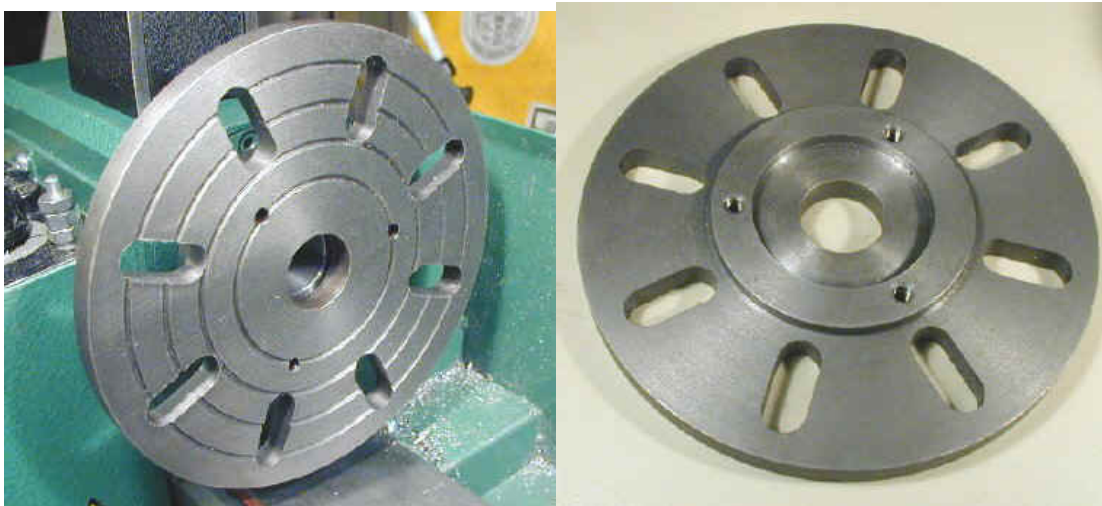
occurs, the welded chips can scour the hole's surface and generally mess things up. Additionally, while extremely thin, the TiN coating is very hard and is said to increase drill bit life. I have had good service from TiN coated drills and can definitely recommend them. Usually these are better quality drills and selecting them makes it much less likely that you will end up with really poor quality drills.

A final consideration is the material that the drills are made from. Stay away from ordinary carbon steel drill bits - these are often sold in hardware stores and are suitable only for wood and soft materials. The only practical choices for lathe work are high speed steel (HSS) and Cobalt steel. The cobalt ones are harder and will last longer, but are also more expensive. HSS is a good choice if, like most of us, you are on a budget. One tip to extending the life of any drill set: when drilling a hole, such as a pilot hole, for which the exact drill diameter is not critical, choose an off-size bit. This will save wear and tear on the commonly used sizes such as 1/4" and 3/8".

This covers just the real basics of drill selection. There are dozens, if not hundreds, of specialized drill types used in industry, but for our needs the basic jobber drills in HSS are generally adequate.

Faceplate

A faceplate is a handy accessory for turning odd-shaped work that cannot easily be held in a chuck.



The faceplate mounts to the 9x20 spindle in the same way as the chuck, but the studs required are not provided. You can make your own by parting the heads off of some bolts, if you do not have a source for the studs.

After mounting the faceplate on the spindle, it is standard practice for faceplates to take a light one-time facing cut to ensure that the face of the plate is square with the lathe. Cast iron dust is very harmful to breathe, so

I strongly recommend wearing a dust mask during this operation and until you have vacuumed up the resulting dust.

When using a faceplate, always ensure that the work is securely clamped down and balanced by some offsetting piece of metal, if necessary. Work at low RPMs.

Follower Rest

A follower rest is similar to a steady rest, but is attached to and travels with the carriage to provide a moving support for the work behind the cutting tool. This is very handy when trying to turn limber work which would otherwise bow out away from the tool. If you have ever wondered about the two screw holes on the left edge of the carriage, now you know what they are for - they are the mounting holes for the follower rest.



Live Center

Centers are often used in the tailstock to support the end of a relatively long and limber workpiece. The lathe comes with a #2 MT "dead" center. The drawback of a "dead" center is that the center does not rotate, while the workpiece that it supports does, leading to friction and possibly overheating. By contrast, the tip of a "live" center rotates freely in bearings, and rotates with the workpiece so that the friction is greatly reduced.



Live Center mounted in tailstock ram

The Harbor Freight used to (and may still) come with a #2 MT "live" center. You can purchase them from a variety of sources, but you will want a very small one with a #2 Morse Taper arbor to work effectively on the lathe.

Milling Attachment (10/05/02)

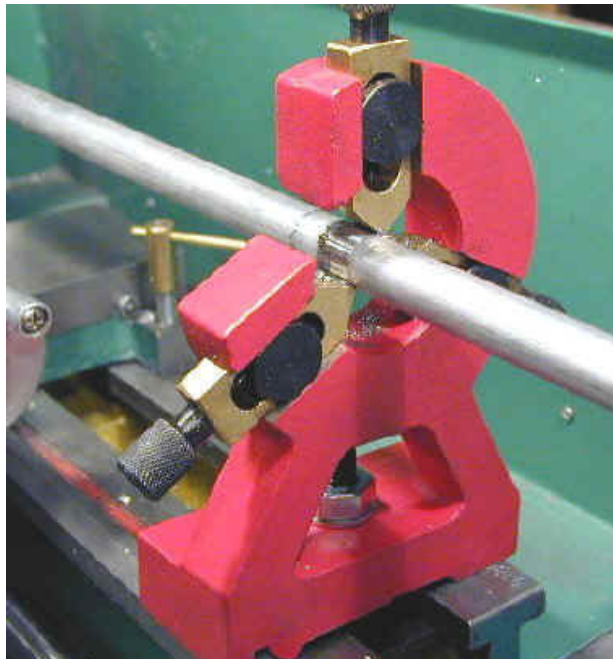
Before this accessory became available, many lathe owners made their own versions based on a standard milling vise. The commercial version, shown below, uses socket head screws to hold the workpiece between the jaws. While by no means a substitute for mini-mill, this accessory is an inexpensive way to add limited milling capability to your lathe while you save up for a mill.

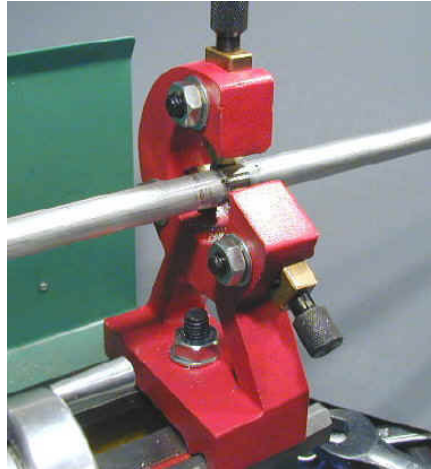


Milling Adapter from LMS

Steady Rest

A steady rest, for the uninitiated, is clamped to a fixed point on the ways, usually near the end, and has adjustable 'fingers' that are adjusted so that they lightly contact the outside of a long and/or limber workpiece to keep it from wobbling or thrashing.





On this model, the fingers are brass and are adjusted by means of thumb screws and then locked in place by means of lock nuts. Use a few drops of oil to lubricate the contact surface between the work and the fingers to keep the work from heating up and binding. The steady rest clamps to the ways with a clamp much like that on the tailstock, and with the same frustrating tendency to rotate to an orientation which does not fit between the ways.

T-handle Metric Hex Wrench Set



OK, these aren't *really* essential, but they are so handy and so cheap you will kick yourself if you live without them and then later try them. I use these for nearly all the hex head screws on the lathe.

Tailstock Chuck and Arbor

Drilling is a fundamental lathe operation and you will need a chuck and #2 Morse Taper arbor to do it. The arbor has a thread or Jacobs taper on one end - to mate with the chuck – and a #2MT on the other end to mate with the tailstock cylinder.



It is desirable to keep the ram extended as little as possible in order to maximize its rigidity during the drilling operation.

To remove the arbor, place the chuck in your bench vise with the jaws open just a little wider than the arbor diameter (not clamping the arbor). Open the jaws up and use a short piece of round stock or a drift pin to drive the arbor out of the back of the chuck. This should only require a fairly light tap of the hammer. It's a good idea to position a rag underneath the arbor to catch it so that it does not get dinged up by falling to the floor.



Tool Blanks

Yes, you *could* buy ready-made carbide tool bits, but learning to grind your own HSS tool bits is a valuable skill and part of the fun of learning to use the lathe. You will soon learn to grind special tools for parting, boring and other operations.



Tool blanks are available from many sources; usually found at [Enco](#) (on sale for as little as 80 cents each) but have also bought them from Grizzly, J&L and other suppliers

Transfer Punch Set

Transfer punches come in sets sized like twist drills. Import sets are available for under \$10 and are fine for hobbyist use since they are used infrequently. They are hardened steel with a pointed tip and are used to 'transfer' the location of a hole in one workpiece to a mating surface in another workpiece. A punch is selected that just fits within the original hole and is struck lightly with a hammer to make a punch mark that will be at the center point of the hole. They are also handy as for determining the inside diameters of small holes.



Wet/Dry Sandpaper



Wet / Dry sandpaper is very handy for putting a very fine shiny finish on metal workpieces in the lathe. In that application it is generally used dry, but when used wet on a flat backing surface it does a great job of smoothing flat metal workpieces from milling operations. You will need a variety of grits such as 220, 320, 400 and 600 and can sometimes find assortment packs that contain such a variety. You can find it in the paint section of most hardware and automotive stores.

Links and Recommended Reading

This work represents mere guidelines for the successful operation of an Asian 9x20 Lathe. The following sites contain vast amounts of data in its archives and may be considered a full treatise on the subject:

Note: The yahoo groups “Search Archive” feature is practically useless. A program called PG OFFLINE was used to effectively search yahoo group archives.

Lathe Groups:

<http://groups.yahoo.com/group/9x20Lathe/>

<http://groups.yahoo.com/group/9x20Lathe2/>

<http://groups.yahoo.com/group/9x20Lathe/database?method=reportRows&tbl=1>

JWE's MW Groups

A post by JW Early in the 9x20lathe group, message #23598

As per recent discussion here is an updated list of the metal working related groups I am maintaining at the current time. There are some others I have contributed content to that are owned and moderated by others. These groups are for educational purposes and not for personal profit whether for myself or others. Because they are educational in nature discussion might drift to points some might consider Off Topic because of the need to explain reality to some who might not have had a complete education. Archive group for the Geometer Beginners Workshop series from the mid 1950s and the Hints & Tips series from the late 1950s and early 1960s.

(01) <http://groups.yahoo.com/group/Geometer/>

Archive group for the Novices Workshop series from the early 1950s as well the tutorial Chuck cartoons by Terry Aspin from the early 1960s are archived here.

(02) <http://groups.yahoo.com/group/beginnerswkshp/>

Archive group for the useful hints and tips from Model Engineer that were posted on mlprojects.

(03) <http://groups.yahoo.com/group/hintstips/>

Searchable indexes for Model Engineer magazine from 1939 to 1999. Also partial indexes for some other magazines with metal working content.

The one for HSM and PiM/MW are the most complete but not fully verified or commented.

(04) <http://groups.yahoo.com/group/meindex/>

Lathe modification and improvement articles for the MiniLathe and other home shop machines. Alignment, collets, fine feed ideas, tailstock mods and ideas and ideas for quick change gearboxes.

(05) <http://groups.yahoo.com/group/mlathemods/>

Dials, reconditioning tricks and taper attachments.

(06) <http://groups.yahoo.com/group/mlathemods2/>

Making keyways, spherical turning attachments, taper turning attachments and worm gear attachments.

(07) <http://groups.yahoo.com/group/mlathemods3/>

Attachments for milling in the lathe, threading and quick change or improved tool posts.

(08) <http://groups.yahoo.com/group/mlathemods04/>

An idea on how to make a flex shaft tool and the article archive for the Jeynes series.

(10) <http://groups.yahoo.com/group/mlathemods4/>

Discussion and aid with control and motor problems on the mini lathe.

(11) <http://groups.yahoo.com/group/mlathemods5/>

Model building projects mostly related to I/C engines, plus some OT content from time to time as this group was originally started to draw the OT content away from the 7x10 group.

(12) <http://groups.yahoo.com/group/mlprojects/>

I.C. engine projects.

(13) <http://groups.yahoo.com/group/mlprojects1/>

Steam engines and related materials.

(14) <http://groups.yahoo.com/group/mlprojects2/>

(15) <http://groups.yahoo.com/group/mlprojects2a/>

Steam engines and boilers

(16) <http://groups.yahoo.com/group/mlprojects2b/>

Steam traction and road going equipment.

(17) <http://groups.yahoo.com/group/mlprojects3/>

Steam powered fire engines and pumps, also some interesting articles on hydraulic rams.

(18) <http://groups.yahoo.com/group/mlprojects4/>

Some old clock building articles from ME and as well discussion on building clocks and special tools and tricks for doing the job.

(19) <http://groups.yahoo.com/group/mlhorology/>

Metal working hints & tips for the beginner as well the more experienced user. Mechanical drawing, measuring tools, cross drilling and pattern-making.

- (20) <http://groups.yahoo.com/group/mwhints/>
Duplex on threading and some other hints and projects from this super team.
- (21) <http://groups.yahoo.com/group/mwhints2/>
Drill sharpening, tool sharpening, knurling and parting off.
- (22) <http://groups.yahoo.com/group/mwhints3/>
Centering work, use of chucks and other work-holding devices and more information on tapping.
- (23) <http://groups.yahoo.com/group/mwhints4/>
Some ideas for building your own lathe or improving the one you have.
- (24) <http://groups.yahoo.com/group/mwlatheideas/>
Tools and ideas for the small shop milling machine and shaper.
- (25) <http://groups.yahoo.com/group/mwmills/>
Indexing, dividing and gear cutting on the lathe and mill.
- (26) <http://groups.yahoo.com/group/mwmills2/>
Power hacksaws, saws and other secondary but useful shop equipment.
- (27) <http://groups.yahoo.com/group/mwsmalltools/>

Must have free Literature:

http://images.grizzly.com/grizzlycom/manuals/G4000_m.pdf

<http://www.metalworking.com/tutorials/ARMY-TC-9-524/9-524-index.html>

Technical Book source

<http://www.lindsaybks.com/HomePage.htm>

Recommended readings

<i>Title</i>	<i>Contents</i>	<i>Pros</i>	<i>Cons</i>	<i>Year/pages</i>	<i>Source</i>
How to Run A Lathe by South Bend	Good concise introduction to basic lathe operations	Good introduction to basics	A bit antique -- lantern tool holders etc	1942 / 128 pages	\$7.95 Lindsay Pub
Machine Shop Methods by Milne	Machine shop introduction	Clear diagrams. Nice section on helpful projects	Limited depth on some topics	1950 / 376 pages	\$18.95 Lindsay Pub
Machine Shop Practice Vol 1 by Moltrecht	A textbook on drilling and lathe work	Modern book with a great amount of detailed info	Emphasis on huge factory equipment	1981/ 496 pages	\$17 Enco
Machinist's Ready Reference Compiled by C. Weingartner	Mini Machinery's Handbook at a much better price	Spiral bound to lay flat. Easy to understand		308 Pages - 9th Edition - February 2001	24.95 new 15.97 used

Video rentals for \$10 per tape for a week. (USA only)

<http://technicalvideorental.com>

Compare 9x20 Lathes and prices

<http://www.grizzly.com/products/item.cfm?ItemNumber=G4000>

<http://www.harborfreight.com/cpi/ctaf/Displayitem.taf?itemnumber=45861>

http://www.southern-tool.com/store/birmingham_ycl-920_lathes.html

<http://www.penntoolco.com/catalog/products/products.cfm?categoryID=4799>

<http://www.penntoolco.com/catalog/products/products.cfm?categoryID=139>

http://catalog.jlindustrial.com/jlindustrial/circular_browse_page_large.asp?BPM=1&displaypagenumber=1951&rapid=33440&promotioncode=JnLIndustrail-Vol74

<http://www.use-enco.com/CGI/INSRIT?PARTPG=INSRAR2&PMAKA=110-0821&PMPXNO=954066>

<http://www.wttool.com/c/30030030p>

<http://www.wmhtoolgroup.com/index.cfm?area=shop&action=detail&iid=11017>

<http://www.busybeetools.com/> CANADA

Frequently asked questions of the 9X20 Lathe

Extracted from the yahoo group 9x20lathe database

Accessories	What accessories are available to use with the Lathe?	Standard accessories include a 4 jaw chuck, Faceplate and centers, steady rest, follower rest, and tailstock chuck. Owners have made milling attachments, travel stops, tool holders, and many others.
Accessories	What are the accessories included with the HF lathe?	The HF comes with a 4" 3-jaw chuck, a faceplate, simple centers, a bearing center, a steady rest, a follower rest, a set of change gears, two spare plastic 80T gears, some cheap tools and a leaky oil can.
Compound Slide	The compound slide seems flimsy. How can I improve its' stiffness?	The standard 2 bolt hold-down has been modified to a 4 bolt style with great success by some lathe owners. See details in the files section. http://groups.yahoo.com/group/9x20Lathe/files/
Crossslide	Is the crossslide powered?	No. The standard lathe does not have a powered crossslide. However, some owners are working to add it to their lathes. Yes, see msg 171 or
Drivetrain	Can you cut 4mm threads?	http://groups.yahoo.com/group/9x20Lathe/files/Photos/Collet%20Chuck/gear.jpg (you need to select position 1 on the quick-change)

		IF YOU HAVE A 160XL COGGED BELT YOU NEED TO USE A 5M720 V BELT. IF YOU HAVE A 170XL COGGED BELT YOU NEED A 5M730 V BELT. IF YOU USE THE 5M720 V BELT WITH THE 170XL COGGED BELT THE V BELT WILL BE TOO SHORT.
Drivetrain	Do I use a short or long belt when replacement is necessary?	
Drivetrain	How can I get finer feeds than what are listed in the specs?	The finest feed with the gears that are supplied with the lathe is around 0.0005" per rev. The geartrain for this is as follows: Position A: 28t stacked with 40t. Position B: 127t at outermost position. Position C: 30t stacked with 120t. QCGB: Position 9. When mounted at Position B, the bottom of the 127t gear is lower than the bottom of the lathe feet. Thus, you may need to raise the lathe in order to clear the gear. See related question on coarser feeds.
Drivetrain	What is the effective gearing of the worm/rack in the apron?	The worm/rack in the apron acts like a 55tpi leadscrew.
Drivetrain	Does the lathe include a tumbler reverse on the leadscrew?	No. The standard lathe does NOT include the ability to reverse the leadscrew rotation. Several members HAVE added this VERY useful feature to their lathes. See the bookmarks section for details...
General Information	Where can I buy a 9x20 Lathe?	The 9x20 Lathes are available from several sources. Enco, Jet, Busy Bee (in Canada) and Harbor freight tools carry them. See "Compare 9X20 Lathes and Prices" above.
Headstock	What type of spindle bearings are in the headstock?	The standard 9x20 has tapered roller bearings.
Lathe Bed	What is the construction of the Lathe bed? Are the ways hardened and ground?	The lathe bed is grey cast iron. Yes, the ways are hardened and ground.
Lathe Mounting	What size bolts should be used for mounting my lathe to my workbench surface.	12mm Diameter x whatever length is necessary to go through your surface. Washers and locking type nuts are recommended. Tighten the headstock side only (not excessively) and just snug the tailstock side so as not to warp the bed and twist the ways.

Conclusion

I hope this has been an informative document that answers some of your questions. For more in-depth technical information on the 9x20 lathe, download the free 9X20 Rebuild Manual or “Tricking out” (a 9x20lathe modification guide) compiled by Cletus Berkeley which have been compiled in conjunction with this document.

It’s been fun compiling this document and I would like to thank Cletus Berkeley for his support and guidance, Chris Boyer for his helping attitude and Rich Hare for his editing patience. I would also like to thank all past and present 9x20lathe group members for their contributions to help the “newbies” like me. In particular:

Peter Bowen, Jack Fuseler, J.W. Early, Ballendo, Terry Brown, Chris "cba_melbourne", Tom “turyga1963”, Gene Furr, Ed Kadlec, Des Young, Dave Knutsen, Karl Kulhavy, and James J. Cullen, Ph.D.

Colin

This page intentionally left blank

Because I didn't know how to delete it.<G>