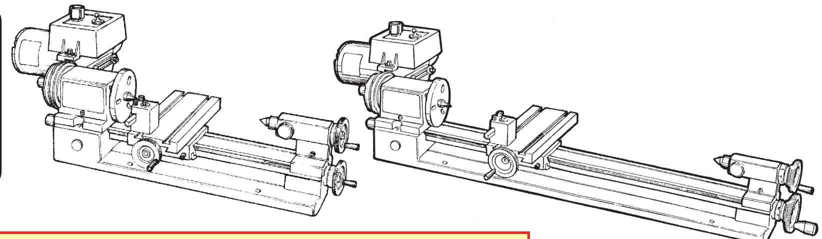


SHERLINE PRODUCTS

INCORPORATED 1974

CAUTION!
Read All Operating Instructions Carefully Before Attempting Any Machining Operations.

Lathe Operating Instructions

Leveling the Cutting Tool

Each type of turning work requires the correct tool for the job. It is important that the cutting tool be sharp and correctly set up in the tool post. The cutting edge of the tool should be exactly level with the center height of the lathe. Check this by bringing the tool tip up to the point of either the headstock center or tailstock center. (See Figure 1 A.) We also manufacture a simple tool height adjustment gauge that allows you to check tool height at any time by measuring from the table surface. (See Figure 1 B.)

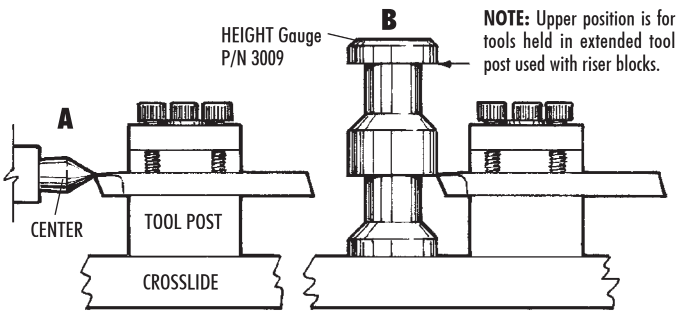


FIGURE 1—Leveling the tool using (A) the tip of a head- or tailstock center or (B) Sherline's tool height gauge P/N 3009.

The standard Sherline tool post is designed to hold common 1/4" square tool bits which have had a few thousandths of an inch (.1 mm) ground off the top edge for sharpening. Loosen the hold-down bolt and slide the tool post as close to the point of the dead center as possible. Inspect with a magnifying glass. The tip of the tool bit may be raised or lowered by sliding a shim* underneath it. The cutting edge must be on center or just below center (0.004" or .01 mm maximum). Ensure that the tool is fixed securely in position by firmly tightening the socket head screws. Try not to have the tool cutting edge protruding more than 3/8" (10 mm) from the tool post.

* **NOTE:** Thin metal shim stock is available for this purpose. If you don't have any metal thin enough, a single thickness of paper business card stock will usually do the job. Do not use more than one thickness as it will compress too much. Our optional rocker tool post (P/N 3057) allows this adjustment to be made without shims. It comes standard with the Model 4400/4410 long bed lathe.

Initial Test Cutting

If you have never operated a lathe before, we suggest that you make a trial cut on a scrap of material to learn the operation of the machine. In a 3- or 4-jaw chuck, secure a piece of round aluminum stock approximately 3/4" (19 mm) diameter and 1-1/2" (38 mm) long. Secure the pre-sharpened 1/4" square right-hand cutting tool supplied with the lathe in the tool post, making sure that it is properly positioned. First, turn the speed control all the way counter-clockwise, then turn the motor on. Bring the speed up to approximately 1000 RPM (about 1/3 speed). To establish tool position in relation to the work, bring the tool in slowly until it just starts to scribe a line on the work. Crank the tool towards the tailstock until it clears the end of the work. Advance the tool .010" (.25 mm) using the crosslide handwheel (10 divisions on the inch handwheel scale). Using the bed handwheel, move the tool slowly across the work toward the headstock.

Cutting tools used on lathes are designed to remove metal much as paper is removed from a roll. It takes a positive feed rate to accomplish this. If the feed rate isn't fast enough, it would be similar to tearing an individual sheet of paper off the roll. The results when cutting metal would be shorter tool life, a poor finish and tool "chatter." Chatter is a function of rigidity, but it is controlled by speed (RPM) and feed rate.

Since you already have a piece of aluminum chucked up, experiment with speed and feed rate. You just took a cut of .010" (.25 mm) and probably noticed that the machine didn't even slow down in the slightest. Now take a 1/2 inch long cut .050" or 1 mm deep, which is one complete revolution of the handwheel. If you used the sharpened cutting tool that came with your machine, it should have made the cut easily. If the tool "squealed", reduce the RPM a little and take another .050" cut while feeding the tool faster. You will probably be surprised at how easily your machine takes cuts this heavy.

Inducing Chatter and Learning How to Overcome It

To better understand what is going on, we will now purposely try to make the machine "chatter." Make sure the stock you are cutting is sticking out of the chuck no more than 1 inch (25 mm). Crank the handwheel two turns further in from the last setting which will give you a .100" (100 thousandths of an inch) or 2 mm cut. Set the spindle speed to about 1000 RPM (1/3 speed) and feed the tool slowly into the material. Vary speed and feed until you get a substantial chatter. Without changing the depth of the cut, drop the speed to about 200 RPM and feed the tool into the work with more force. The chatter should disappear. Once you have learned to control chatter by adjusting speed and feed, you will be well on your way to becoming a machinist.

Holding the Workpiece

Work can be held between centers, in 3-jaw or 4-jaw chucks, on the faceplate or with a collet. Sometimes it is necessary to use a chuck and center, and, if the work is spinning fast, a live center should be used. (See Figures 2, 3 and 4.)

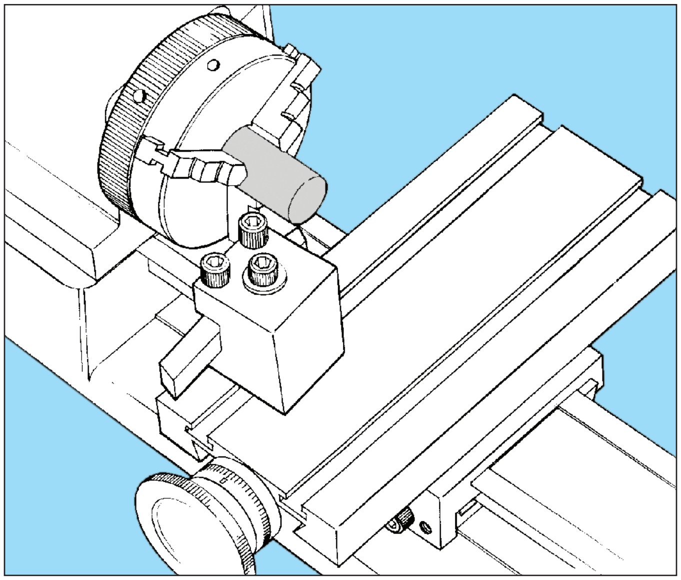


FIGURE 2—Holding a round work piece in a 3-jaw chuck.

Turning between Centers

This is done by fitting the dog to the work which is to be turned and placing the work and dog between the centers in the headstock and tailstock. The maximum diameter that can be held with the dog is 5/8" (15 mm). (See Figure 4.)

The dog is driven by fitting it into one of the faceplate holes. This method of turning is ideal for bar work or turning of steps on a bar. The tailstock center must be greased to prevent overheating. (An optional live center—such as P/N 1191—turning on ball bearings is the solution preferred by most machinists.) The headstock spindle has a #1 Morse taper in the spindle nose. The tailstock spindle has a #0 Morse taper.

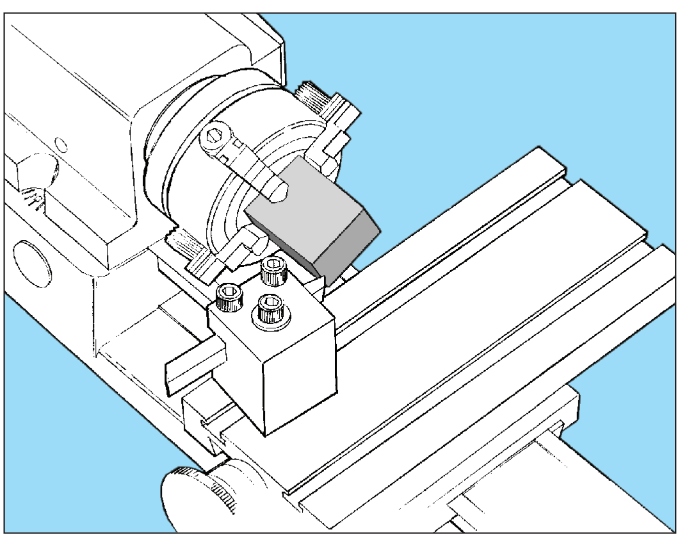


FIGURE 3—Holding a square work piece in a 4-jaw chuck.

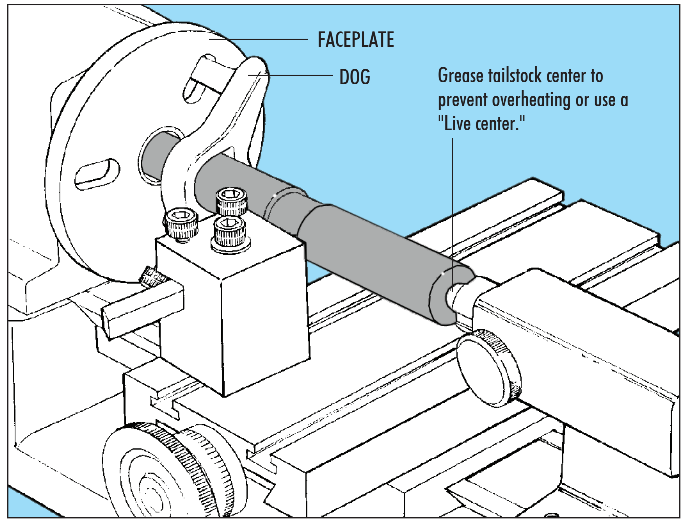


FIGURE 4—Turning between centers with a faceplate and drive dog.

Removing Tools from the Morse Taper Spindles

HEADSTOCK—Accessories held in the Morse #1 taper of the headstock spindle can be removed with the use of a knockout bar (not supplied) approximately 3/8" in diameter and 6" long. The bar is inserted through the back of the spindle, and accessories, such as centers, can be removed with a few taps. Accessories like the drill chuck that are drawn into the spindle taper with a drawbolt are removed by loosening the drawbolt a few turns and then giving the head of the bolt a sharp tap with a mallet to break the taper loose. Supporting the headstock by lowering it onto a block of wood extending to the table on the mill will keep from knocking the column out of alignment.

TAILSTOCK—The tailstock spindle does not have a through hole and a drawbolt is not used. It is equipped with a Morse #0 taper, and accessories such as drill chucks and centers can be removed by turning the handwheel counter-clockwise until the back of the taper hits the inside of the spindle and the accessory is ejected.

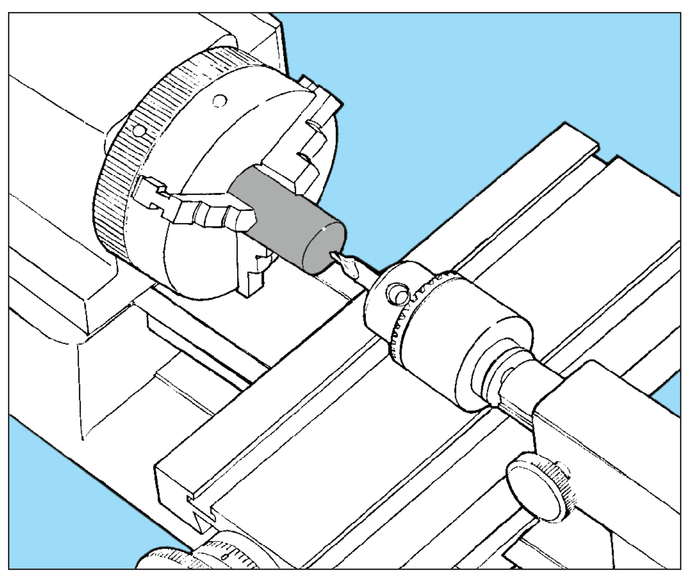


FIGURE 5—Tailstock center drilling. The work turns while the drill is held stationary in the tailstock.

Center Drilling

Because the work turns and the drill does not on a lathe, it is necessary to use a center drill before a standard drill can be used. Due to the flexibility of a standard drill bit, it will tend to wander on the surface of the rotating work, whereas a center drill is designed to seek the center and begin drilling. The 60° point of the center drill makes a properly shaped index hole for the tip of a live or dead center. It also provides an accurate starting point for a standard drill. Cutting oil is recommended for all drilling operations. A center drill should be withdrawn, cleared of chips and oiled several times during the drilling of a hole to keep the small tip from breaking off.

Tailstock Drilling

Hold the work in a 3- or 4-jaw chuck. If the work is longer than approximately 3" (76 mm), support the free end with a steady rest. Seat the drill chuck's #0 Morse arbor into the tailstock spindle and secure a center drill in the chuck. Adjust the tailstock to bring the center drill close to the work and lock it in position. Turn the tailstock handwheel to bring the center drill forward. After the hole is started with the center drill, switch to a standard drill bit of the desired size to drill the hole.

The easiest way to center drill the end of a round shaft that has a diameter too large to be put through the spindle is to support it with a steady rest (P/N 1074) while the end is being drilled. If this isn't possible, find the center with a centering square, prick punch a mark and center drill by hand.

Headstock Drilling

The drill chuck comes fitted with a #0 Morse arbor that fits in the tailstock spindle. To use it in the headstock, you will need to first change to the #1 Morse arbor that is included with your chuck. To change arbors, put the drill chuck key in its hole to give you better purchase to grip the chuck while using a wrench to remove the #0 arbor. Replace it with the larger #1 arbor. Put the drill chuck in the headstock. Then put the drawbolt with its washer through the spindle hole from the other end of the headstock and tighten the drawbolt. DO NOT OVERTIGHTEN! (See Figure 6.)

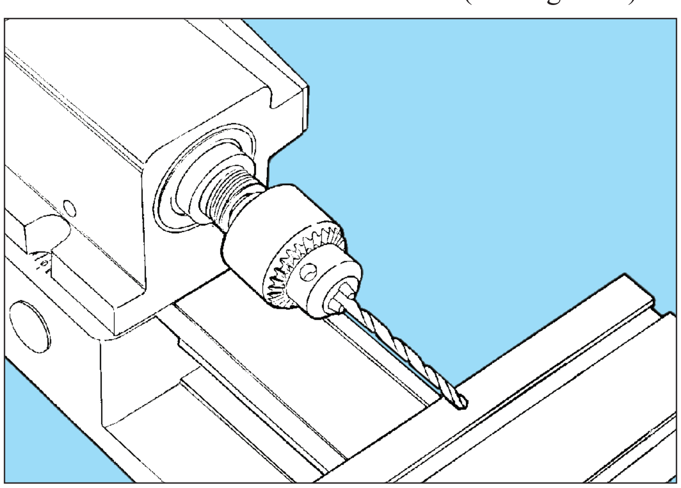


FIGURE 6—Headstock drilling. The drill turns in the headstock spindle while the work is held stationary.

Reaming

Twist drills will generally not drill perfectly accurate sizes, and very small boring tools are not satisfactory in deep holes because of their flexibility. Therefore, reaming is used for holes requiring accuracy within .0005" (.013 mm). Reamers are available in any standard size, but they are rather expensive and are generally not purchased to do one-of-a-kind type work. Use them only when a boring tool cannot be used because of the depth or size of the hole. Because of their length, they cannot always be used on a small lathe.

Reamers are used only to "clean up" a hole. To make an accurate hole, the work is drilled approximately .010" (.25 mm) smaller than the reamer size. The work should be slowly rotated and the reamer slowly fed into the hole while applying plenty of cutting oil. The reamer should be frequently removed and cleared of chips. Never rotate a reamer backwards in the work as this can dull the cutting edges.

Faceplate Turning

The faceplate has three slots that allow work to be bolted to its surface. Flat work can be screwed directly to the faceplate. Extra holes can be drilled to suit odd shaped work unsuitable for a chuck. If the work is mounted off-center, be sure to counterbalance the faceplate and use very low RPM. Don't hesitate to drill holes in or modify the faceplate as needed to do a particular job. That's what they are for. They are inexpensive and you can have several on hand modified for special jobs.

Taper Turning

On some lathes, a taper is cut by offsetting the tailstock. On the Sherline lathe, taper turning is done by removing the headstock key and turning the headstock to any angle away from dead center. To rotate the headstock, the alignment key must first be removed. Loosen the set screw in the front of the headstock, and lift the headstock and motor unit off the locating pin. Tap the alignment key out of its slot on the bottom of the headstock, and replace the headstock unit

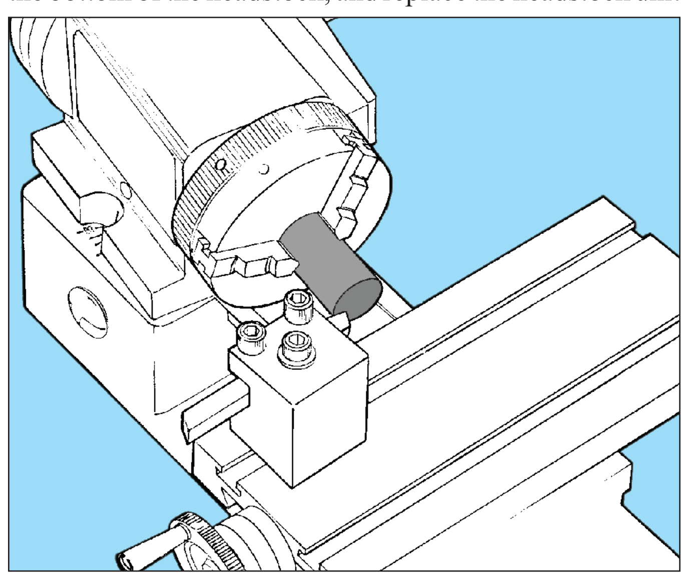


FIGURE 7—Turning a taper with the headstock slightly rotated.

on the pin. While pressing down on the headstock, rotate it to the angle you desire by referring to the angle scale on the bed. The base is calibrated in 5° increments up to 45° on either side of center. When set to the proper angle, retighten the set screw against the pin to lock the headstock into position. Tapers can also be cut without turning the headstock by using a compound slide (P/N 1270).

Short work can be inserted in a 3- or 4-jaw chuck and turned as shown in Figure 7. If the headstock is angled towards the lathe front, the taper will be cut smaller at the right. Tapered holes can also be bored in work held in the 3- or 4- jaw chuck. To machine a taper on longer stock, center drill both ends of the bar, set the headstock angle and mount the part between centers using a faceplate and drive dog. (See Figure 8.)

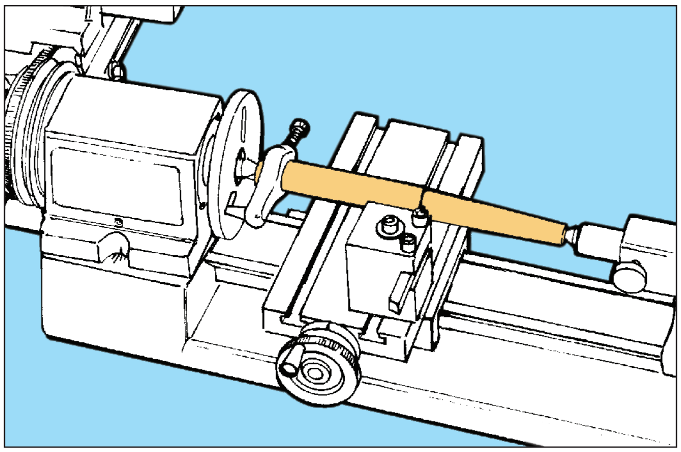
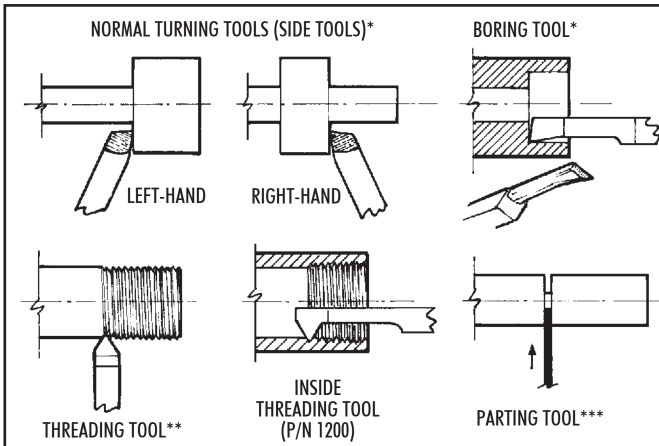


FIGURE 8—Long, shallow tapers can be cut in a continuous pass by pivoting the headstock to the proper offset while supporting the other end with the tailstock. The work is driven by using a drive dog in the faceplate. The dog acts like a "universal joint" as the drive pin slides in the faceplate slot. A dead center is used here in the tailstock but an optional live center could also be used.

Tool Shapes and Grinding Your Own Cutting Tools

The shaping of cutting tools to suitable angles for the type of material and nature of work being performed can be very important to satisfactory work. When tools become dull, gently re-grind and preserve the original angles and shapes. Do not grind the top face of the tools, but confine sharpening to the end and/or sides except form tools which are ground on the top surface. Remember that heavy cuts and rapid feed will cause greater strain on the chuck and lathe. This may induce "spring" or binding of work and tools that can produce a poor finish.



*These shapes are available in high speed steel tool set, P/N 3007.

**The 60° threading tool is included as part of the carbide tool set, P/N 3006 and also comes with the thread cutting attachment (P/N 3100.)

***The parting tool comes with the cutoff tool holders, P/N 3018. Other shapes are custom ground to accomplish special purposes as needed.

FIGURE 9—Cutting tool shapes.

NOTE: Because of the importance of a sharp and properly ground tool to the cutting process, Sherline has prepared a special instruction sheet on Grinding Your Own Lathe Tools. There are a few tips that can make the process a simple one. The instructions are included with each lathe and with cutting tool sets when you order them from us, or you may call us and request a copy. (Cost is $5.00 postage paid.) They are also available from our website at no cost. (See **Grinding Your Own Lathe Tools**.) Unfortunately, space does not permit us to reprint them as part of this booklet.

Cutting tools are ground to various shapes according to their usage. Tools are usually ground to shape as needed by the operator. Some standard tools are described below.

Normal Turning Tool—or RIGHT-hand tool feeds from right to left, is used to reduce work to the desired diameter and is the most frequently used of all tools.

Side Tools—These are used to face off the ends of shoulders and may also be used as normal turning tools. Note that a tool that is fed from left to right and has its cutting edge on the right is called a LEFT-hand side tool because the chip comes off to the left. Cutting tools are named based on which direction the chip comes off, not which side has the cutting face.

Parting Tool—The conventional parting tool or cutoff tool is shaped like a dovetail when viewed from above and is used to cut off work pieces by feeding the end of the tool across the lathe bed and through the work piece. The Sherline parting tool instead uses a thin .040" (1 mm) blade that has a slightly thicker ridge at the top to accomplish the same job of providing clearance for the tool while cutting. Parting tools thicker than .040" (1 mm) will be too thick for use on your Sherline lathe.

Boring Tool—A boring tool is used in the tool post on a lathe, or in an offsettable boring head on a mill, to enlarge holes in a work piece. (See Figure 10.)

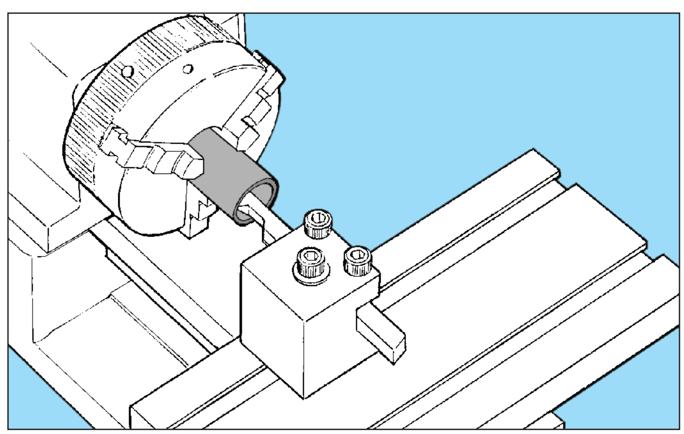


Figure 10—A boring tool in use on the lathe

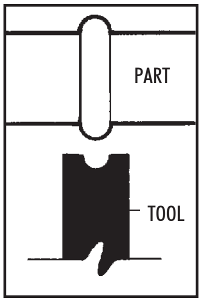


Figure 11—Form tool and part

Form Tool—A custom contour can be ground into a tool to produce a special shape like a radius in a part. The width of the cutting edge must be less than 2-1/2 times the smallest diameter. Cutting speed must be slow to prevent chatter.

The clearances ground behind the cutting edges indicate the type of material for which the tool may be used and the direction in which it is fed along the work. When grinding tool bits, correct clearances are essential or “rubbing” can occur.

The shape shown here would be difficult to grind on a home bench grinder; however, the same form could be achieved by grinding two separate tools with half the needed arc on the outside corner of each tool–a “left” and a “right.” By using a number of simple shaped tools in sequence, complicated forms can be generated.

Turning Tools (Left- and Right-Hand)—Reference to Figure 12 will illustrate the lateral positioning of this tool. Note the clearance behind the point between the end of the tool and the work. Insufficient clearance will cause the tool to “rub,” and excessive clearance will produce a ridged or wavy finish due to the small length of tool edge in contact with the work. This ridging becomes more pronounced with rapid feed. To provide a smooth finish, the sharp cutting point may be slightly rounded with an oilstone, taking care to preserve the side clearance underneath this corner.

This type of tool should not be advanced directly endwise into the work. The depth of cut is set while the tool is clear of the end of the work. The starting procedure is to advance the tool until the point just touches the work. Note the reading on the crosslide handwheel, withdraw the tool slightly and move along until clear of the end of the work. Now advance the crosslide to the above reading, add desired depth of cut and then feed the tool along the work piece the desired distance. Withdraw the tool clear of the work, having noted the reading on the crosslide handwheel. Mentally note the reading on the leadscrew handwheel, return the tool to starting position and advance to the previous reading plus the desired cut.

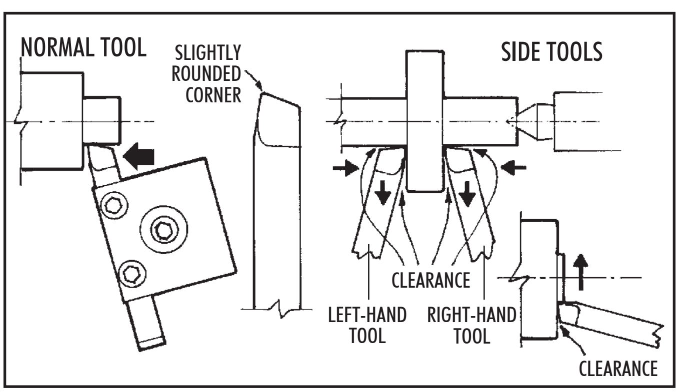


Figure 12—Arrows show direction of tool feed in all diagrams.

NOTE: Sherline offers optional adjustable “zero” handwheels that allow you to reset the handwheel to zero at any time...a handy feature normally found only on larger, more expensive machine tools. New tools may be ordered with them already installed, and existing tools can be retrofitted with them on any axis.

The second feed is now commenced, stopping at the same reading on the leadscrew handwheel as before. This procedure enables turning to accurate length.

Repeat the procedure until the work has been reduced to within about .010" (0.25 mm) of desired diameter, noting that each .015" (0.4 mm) increase in depth of cut will reduce the work diameter by twice this amount; that is, .030" (0.8 mm). For the finishing pass, advance the tool by the required amount and feed along the work just far enough to gauge the finished diameter. Adjust depth of cut if necessary and complete the final pass using a SLOW feed to obtain a smooth finish and exact size.

Using the Cutoff or Parting Tool

(See Figure 13.) After completing a part in the lathe, it is frequently necessary to separate the part from the excess material used for chucking. This operation is best accomplished with the use of a cutoff tool or “parting tool” as it is sometimes called. The Sherline cutoff tool and holder utilizes a very slender, high-speed tool steel cutting blade mounted in a special tool holder. The thinness of the blade (.040") enables it to feed into the part quite easily and at the same time minimizes the amount of waste material. A word of caution: Never use a parting tool on a part mounted between centers. The part may bind on the cutter, resulting in a scrapped part or a broken cutting tool.

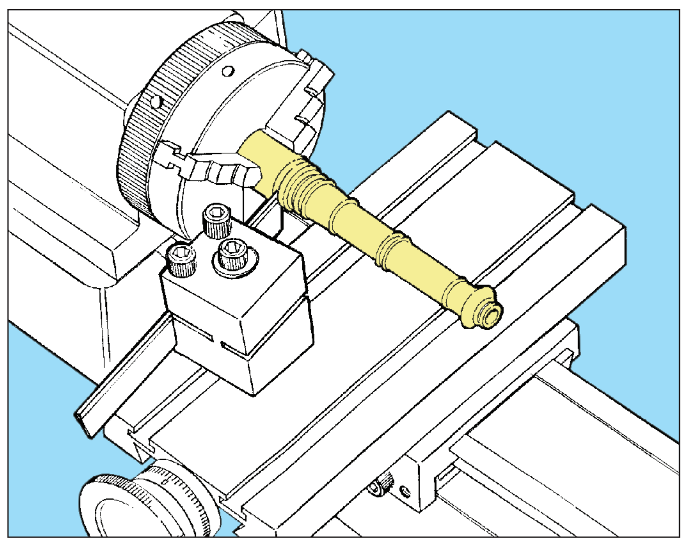


Figure 13—A parting tool used to separate a part from it’s bar stock.

Always try to lay work out so the cutoff tool is used as close to the spindle as possible. Set blade height by sliding the blade back and forth in the slightly angled slot in the tool holder. It should be set so the tip is aligned with the centerline of the part being cut. An unusual diameter may require a shim under the front or rear of the holder to accomplish this. The tool can also be mounted on the back side of the table by using the rear mounting block, P/N 3016.

IMPORTANT!

Always use cutting oil when using the cutoff tool. The cut will be made much smoother, easier and cooler.

The turning speed for parting should be about one-half the normal turning speed, and feed rate should be a little heavy so the chip will not break up in the slot. If speed and feed are correct, there will not be any chatter, and the chip will come out as if it were being unrolled. Cutting oil plays a major roll in this occurring properly.

If the tool chatters, first check to see if the work is being held properly. Then decrease speed (RPM) or increase feed rate or both. Once the blade has chattered, it leaves a serrated finish that causes more chatter. Sometimes a serrated finish can be eliminated by stopping the spindle, adding a liberal amount of cutting oil, bringing the blade up so there is a slight pressure on it without the spindle turning, and then turning the spindle by hand or as slowly as possible with the speed control.

Very small work may be completely cut off when held in a chuck and allowed to fall onto the crosslide. It is too small and light to cause any damage. Hollow articles, such as rings, may be caught on a piece of wire whose end is held in a suitable position.

Side Tools

While these may be, and often are, used as general purpose turning tools, their specific use is for facing the sides of collars and shoulders; that is, finishing these to correct dimension and with a smooth, flat surface. They are also for facing work held on a faceplate or in a chuck. The facing of work in this manner is very useful for the production of truly flat surfaces and for producing articles to an exact thickness. The uses of side tools are illustrated in Figures 9 and 12. The sharp corner at the cutting point should not be slightly rounded, as may be done with the normal turning tool, as knife tools may be required to produce sharp corners.

Boring Tools

The use of this tool requires the existence of a drilled or cored hole, or it may be used to enlarge the bore of a tube. The work must be mounted in a chuck or on a faceplate and the boring tool set as shown in Figure 10. Note the clearance behind the cutting point as shown in Figure 14 below.

A slow rate of feed should be used, as the turnings are not able to escape freely from the hole and can jam the tool. Frequent withdrawal of the tool to allow turnings to escape may be necessary. Care should be taken not to feed the tool beyond the depth required or to feed so deeply as to damage the chuck or faceplate.

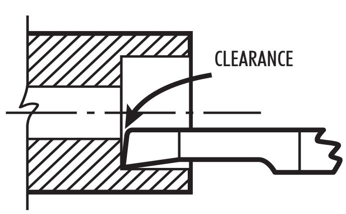


Figure 14—Boring tool clearance

Where a hole must be bored right through the work, it should be shimmed out from the faceplate to provide clearance for the tool to feed through. The leadscrew handwheel graduations can be used to indicate the correct depth at which to stop the feed. Notice that, with boring, the depth of cut is increased by moving the tool and crosslide towards the operator and not away as with normal turning.

The boring of holes often necessitates greater than normal overhang of the tool from the tool post, so the depth of cut and rate of feed should be reduced from normal.

Inserted Tip Carbide Tools

Sherline brings the home shop machinist into the space age with cutting tools that add a new dimension to small lathes. When working with tough metals, high-speed steel tools need constant sharpening and have a relatively short life. Brazed carbide tools cut great but chip easily. Inserted carbide cutting tools are the answer and have replaced those other tools in the modern machine shop. Carbide inserts have the ability to consistently give good finishes and long tool life at a much higher cutting speed. This is especially important with small lathes, because they do not have excessive power at low RPM. With inserted carbide tools you can cut stainless steel at the same RPM you were formerly using to cut aluminum with high-speed steel tools without any sacrifice in quality in surface finish.

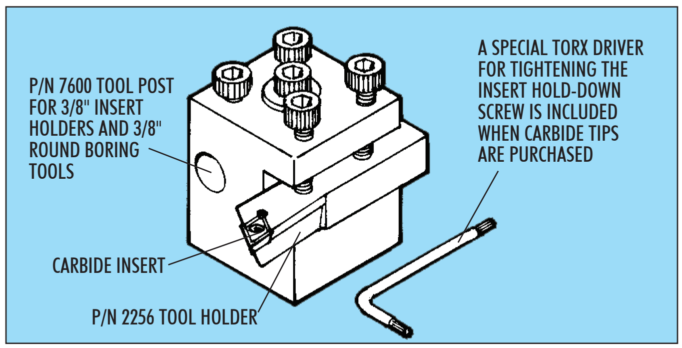


Figure 15—Carbide insert tool and tool post. The tool post holds both 3/8" square and round tools.

These tools are more expensive than high-speed steel; however, they are worth every penny if you have problems grinding your own steel tools or are cutting exotic materials like stainless steel. Sherline offers a tool post (P/N 7600) that holds the larger 3/8" square tool shanks used to hold carbide, ceramic or diamond inserted tips. It also has a 3/8" round hole for boring tools.

A good starting point for an inserted tip tool is the P/N 2256 right-hand holder with a 35° offset. This holder uses the P/N 7605 carbide insert, which is a 55° insert good for turning, facing and profiling. A left-hand tool is also available as P/N 2257, or a set of both left- and right-hand tools is P/N 2258. Tools are also available to hold 80° inserts, which are slightly less versatile but offer longer tool life because of their stronger, more square shape. These tools should not be used to cut hardened steels or piano wire. Materials such as those are normally ground to shape, not cut, although ceramic inserts can sometimes be employed to cut these materials. Abrasive materials such as glass-reinforced plastics can be easily cut with these tools.

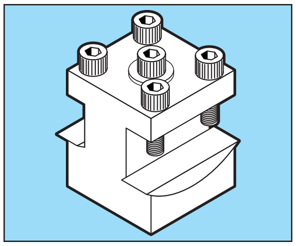


FIGURE 16—Two-Position Rocker Tool Post (P/N 7603)

Another tool available to Sherline machinists that holds carbide inserts is the 1/4"–3/8" two-position rocker tool post, P/N 7603 (See Figure 47). This tool post has slots on two opposite sides to hold both 1/4" and 3/8" square shank tools individually or at the same time. This allows you to switch quickly between tools of the two different sizes simply by rotating the tool post. The 3/8" side is designed to fit the larger 3/8" square tool holders commonly used for carbide or diamond inserted tips. Adding this tool post to your arsenal will allow you to keep both your standard 1/4" high-speed steel tools set up for jobs where they are sufficient and also have a 3/8" carbide insert tool ready for jobs where it is required.

Sherline also offers a ceramic insert and holder, P/N 2265. (See Figure 17.) The 3/8" IC negative rake ceramic indexable holder will bring a lot of enjoyment to your machining, particularly if you choose to turn hard materials such as tool steel or abrasive materials like fiberglass or composites.

When searching for a mirror-like finish on copper or aluminum, diamond inserts are also available. Though expensive, certain jobs can make their use desirable.

While inserted tip carbide, ceramic and diamond cutting tools will improve the performance of the Sherline lathe, they will not correct poor machining technique. Rigid setups are a must for tools such as these.

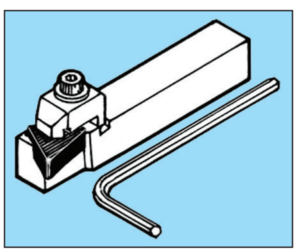


Figure 17—The P/N 2265 negative rake ceramic insert and 3/8" holder make it possible to cut hardened tool steels.

Turning Speeds

The following chart in Figure 18 provides a guide to speeds at which work of differing materials should be rotated. Note that the turning speed is inversely proportional to the diameter of the work; that is, the larger the diameter, the slower the turning speed. Material often differs in hardness, so these figures may have to be adjusted. The harder the material, the slower the turning speed should be.

Keep in mind that, apart from possible production of excessive heat and the fact that excessive speed may damage the cutting edge or cause it to "rub" instead of cutting, turning speeds are not too critical. Slower than normal speeds cause no harm, except by increasing the time involved. Aluminum, however, usually gives a better finish turned at high speed and with the use of lubrication (coolant).

Guide to Approximate Turning Speeds

Material	Cut Speed S.F.M.	1/4" (6mm) Diameter	1/2" (13mm) Diameter	1" (25mm) Diameter
Stainless, 303	67	1000 RPM	500 RPM	250 RPM
Stainless, 304	50	800	400	200
Stainless, 316	47	700	350	175
Steel, 12L14	174	2600	1300	650
Steel, 1018	87	1300	650	300
Steel, 4130	82	1250	650	300
Gray Cast Iron	57	900	450	220
Aluminum, 7075	400	2800	2800	1400
Aluminum, 6061	375	2800	2800	1400
Aluminum, 2024	268	2800	2000	1000
Brass	400	2800	2800	1400

Figure 18—High-speed steel cutting tool turning speeds

Accessories for Your Lathe

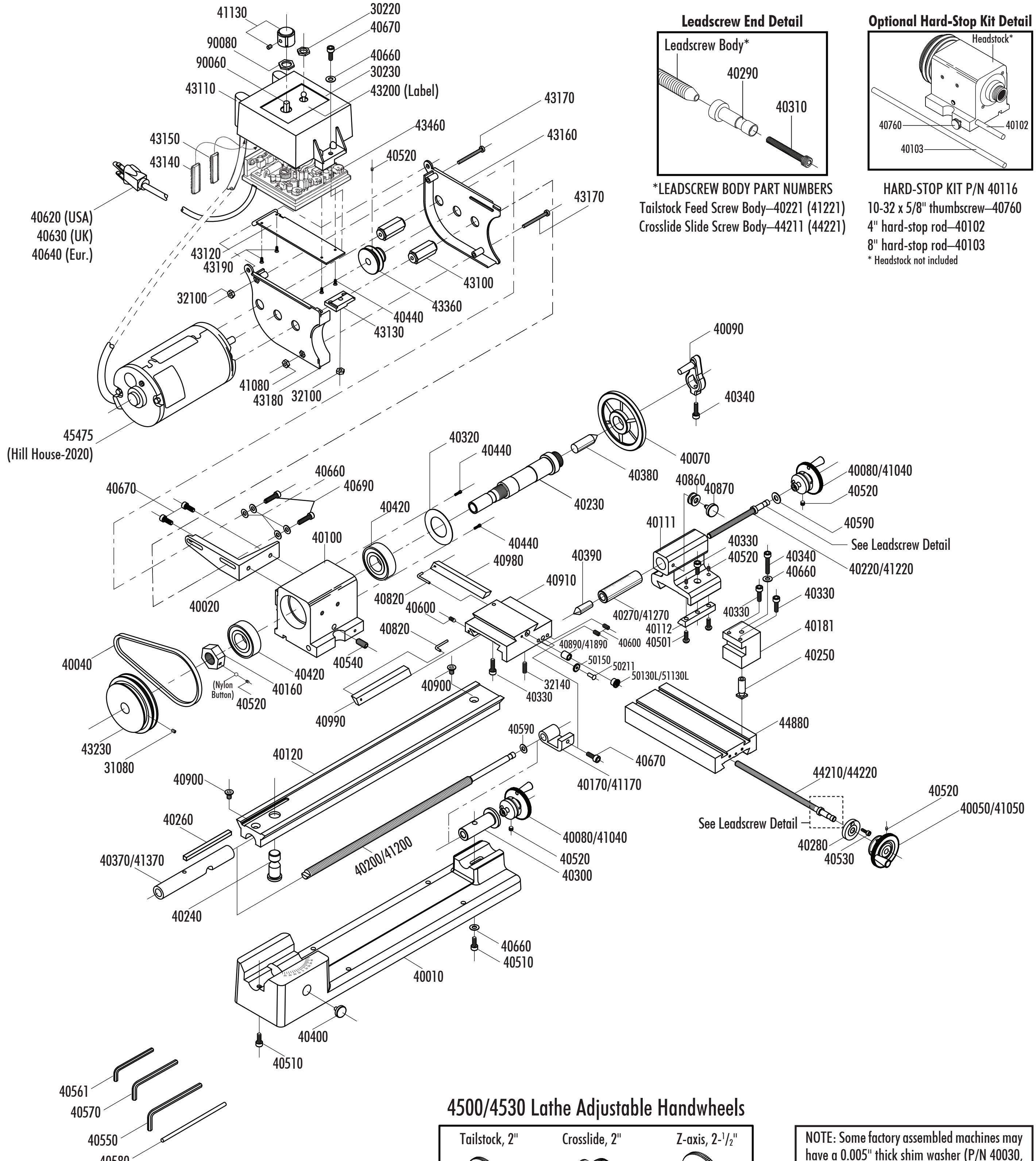
Your lathe can be made more versatile with the addition of suitable attachments and accessories. These include various chucks and collets, a thread-cutting attachment, vertical milling column, knurling tool, a live center and many others. Remember that accessories and attachments must be cared for in the same way as the lathe. Always make sure that threads are free from metal chips and dirt. Chucks should be lightly oiled frequently so that they continue to function smoothly and accurately. Gears in the thread-cutting attachment should be lightly greased when in operation. Some attachments have moving slides, and these should be lubricated in the same way as the slides in your lathe. Each accessory comes with complete instructions for its use when it is purchased.

Learning about other Accessories for Your Lathe

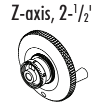
The best place to learn about Sherline accessories is on our website. A complete list of accessories with links to instructions is on our website at **Accessory Instructions**. If you do not have an Internet connection, Sherline offers a collection of printed instructions called the *Sherline Accessories Shop Guide*, P/N 5327. You can request a color catalog featuring the tools and accessories by calling (800) 541-0735 or (760) 727-5857.

Sherline 4000 and 4500 Lathe Exploded View and Part Numbers

NOTE: Where different, Inch part number is given first, followed by Metric part number.



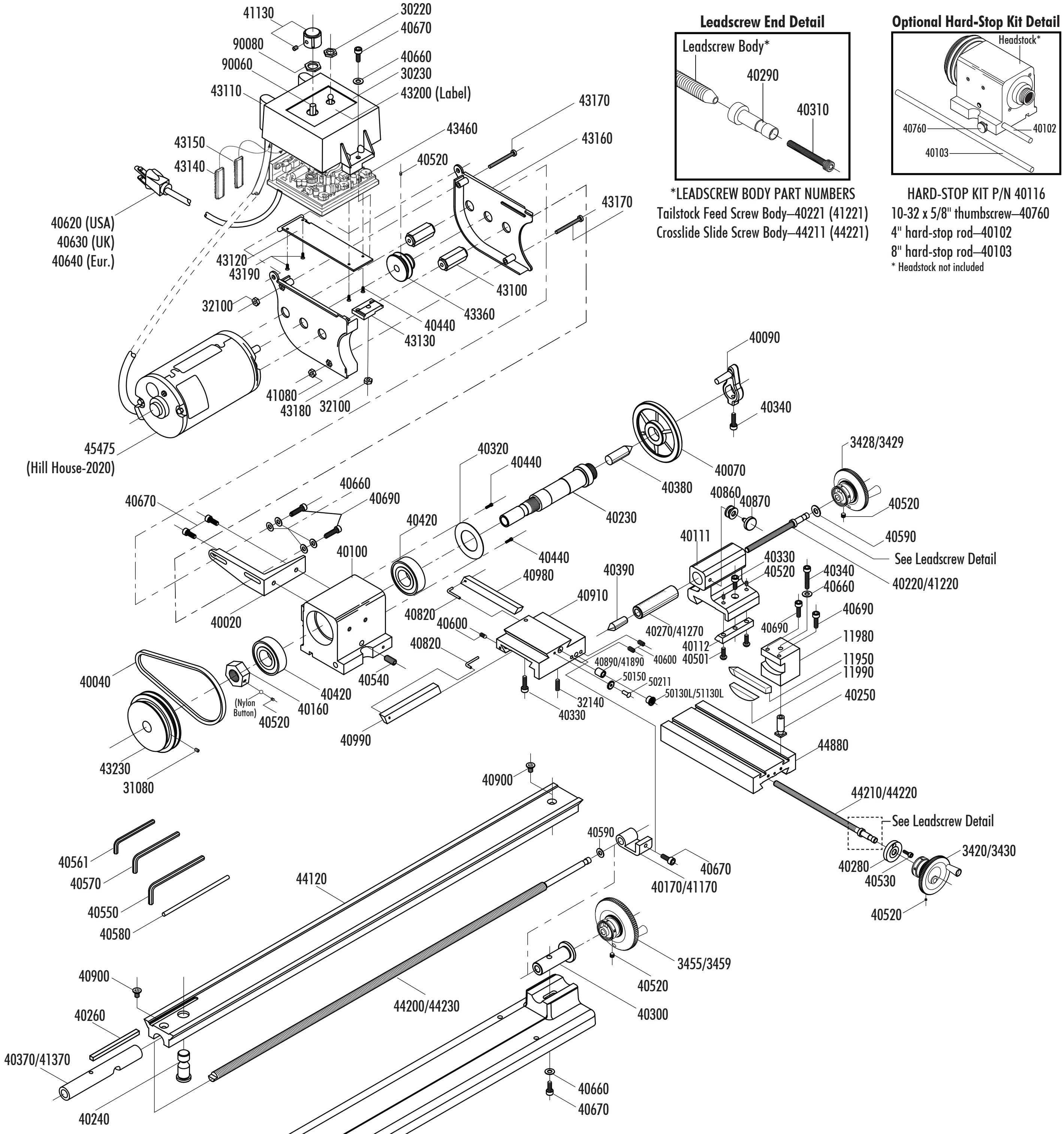
4500/4530 Lathe Adjustable Handwheels

Tailstock, 2"	Crosslide, 2"	Z-axis, 2-1/2"
		
P/N 3428/3429	P/N 3420/3430	P/N 3455/3459

NOTE: Some factory assembled machines may have a 0.005" thick shim washer (P/N 40030, not shown) placed between the handwheel and thrust to adjust backlash. If removed when servicing the machine they should be replaced in the same position.

Sherline 4400 Lathe Exploded View and Part Numbers

NOTE: Where different, Inch part number is given first, followed by Metric part number.



NOTE: Some factory assembled machines may have a 0.005" thick shim washer (P/N 40030, not shown) placed between the handwheel and thrust to adjust backlash. If removed when servicing the machine they should be replaced in the same position.

SHERLINE Lathes and Mills

KEY TO MATERIALS: A=Aluminum, B=Brass, C=Composite, DC=Die Cast, P=Plastic, U=Urethane, S=Steel

PART NO.	DESCRIPTION	MATERIAL
11950	1/4" HSS cutting tool (RH)	S
11980	Rocker Tool Post Body	A
11990	Rocker Tool Post Rocker	S
30220	Toggle Switch Retaining Ring	S
30230	Toggle Switch	—
31080	10-32 x 3/8" Flat Pt. Set Screw	S
34000 (34100)	Oversize Handwheel, Inch (Metric)	A
34060	Thrust Bearing Washer Set	Ball
34210	2" handwheel Body	A
34220	Handwheel Locking Nut	S
34230 (34240)	Y Axis/Crosslide Collar, Inch (Metric)	A
34250	6-32 x 7/8" Pan Hd. Screw	S
34260 (34270)	X, Z Axis and Leadscrew Collar, Inch (Metric)	A
34410	2-1/2" Handwheel Body	A
40020	Motor Bracket	DC
40040	Drive Belt	U
40050 (41050)	1-5/8" Handwheel, Y Axis/Crosslide, Inch (Metric)	A
40070	Faceplate	DC
40080 (410400)	1-5/8" Handwheel, X Axis/Leadscrew, Inch (Metric)	A
40090	Drive Dog	DC
40100	Headstock Casing	A
40111	Tailstock Casing (Gib style)	A
40112	Tailstock Gib	B
40120	15" Lathe Bed	DC
40160	Preload Nut	S
40170 (41170)	Saddle Nut, Inch (Metric)	B
40180	Tool Post	A
40200 (41200)	Tailstock Feed Screw (complete), Inch (Metric)	S
40221 (41221)	Tailstock Feed Screw body only, Inch (Metric)	S
40220 (41220)	Feed Screw, Inch (Metric)	S
40230	Headstock Spindle	S
40240	Headstock Pivot Pin, Lathe	S
40250	Tool Post Tee Nut	S
40260	Head Key	S
40270 (41270)	Tailstock Spindle, Inch (Metric)	S

PART NO.	DESCRIPTION	MATERIAL
40280	Thrust Collar	S
40290	Leadscrew end	S
40300	Leadscrew Thrust	S
40310	5-40 x 1" SHCS	S
40320	Bearing Washer	S
40330	10-32 x 5/8" Skt. Hd. Cap Screw	S
40340	10-32 x 1" Skt. Hd. Cap Screw	S
40345	(See 45450)	—
40370	Leadscrew Support	S
40380	#1 Morse Center	S
40390	#0 Morse Center	S
40400	Plug Button	P
40420	Headstock Bearing	Ball
40440	Self Tapping Screw	S
40500	10-24 x 7/8" Skt. hd. Cap Screw	S
40501	10-32 x 1/2" Button Head Hex Screw	S
40510	10-32 x 3/8" Skt. Hd. Cap Screw	S
40520	10-32 x 3/16" Cup Pt. Set Screw	S
40530	5-40 x 3/8" Skt. Hd. Cap Screw	S
40540	5/16-18 x 3/4" Cone Pt. Set Screw	S
40550	5/32" Hex Key	S
40560	3/16" Hex Key	S
40570	3/32" Hex Key	S
40580	Spindle Bar	S
40590	1/4" I.D. Washer	S
40600	10-32 x 1/4" Flat Pt. Set Screw	S
40620	Power Cord, USA	—
40630	Power Cord, UK	—
40640	Power Cord, Europe	—
40660	3/16" I.D. Washer	S
40670	10-32 x 1/2" Skt. Hd. Cap Screw	S
40690	10-32 x 3/4" Skt. Hd. Cap Scrw	S
40760	10-32 x 5/8" Thumbscrew	S
40820	Gib Lock	S
40860	Tailstock Locking Screw Grommet	P
40870	Tailstock Spindle Locking Screw	S
40890 (41890)	Slide Screw Insert, Inch (Metric)	B
40900	10-32 x 3/8" Flat Hd. Skt. Screw	S
40910	Saddle	A
40980	Crosslide Gib	C

PART NO.	DESCRIPTION	MATERIAL
40990	Saddle Gib	C
41080	6-32 Hex Nut	S
41110	Tailstock Casing	A
41130	DC Speed Control Knob and Set Screw	P/S
43100	DC Motor Standoff	A
43110	DC Speed Control Case	P
43120	DC Speed Control Hinge Plate	P
43130	DC Speed Control Cover Mounting Plate	P
43140	DC Speed Control Tab, Small	P
43150	DC Speed Control Tab, Large	P
43160	Belt Guard, Outer	P
43170	6-32 x 1-3/8" Pan Hd. Screw	S
43180	Belt Guard, Inner	P
43190	#2 x 1/4" Flat Hd. Sheet Metal Screw	S
43200	DC Speed Control Foil Label	Foil
43230	Stepped Main Spindle Pulley	A
43360	Stepped Motor Pulley	A
43450	(See P/N 45450)	Motor
43460	DC Speed Control Electronics	—
44010	24" Lathe Base	DC
44120	24" Lathe Bed	S
44200 (44230)	24" Leadscrew, Inch (Metric)	S
44210 (44220)	Slide Screw, Inch (Metric)	S
44211 (44221)	Slide Screw body only, Inch (Metric)	S
44880	Crosslide	A
45010 (45160)	Leadscrew, Z Axis, Inch (Metric)	S
45040	Saddle, Z Axis	A
45170	Column Saddle Lock	P
45180	3/16" Ball Bearing	S
45190	#10 Type B Washer	S
45450 or 45460	45450=DC Motor with externally replaceable brushes (Leeson) 45460=DC Motor with externally replaceable brushes (Hill House) (**NOTE:** We purchase motors from two different manufacturers to keep pricing competitive. Specifications on both are the same, but replacement motors should be ordered with the same part number as the original. Part number is printed on motor.)	Motor
90060	DC Speed Control 5K Potentiometer	—
90080	3/8-32 Hex Nut	S