Smithy®

CB-1239 XL Lathe-Mill-Drill Operator's Manual

Installation, Operation, and Maintenance

Smithy Co.
170 Aprill Drive
PO Box 1517
Ann Arbor, Michigan 48106-1517
(800) 298-5111

Contents

SECTION ONE
INTRODUCTION

Congratulations on purchasing a Smithy CB-1239 XL lathe-mill-drill. We are pleased you chose Smithy to fulfill your machining needs.

The purpose of this manual is to give the machinist, beginning or advanced, the information he needs to operate the Smithy CB-1239 XL. It will teach you about the machine's parts and how to care for them. In fact, education is our primary goal. We'll explain how to grind cutters, set up tools, hold workpieces, and do all basic machining operations.

Please read this Operator's Manual carefully. If you don't understand how the CB-1239 XL works, you may damage your machine, your project, or yourself. If you want to learn more about machining practices, Smithy offers books that meet the needs of machinists at all levels of experience. We also suggest using your local library as a resource. Enrolling in a machining class will give you the best knowledge of machining.

If you have any questions not covered in this manual, please call Smithy. Our trained technicians will help you with any machining problems you run into. Dial our toll-free number—1-800-298-5111—Monday through Friday, 8:00 am to 5:00 pm Eastern Time.

We are always interested in your suggestions to improve our products and services. Feel free to contact us by phone or in writing. If you have comments about the CB-1239 XL Operator's Manual, or if you have any projects you'd like to share with other Smithy owners, contact the Education Manager, Smithy Co., 170 Aprill Drive, PO Box 1517, Ann Arbor, MI 48106-1517.

We look forward to a long working relationship with you. And thanks again for putting your trust in Smithy.

1.1 Customer information

This manual should remain with the CB-1239 XL. If ownership changes, please include the owner's manual with the machine.

Model # ______________________________

Serial # ______________________________
(on the front of the headstock)

Purchase date ______________________________

Delivery date ______________________________

Salesperson ______________________________

1.2 CB-1239 XL specifications[1]

General Dimensions

67″ long × 26″ wide × 36″ high
Shipping weight: 1300 lb
Machine weight: 880 lb
Power: 110 V
Mill/drillhead swivels 360°
Calibrated dials
- Leadscrew: 0.015″
- Cross slide: 0.002″
- Tailstock: 0.001″
- Mill and drill: 0.040″

Lathe

Swing — Over bed: 12″
Over table: 6-1/2″
Distance between centers: 39″
Spindle bore: 1.5″
Taper — Headstock: Morse taper #5
Tailstock: Morse taper #3
Speeds — Lathe: nine (250 –1600 rpm)
Powerfeed: two
Pitches — Inch-thread: 20 (11 – 40 tpi)
Metric: 14 (0.5 – 3 mm)
Hardened, ground dovetail ways
6″ three-jaw, self-centering chuck with inside and outside jaws
Travel — Cross slide: 8″
Longitudinal: 39″

Mill

Millhead taper: Morse taper #3
Spindle nose to table: 4-1/4 – 15″
Spindle to support column: 13″
Table: 8″ wide × 17″ long
T-slot width: 7/16″ (12 mm)
Speeds: nine (315 – 2000 rpm)
Quill travel: 5″
Drawbars Standard: 3/8 – 16 tpi
Metric: 12 mm – 1.75-mm pitch

Drill

1/2″ precision chuck
Chuck arbor taper: Morse taper #3
Drill chuck to table: 1-1/4 – 12″
Quill travel: 5″
Spindle to support column: 13″
Drill speeds: nine (315 – 2000 rpm)
Calibrated dials[2]
- Leadscrew: 0.015″
- Cross slide: 0.002″
- Mill/drill press: 0.040″

---

[1] Specifications are subject to change.

[2] Calibrated dials are a measuring guide. To assure accuracy, use measuring instruments.

SECTION TWO
SAFETY

Your workshop is only as safe as you make it. Take responsibility for the safety of all who use or visit it. This list of rules is by no means complete—you must add common sense to make your workshop safe and enjoyable.

• *Know your machine.* Read this manual thoroughly before attempting to operate the lathe-mill-drill. Don't try to do more than you or your machine can handle. Understand the hazards of operating the CB-1239 XL. In particular, remember these two warnings: never change speeds or setups until the machine is completely stopped, and never operate it without first rolling up your sleeves or tying them at your wrists. Open or frayed sleeves can catch in revolving parts, causing serious accidents.

• *Ground the machine.* The CB-1239 XL has a three-conductor cord and a three-prong grounding-type receptacle. Never connect the power supply without properly grounding the machine.

• *Remove all adjusting keys and wrenches before operating the machine.* A chuck key or misplaced Allen wrench can be a safety hazard.

• *Keep your work area clean and organized.* Cluttered work areas and benches invite accidents. Have a place for everything and put everything in its place.

• *Keep children away while machine is in use.* Childproof your shop with padlocks, master switches, and starter keys, or store the machine where children do not have access to it.

• *Wear appropriate clothing.* Avoid loose-fitting clothes, gloves, neckties, or jewelry that could get caught in moving parts. If you have long hair, tie it up or otherwise keep it from getting into the machine.

• *Use safety glasses, goggles, or a face shield at all times.* Use glasses designed for machinery operation; regular glasses are not adequate. Have extras available for visitors. Know when to wear a face mask. Noise may be an irritant, as well, so wear earplugs when necessary.

• *Check for damaged parts.* Make sure the machine will run properly before operating it.

• *Disconnect the machine before servicing and when changing accessories.* Shut power off before making changes, removing debris, or measuring work. Don't reach over the machine when it's in operation. Keep your hands out of the way.

• *Avoid accidental starts.* Turn switches to Off before plugging in the machine.

• *Secure your work.* Flying metal is a dangerous projectile. Loose work can also bind tools, damaging the tools, the machine, your workpiece, and/or you.

• *Use the recommended accessories.* Understand how to use the three and four-jaw chucks, faceplate, etc., before trying them out.

• *Use the correct tool for the job.* Don't try to make a tool into something it isn't.

• *Keep your mind on your work.* Pay attention to these simple rules and you will spend many safe, enjoyable hours in your workshop.

Remember: your safety depends largely on your practices.[3]

---

[3]Modifying your machine may void the warranty and create potential hazards.

SECTION THREE
CARING FOR THE CB-1239 XL

The CB-1239 XL is a delicate, precision tool with hardened ways and hand-scraped bearing surfaces under the table and carriage. Any rust spot or battering of the ways, any chips or grit between close-fitting parts, is certain to affect the accuracy of this fine tool. Follow these guidelines whenever you use your Smithy machine:

- When you finish working, wipe machined surfaces of the CB-1239 XL with an oily rag. Never leave it without this thin film of protective oil over all parts that might rust, especially ground finished parts.

- Never lay wrenches, cutting tools, files, or other tools across the ways of your lathe. The slightest dent or burr will impair its accuracy.

- Before inserting collars, centers, adapters, or drawbar attachments in either the spindle or tailstock spindle, wipe them with a clean, oily rag. Also, wipe all internal surfaces carefully with an oily rag on a ramrod. Chips or dirt on the centers or in the spindle nose can scratch or mar surfaces and interfere with the assembled part's alignment.

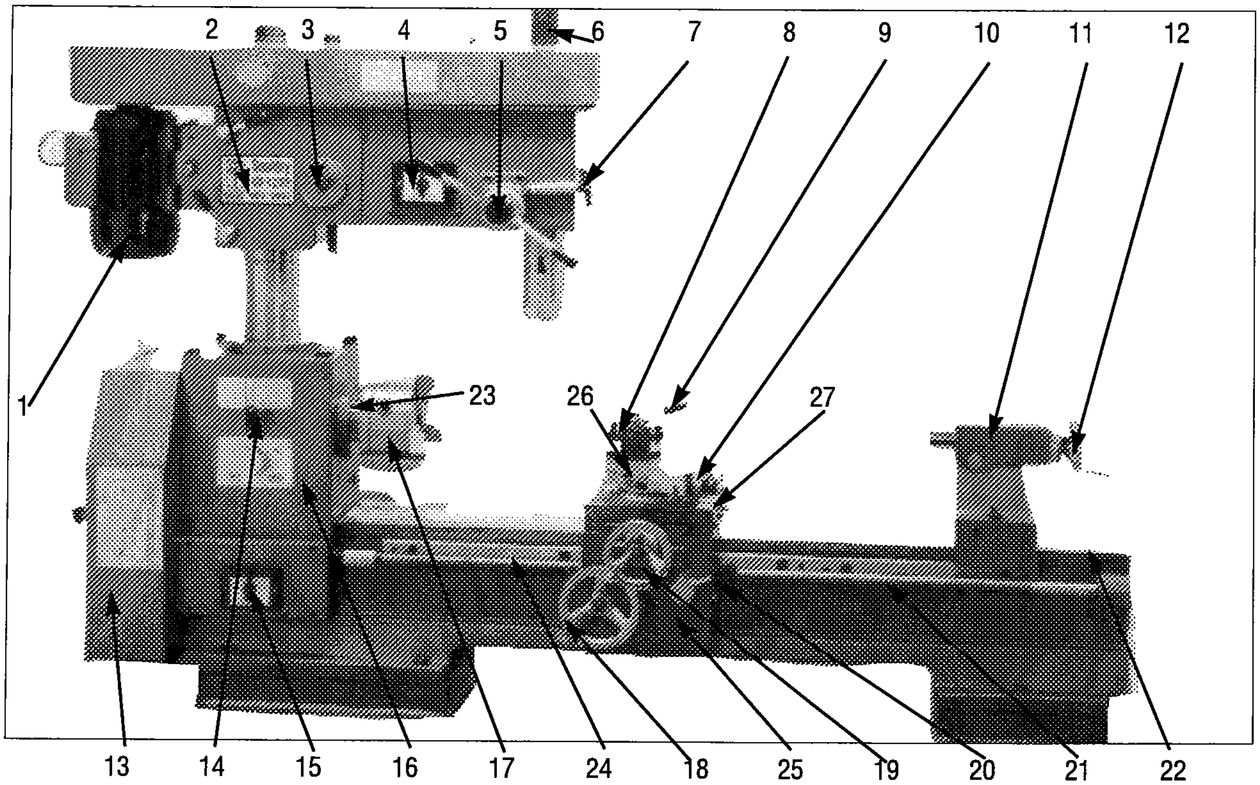
- Lubricate the machine before each use (see *Section* 5.4).

SECTION FOUR

BASIC PARTS OF THE CB-1239 XL

To learn the operations of the CB-1239 XL, you must understand the functions of its basic units (*Figure* 4.1).

4.1 Major controls of the CB-1239 XL



1 Millhead motor
2 Millhead
3 Millhead height adjuster
4 Mill Forward/Off/Reverse switch
5 Fine-feed engage clutch
6 Mill spindle
7 Fine-feed handwheel
8 Toolpost turret
9 Toolpost clamping lever
10 Compound slide
11 Tailstock
12 Tailstock handwheel
13 Gearbox
14 Speed selector
15 Lathe Forward/Off/Reverse switch
16 Headstock
17 Lathe chuck
18 Carriage handwheel
19 Cross-slide handwheel
20 Half-nut lever
21 Leadscrew
22 Lathe bed
23 Lathe spindle
24 Rack
25 Apron
26 Compound rest
27 Cross slide

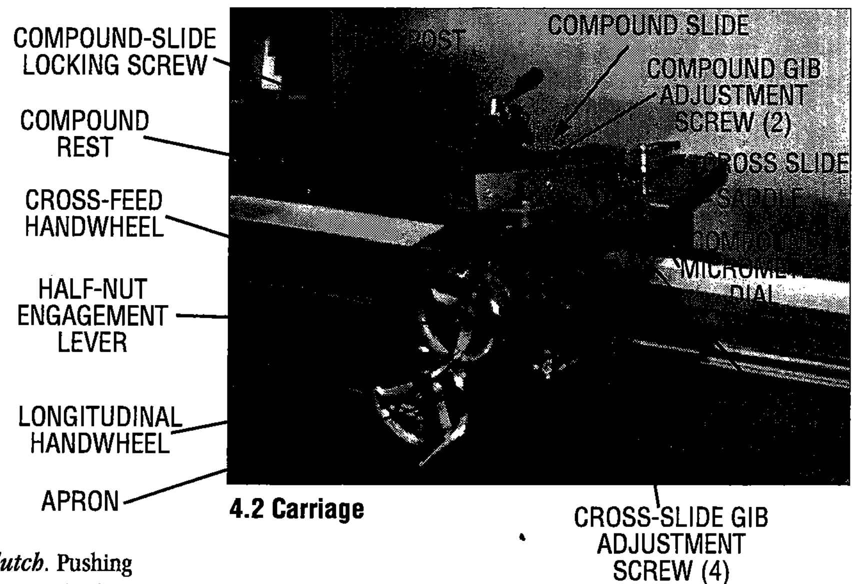
• *Apron.* The apron (*Figure* 4.2), the part of the carriage facing the operator, contains longitudinal handfeeds, cross-feed handwheel controls, and half-nut engagement lever.

• *Bed.* The bed (*Figure* 4.1) is the machine's foundation. It is heavy, strong, and built for absolute rigidity. The two ways on the top are the tracks on which the carriage and tailstock travel. To maintain an exact relationship between toolpoint and workpiece from one end of the machine to the other, the ways must be absolutely true and accurately aligned to the line of centers and to one another.

• *Carriage.* The carriage (*Figure* 4.2) consists of the saddle and apron. It moves by hand or power along the bed, carrying the cross slide, compound rest, and toolpost. Its function is to support the cutting tool rigidly and move it along the bed for different operations. It locks by tightening the carriage lock.

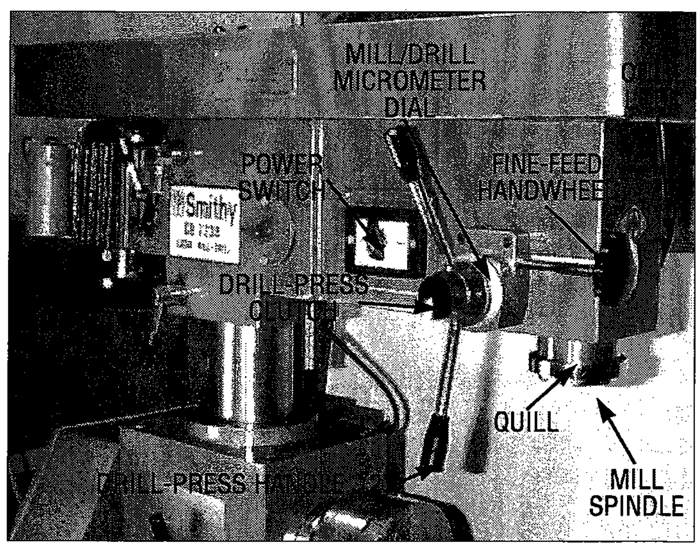
• *Compound rest.* Mounted on the cross slide, the compound rest (*Figure* 4.2) swivels to any angle horizontal with the lathe axis to produce bevels and tapers. Cutting tools fasten to a toolpost on the compound rest. The calibrations on the front of the base are numbered in degrees from 55° right to 55° left.

• *Cross slide.* The T-slotted cross slide (*Figure* 4.2) moves crosswise at 90° to the lathe axis by manual turning of the cross-feed screw handle. It also serves as the milling table.



4.2 Carriage

• *Drill press and fine-feed clutch.* Pushing in the drill press clutch (*Figure* 4.3) engages the fine feed. To work the clutch, release the spring tension by rotating the drill press handles clockwise. Pull the clutch out to use it as a drill press or push it in to use the fine feed. Use the fine-feed handwheel to move the quill up and down.



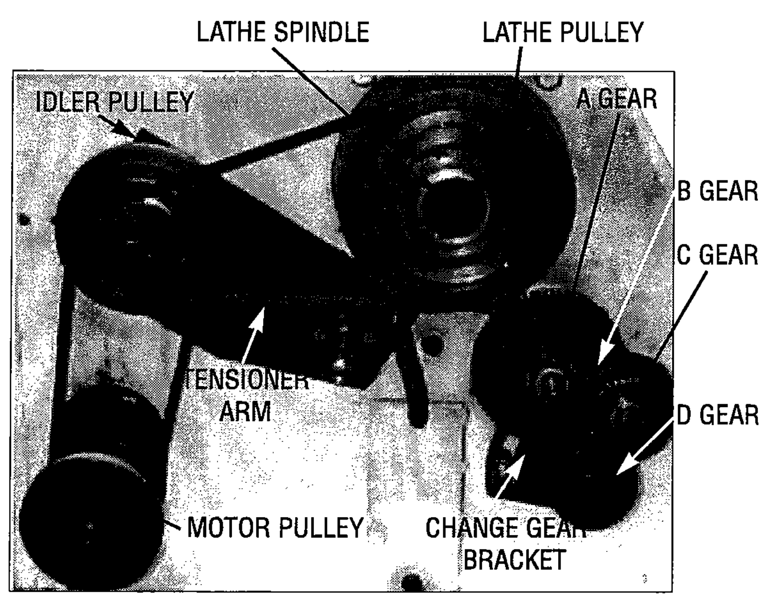
4.3 Mill/Drill Quill

• *Gearbox.* The gearbox (*Figure* 4.4) houses the belts that drive the spindle and the change gears for the powerfeed.

• *Headstock.* The headstock (*Figure* 4.5), which is secured to the bed, houses the gears that drive the leadscrew.

• *Lathe belt tensioner.* To adjust the lathe belt tensioner (*Figure* 4.6), turn the knurled knob clockwise to tighten the belt, counterclockwise to loosen it.

• *Lathe spindle.* The end of the lathe spindle (*Figure* 4.4) facing the tailstock is the spindle nose. The spindle nose, which has an MT5 taper, drives lathe chucks and other workholding devices and rotates the workpiece. The spindle flange (*Figure* 4.7) is bolted to the spindle nose. All attachments (three-jaw chuck, four-jaw chuck, faceplate, etc.) bolt to the flange either directly or via an adapter plate.



4.4 Gearbox

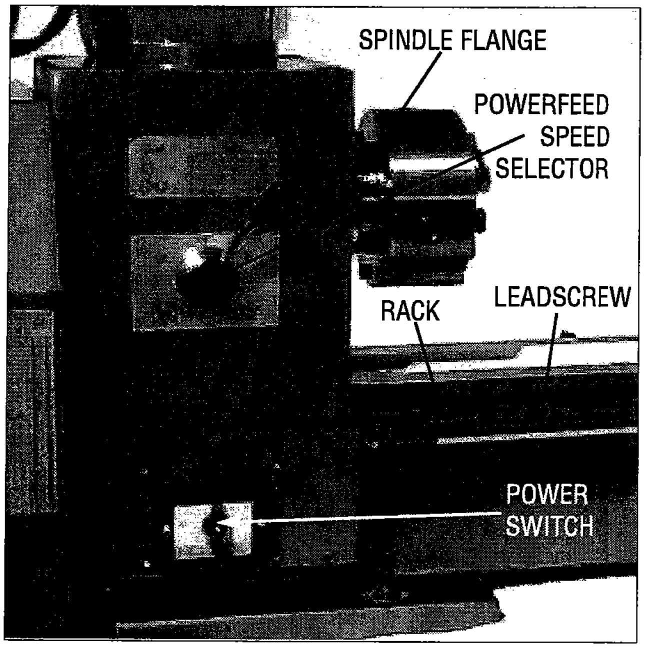
• *Leadscrew.* The leadscrew (*Figure* 4.5) runs the length of the bed, and its external thread is cut with great precision. The leadscrew moves the carriage for thread cutting or lathe turning. It can be used with the mill as well.

• *Locks.* Locks on the cross slide

(*Figure* 4.2), carriage (*Figure* 4.2), quill (*Figure* 4.3), and tailstock (two), *Figure* 4.8, keep them from moving. During machining, lock all axes except the one you want to move.

• *Micrometer control and calibration*. Just inside the handles of the tailstock (*Figure* 4.8), crossfeed, drill press (*Figure* 4.3), compound slide (*Figure* 4.2), and leadscrew are collars calibrated in inches. The dial on the compound feed is also calibrated in millimeters. The compound feed and crossfeed are calibrated in two thousandths, the tailstock in thousandths, the long feed in fifteen thousandths, and the drill press in forty thousandths.

These micrometer dial collars can move independently around the handle shafts. This independent motion is called *float*. The CB-1239 XL has floating dials on the cross slide, tailstock, and long-feed mill/drill. They let you zero the collars at any point and read the feed travel from that point on the dial for increased accuracy.

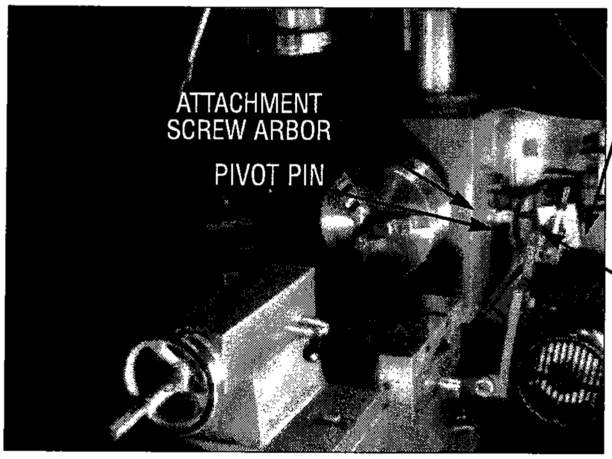


4.5 Headstock

• *Mill-belt tensioner*. To adjust the mill-belt tensioner (*Figure* 4.9), loosen the tensioner lock handle. Tighten the handle to hold it in place.

• *Millhead height adjuster*. Unlock the two millhead locks (top and bottom locks at the right of the mill motor), then use the height adjuster (*Figure* 4.9) to raise and lower the mill-head. Middle lock is for the motor.

•*Mill/drill motor.* Located on the mill-head and controlled by a three-way switch, the mill/drill motor (*Figure* 4.9) powers the spindle. The motor lock and tensioner are next to the motor.



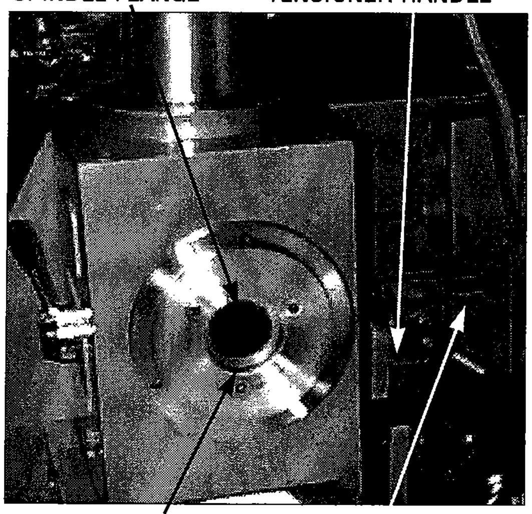
4.6 Lathe belt tensioner

• *Mill spindle*. The mill spindle (*Figure* 4.3) gets power from the motor. It attaches to the quill, which moves in and out of the head. The quill lock keeps the spindle still when you install or remove tools from it. Usually, tools fit into collets that attach through the spindle via drawbars.

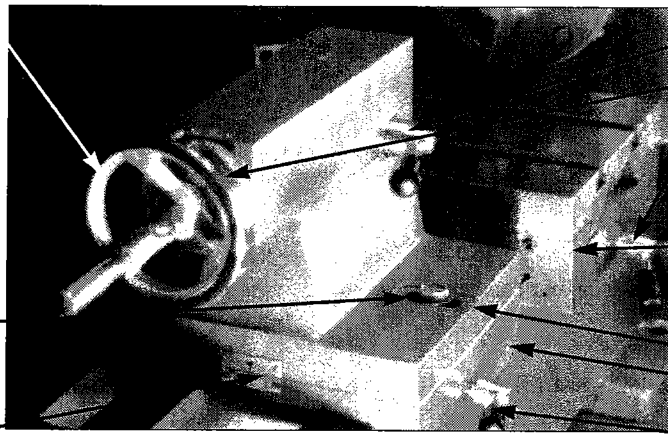
• *Powerfeed speed selector*. The two-speed selector (*Figure* 4.5) for powering the leadscrew is on the front of the head-stock. Speed II turns the leadscrew twice as fast as Speed I. The speed selector does not affect the lathe spindle speed, which is controlled by the belts in the gearbox.

• *Rack*. Running the length of the bed, the rack is geared to the carriage handwheel in front of the apron (*Figure* 4.5). The gears that link the carriage handwheel to the rack allow manual lengthwise movement of the carriage.

• *Saddle*. The saddle (*Figure* 4.2) slides on the carriage ways, supporting the cross slide and compound rest. To move it, turn the handwheel geared to the rack.



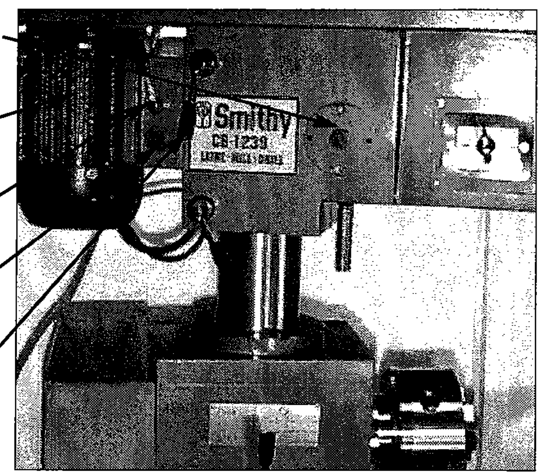
4.7 Mill spindle



4.8 Tailstock

• *Tailstock*. The tailstock (*Figure* 4.8), which provides right-end support for the work, moves along the bed and can stop at any point on it. It holds centers, drills, reamers, taps, and other tools. To move the tailstock spindle, which has an MT3 taper, turn the tailstock handwheel. The scale of offset calibrations on the back of the tailstock is in millimeters.

To offset the tailstock, loosen the two base locking bolts (*Figure* 4.8). To offset to the left, loosen the left adjusting bolt and tighten the right. To offset to the right, loosen the right adjusting bolt and tighten the left. When adjustment is complete, tighten the base locking bolts.



4.9 Millhead

SECTION FIVE
UNCRATING AND SETTING UP THE CB-1239 XL

5.1 Transporting the machine

Moving a machine tool is a potentially dangerous operation. Improper techniques and methods can cause personal injury and/or damage the machine. To find a professional to move and site your CB-1239 XL, look in your local Yellow Pages under "Machine Tools, Moving and/or Rigging." If there is no such listing or your community does not have a rigging specialist, a local machine shop or machinist may be able to provide a referral.

When you pick up the machine at the shipping terminal, bring a crowbar, tin snips for cutting the metal straps, and a hammer. If there is obvious shipping damage to the crate, you'll be able to inspect the machine before signing for it. Note any damage on the bill of lading (shipping document). Fill out the claims forms and notify both Smithy Co. and the shipping terminal about the damage. Failure to notify both parties can complicate and/or invalidate any claims process.

Trucking company terminals usually have forklifts to assist customers. Trucks without canopies and large vans are the most convenient ways to transport the machines.

5.2 Uncrating and positioning the machine

The CB-1239 XL is assembled, inspected, and ready to go on its stand. It's wrapped in a water and greaseproof cover, strongly braced, and crated. A box of accessories is also in the crate.

The metal bands that encircle the crate are under tension. Wearing eye protection and gloves, cut the metal bands with tin snips. Be careful—the cut edges are sharp. The crate top is secured to the base by the bands.

After removing the bands, lift off the crate top. Tip the crate from the tailstock end up and over the machine. Do not damage the crate; you may need it another time to transport the machine.

Now open the accessories box. Check the items in it against the Accessory Checklist. After accounting for all parts, you're ready to assemble the stand.

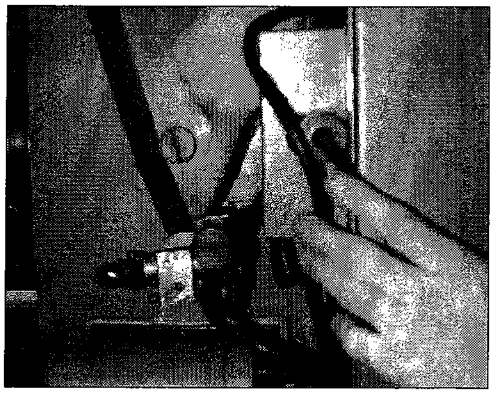
The stand is in two cardboard boxes strapped to the top of the crate. The larger part supports the headstock, the smaller, the tailstock. The chip pan and center shield (the piece of metal that spans the gap between the headstock and tailstock stands) are nailed to the inside of the shipping crate. The left-hand center-shield bracket screws onto the headstock stand; the right-hand bracket screws onto the tailstock stand.

Assemble the headstock and tailstock stands and attach the center-shield brackets. Do not put the shield in place until the machine is on the stand. Put the preassembled stand into position.

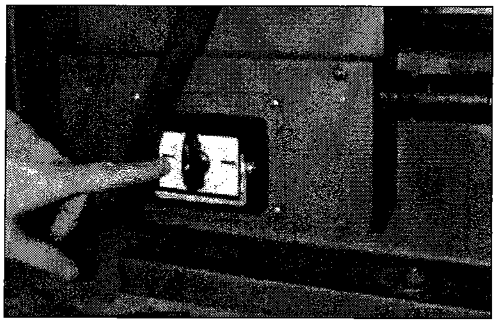
If you make your own stand, be sure it is strong and rigid. It should be at least 64″ long × 24″ wide × 28 – 33″ high.

You're ready now to put the CB-1239 XL on its stand. Move the tailstock to the end of the lathe bed and the saddle about two-thirds of the way down the lathe. Remove the mill pulley cover and all boxes from the pallet base. We recommend using a cherry picker, forklift, or hoist to raise and move the machine with the help of three sturdy straps rated at 2000 lb.

Strap #1 supports the headstock end of the machine. Pass it under the casting to the left of the headstock mounting base between the motor and lathe bed (*Figure* 5.1). Don't let it go under the gear/pulleybox. It should also



5.1 Strap #1 goes between the motor and the lathe bed.

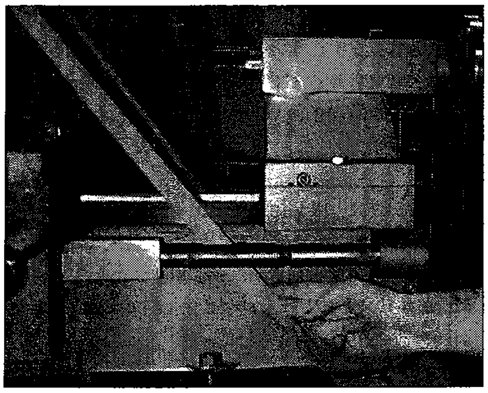


5.2 Be sure the strap misses the switch on the front of the lathe.

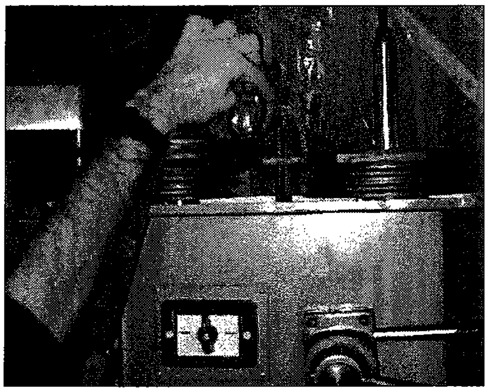
miss the switch on the front of the lathe (*Figure* 5.2). Protect the strap from sharp edges with padding, especially the corners where the strap touches the casting and mill guard base.

Strap #2 supports the tailstock end of the machine. Pass it under the lathe-bed casting to the right of the tailstock mounting base between the leadscrew and lathe bed (*Figure* 5.3). Pad it to protect it from the edge of the casting.

Strap #3 passes directly under the lathe bed at the balance point (about halfway between the mill spindle and On/Off switch), *Figure* 5.4, between the leadscrew and lathe bed. Pad the strap where it touches the ways. Equalize the tension on the straps and have them meet at the balance point. Unbolt the machine from the pallet base and remove the packing crate bolts. Replace them with the mounting bolts provided. Use these bolts as guides for positioning the machine on the stand.



5.3 Strap #2 goes between the leadscrew and lathe bed.

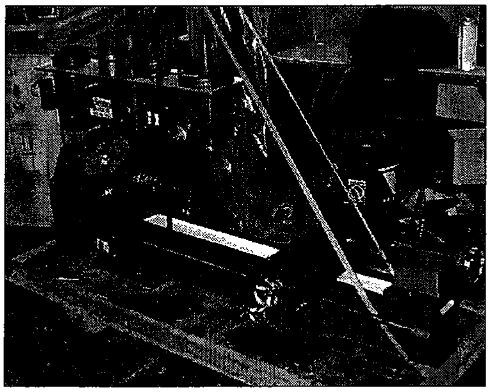


5.4 Put strap #3 under the machine at the balance point.

Carefully lift the machine (*Figure* 5.5), move it over the stand, and lower it into position. ***Do not place any part of your body between the machine and the stand.*** Bolt the machine to the stand, using one flat washer and one lock washer per bolt. If you prefer using the packing-crate bolts, shorten them so they don't interfere with the movement of the stand drawers.

Before permanently anchoring the CB-1239 XL, be sure the bed is level (*Figures* 5.6 and 5.7). Use a precision level, both along and across the bed. Shim up any low points with sheet metal or other noncompressible material. After tightening the anchor bolts, check the bed again.

Install the V-belts on the machine and place the drawbar cover on the mill/drillhead. To install the belts in the gearbox, select the desired speed from the chart attached to the side of the gearbox cover and on the mill/drill-head. Place each belt in the proper slot on the spindle pulley. Use the motor pulley to roll the belts on. Do not let the belts twist in the slots. To remove a belt, use the motor pulley as a handwheel to turn the belt while you pull it toward you. The belt will roll off. Loosen the tensioner to position the belts more easily.



5.5 Raise the machine using the three sturdy straps.

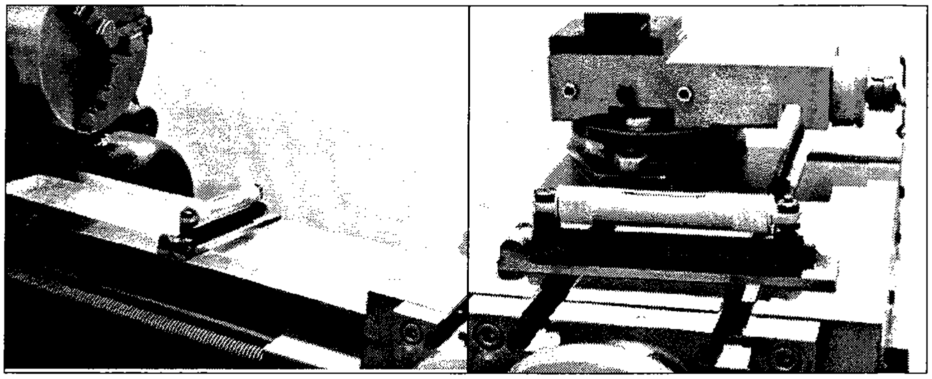
5.3 Selecting a location

There are six major considerations for selecting a location for your CB-1239 XL:

- Operation is from the apron side, so allow at least 40 – 48″ operator clearance in front of the machine.

- The machine should be on a 30-amp circuit, positioned as close as possible to the power supply. It's preferable not to use an extension cord. If you must use one, check with an electrician for the proper size.

- Provide ample working light over the operator's shoulder.

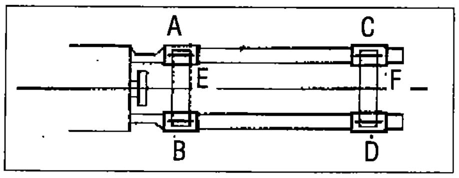


5.6 Check along and across the bed to make sure it is level.

• Place the machine on a solid foundation—concrete, if possible. If you will put it on a wood floor, make sure the floor is adequate. Brace it if necessary to prevent sagging or settling. Place the CB-1239 XL solidly and squarely on the bed, and make sure it is level.

• Make allowances at the back of the machine tool, at the end, and above it for later additions, attachments, and/or accessories. Provide clearance on the left end for bar stock to be fed through the spindle. If you are considering placing more than one machine in an area, allow enough floor space to feed long bar stock to each machine.

• Mount the machine on a heavy, rigid stand.



5.7 To check bench and bed level accuracies, successively place level at A, B, C, D (longitudinal positions), and E and F (transverse positions). Dovetail flatness is 0.003″ / 30°.

5.4 Cleaning and lubricating the machine

CB-1239 XLs are shipped with a protective grease coating. Use noncorrosive kerosene or white mineral spirits to remove it.

Once it's set up and cleaned, your CB-1239 XL is ready for lubricating. You must do this carefully and thoroughly before starting the machine. Use a pressure oil can and a supply of good-quality SAE No. 20 or 30-weight machine oil.

To be thorough and complete, use the following routine:

Oiling the headstock

• Swing out the gearbox door to expose the pick-off gears. Oil the button in the hub of the pick-off gears. Then put a few drops of oil on the teeth of all the gears.

Oiling the ways

• Run the carriage as far to the left as possible. Put a few drops of oil on the ways. Run the carriage to the extreme right and repeat. You may want to use Waylube, an oil specially formulated for ways.

Oiling the carriage

• Lubricate the oil buttons in the cross-feed table. There are two buttons on the left side of the saddle for the bedways and two on the front of the cross slide for the cross-slide ways.

• Put a few drops of oil on the compound and cross-slide feedscrews.

• Put a few drops of oil on the compound slides.

Oiling the tailstock

• Oil the buttons on top of the tailstock.

Oiling the apron

• Put oil in the button just behind the cross-feed wheel handle.

Oiling the leadscrew

• Put oil in the oil buttons in the left trestle and the support for the right end of the leadscrew.

• Put a few drops of oil along the leadscrew and feed shaft.

Oiling the compound

- Put oil in the two buttons on the top.

To keep your machine in peak condition, lubricate it daily after removing any debris. Do not fill the gearbox sight glass more than halfway; too much oil will make the motor lug and sling oil out from behind the chuck and inside the belt box.

5.5 Adjusting belt tension

To adjust the tension on the mill belt, release the lock handle (*Figure* 4.9) to the right of the motor. Using the crank, turn the tensioner clockwise to release the tension and counterclockwise to tighten the belts. The idler pulley is on an eccentric shaft so both belts tighten together. The lock handle holds the motor in place.

To tighten the lathe belts, move the tensioner handle (*Figure* 4.6) so it points toward the lathe head. Turn the screw arbor with the knurled knob clockwise to tighten the belt. If you run out of adjustment (the bolt bottoms out) before the belt is tight, remove the pivot pin, back out the screw arbor with the knurled knob, and screw out the attachment screw arbor. Then reattach it with the pivot pin. Repeat this process. Adjust the locating sleeve to the left of the tensioner handle so the two capscrews act as a stop when the eccentric wheel goes just past center.

5.6 Running in

Though all CB-1239 XLs are run at the factory and again before shipping, it is wise to put your new machine through a breaking-in run before putting it to work. After oiling the machine, check the belts to make sure the tension is adjusted correctly. *Do not plug in the machine yet.*

Follow these steps:

- Set the lathe to 250 rpm.

- Plug the machine into a grounded 30-amp circuit.

- Start the lathe motor by selecting Forward or Reverse on the Forward/Off/Reverse switch. Let the lathe run for 15 minutes. Do the same with the mill motor.

- During the run-in, try the controls on the apron. Get the feel of your machine.

SECTION SIX
TURNING

In turning, the lathe rotates a workpiece against a cutting edge. With its versatility and numerous attachments, accessories, and cutting tools, it can do almost any machining operation.

6.1 Cutting Speeds for Various Diameters

FPM	50	60	70	80	90	100	110	120	130	140	150	200	300
DIAM	RPM												
1/16″	3056	3667	4278	4889	5500	6111	6722	7334	7945	8556	9167	12229	18344
1/8″	1528	1833	2139	2445	2751	3056	3361	3667	3973	4278	4584	6115	9172
3/16″	1019	1222	1426	1630	1833	2037	2241	2445	2648	2852	3056	4076	6115
1/4″	764	917	1070	1222	1375	1538	1681	1833	1986	2139	2292	3057	4586
5/16″	611	733	856	978	1100	1222	1345	1467	1589	1711	1833	2446	3669
3/8″	509	611	713	815	917	1019	1120	1222	1324	1426	1528	2038	3057
7/16″	437	524	611	698	786	873	960	1048	1135	1222	1310	1747	2621
1/2″	382	458	535	611	688	764	840	917	993	1070	1146	1529	2293
5/8″	306	367	428	489	550	611	672	733	794	856	917	1223	1834
3/4″	255	306	357	407	458	509	560	611	662	713	764	1019	1529
7/8″	218	262	306	349	393	426	480	524	568	611	655	874	1310
1″	191	229	267	306	366	372	420	458	497	535	573	764	1146
1-1/8″	170	204	238	272	306	340	373	407	441	475	509	679	1019
1-1/4″	153	183	216	244	275	306	336	367	397	428	458	612	918
1-3/8″	139	167	194	222	250	278	306	333	361	389	417	556	834
1-1/2″	127	153	178	204	229	255	280	306	331	357	382	510	765
1-5/8″	117	141	165	188	212	235	259	282	306	329	353	470	705
1-7/8″	102	122	143	163	183	204	224	244	265	285	306	408	612
2″	95	115	134	153	172	191	210	229	248	267	287	382	573
2-1/4″	85	102	119	136	153	170	187	204	221	238	255	340	510
2-1/2″	76	91	107	122	137	153	168	183	199	214	229	306	459
2-3/4″	69	82	97	111	125	139	153	167	181	194	208	278	417
3″	64	76	89	102	115	127	140	153	166	178	191	254	371

6.1 Table provides exact speeds (rpm). It does not take machine speed limitations into account. The CB-1239 XL has nine mill speeds (315 – 2000 rpm) and nine lathe speeds (250 – 1600 rpm). Determine the desired rate of speed and find the closest speed available on your machine.

The modern lathe offers the following:

- The strength to cut hard, tough materials
- The means to apply power
- The means to hold the cutting point tight
- The means to regulate operating speed
- The means to feed the tool into or across, or into ***and*** across, the work, either manually or by engine power, under precise control
- The means to maintain a predetermined ratio between the rates of rotating works and the travel of the cutting point or points.

6.1 Turning speeds

You may wonder why it's necessary to slow or change the speed of the lathe spindle. When metal cuts metal at too high a speed, the tool burns up. You can machine soft metals like aluminum at fast speeds without danger or trouble, but you must cut hard steels and other metals slowly.

You must also consider the diameter of the workpiece (*Figure* 6.1). A point on a 3″-diameter shaft will pass the cutting tool three times as fast as a point on a 1″-diameter shaft rotating at the same speed. This is because the point travels a tripled circumference. For work in any given material, the larger the diameter, the slower the speed in spindle revolutions needed to get the desired feet-per-minute (fpm) cutting speed.

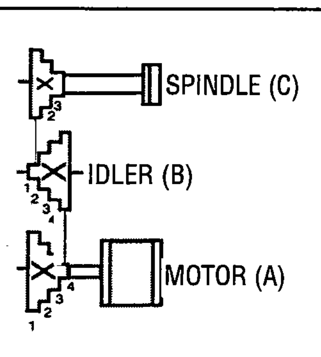
The term "screw-cutting" comes from the fact that the lathe cuts screws or threads in various numbers per inch of material threaded, according to the operator's needs. The CB-1239 XL cuts threads to metric or inch standards.

In screw cutting, or thread cutting, the carriage carries the thread-cutting tool and moves by engaging the half-nut on the rotating leadscrew (*Figure* 4.5). The basic thread-cutting principle is that the revolving leadscrew pulls the carriage in the desired direction at the desired speed. The carriage transports the toolrest and the threading tool, which cuts the screw thread into the metal piece being machined.

The faster the leadscrew revolves in relation to the spindle, the coarser the thread. This is because the threading tool moves farther across the revolving metal with each workpiece revolution.

The lathe spindle holding the workpiece revolves at a selected speed (revolutions per minute, or rpm) according to the type and size of the workpiece. The leadscrew, which runs the length of the lathe bed, also revolves at the desired rpm. There is a definite and changeable ratio between spindle and leadscrew speeds. *Figure* 6.2 shows belt positions for various speeds.

6.2 Lathe Belt Positions for Various Speeds (rpm)



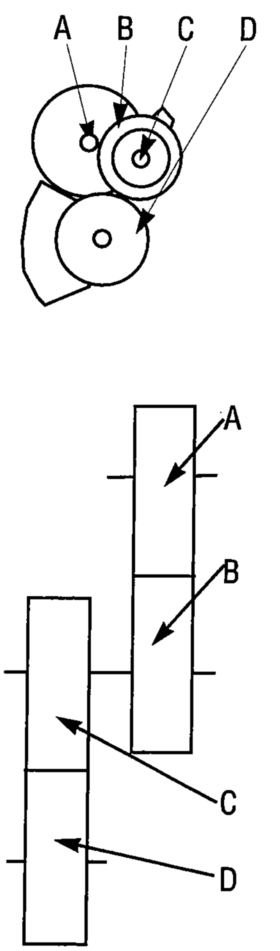
RPM					
$\frac{A4}{B4} \times \frac{B1}{C1}$	250	$\frac{A3}{B3} \times \frac{B1}{C1}$	500	$\frac{A2}{B2} \times \frac{B3}{C3}$	1000
$\frac{A4}{B4} \times \frac{B2}{C2}$	315	$\frac{A2}{B2} \times \frac{B1}{C1}$	630	$\frac{A1}{B1} \times \frac{B2}{C2}$	1250
$\frac{A4}{B4} \times \frac{B3}{C3}$	400	$\frac{A3}{B3} \times \frac{B2}{C2}$	800	$\frac{A1}{B1} \times \frac{B3}{C3}$	1600

6.2 Gear ratios

The lathe lets you use various indicated gear combinations to cut the desired number of threads per inch (tpi), or the metric equivalent, automatically. You need not understand fully the principles of gear ratios and arrangements any more than you must understand an internal combustion engine to drive a car

The CB-1239 XL has a pick-gear gearbox (*Figure* 4.4); gears are picked and placed to change the gear ratios. The gearbox mechanism determines the leadscrew's rotation rate in relation to the spindle's for threading and turning. To change the thread pitch (tpi), replace the gears per *Figure* 6.3.

6.3 Thread Pitches



Threads per inch

Powerfeed Speeds 1	2	Gear Position A	B	C	D
22	11	40	27	50	22
24	12	40	27	50	24
26	13	40	27	50	26
28	14	40	27	50	28
30	15	40	27	50	30
32	16	50	27	35	28
34	17	50	27	40	34
36	18	50	27	30	27
38	19	50	27	40	38
40	20	40	27	35	28

Metric Pitches

Powerfeed Speeds 1	2	Gear Position A	B	C	D
0.12	0.15	25	50	35	40
0.50	1.00	35	40	50	30
0.60		35	24	30	25
0.70		35	24	35	25
0.75	1.50	35	40	50	20
0.80		35	24	40	25
1.75		35	24	35	20
1.00	2.00	35	24	40	20
1.25	2.50	50	24	35	20
1.50	3.00	50	20	35	20

← Original setup

SECTION SEVEN
METALCUTTING THEORY

A machine tool is no more efficient than its cutting edge. Because lathe operation requires continuous regrinding and resharpening of the machine's cutting tools, operators should know some metalcutting theory.

All cutting with a sharp edge, whether with the thin blade of a knife or the almost square edge of a closely supported carbide tool, is basically a wedging-apart action. The first essential of any wedging tool is a penetrating edge. The narrower the blade, the less force or power is required to wedge it through the material to be cut. Therefore, when cutting comparatively soft materials with a cutting tool made from a much harder, stronger substance, the blade can be very thin and sharpened to a long, thin edge.

As the material hardness (or resistance to separation) increases, the strength of the cutting edge must also increase. A knife whose edge is too thin dulls quickly, even when cutting comparatively soft materials. This explains why, in *Figure* 7.1, the knife edge breaks off almost upon contact with the metal while the more obtuse cutting edge of a cold chisel stands up to continuous pounding.

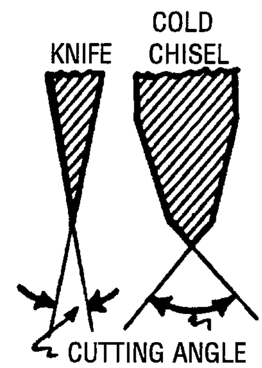
The primary requirements of the cutting edge of any metalcutting tool are that it must be (A) strong and (B) closely supported. This is understandable when we realize how much downward pressure is exerted against the cutting edge. Pressure against cutting tools as great as 250,000 pounds per square inch (psi) have been measured on large metalcutting lathes.

While the workpiece revolves, a strong, rigidly held cutting edge is forced under its surface. As it presses down against the cutting edge, small chips or a continuous ribbon of metal wedges away (*Figure* 7.2). Only in soft, ductile materials is this wedging action continuous.

On harder substances, the wedging force compresses, rises to the shearing point, and shears. Then it builds up and shears again, repeatedly. You can see this in the distortion of chips when cuts are heavy and materials hard.

When the shearing vibration synchronizes with the natural vibration period of any part of the tool, toolholder, or workpiece, chatter occurs. You eliminate chatter by changing one of the harmonizing factors: making the tool more rigid, holding the cutter closer up in the toolholder, backing the toolholder farther into the toolpost, or altering the feed of the tool, operating speed of the lathe, or angle of the cutter bit to the workpiece.

Long years of experience have established certain principles of efficiency for tool design, tool approach, and tool and cutting angles for various operations. The average operator needs to know only the effects of the cutters' grinding.

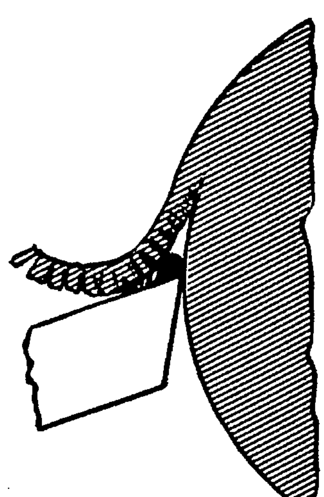


7.1 The knife edge breaks off almost upon contact with the metal because its cutting edge is not as strong as that of a cold chisel.

7.1 Tool sharpness

Instead of being the all-important factor in determining tool performance, keenness of the cutting edge is just one of many performance factors on metalcutting tools. On rough or heavy cuts, it is far less important than strength, because a false cutting edge or crust usually builds up on the tool edge, and though the edge dulls, its angle often increases the cutting tool's efficiency by increasing its wedging action. Cutter shape is usually more important than edges, which generally are rough-ground and must be honed for fine finishing cuts or work in soft, ductile materials like brass or aluminum.

Lack of clearance, which lets a tool drag on the work below the cutting edge, acts as a brake on the lathe, greatly reducing pressure on the cutting point and interfering with tool performance more than edge dullness. At the same time, excessive clearance weakens a tool because of insufficient support to the cutting



7.2 Wedging-off action in cutting hard steel. Note the false edge or crust that builds up on the cutting edge.

7.3 Cutting Speeds and Feeds for High-Speed-Steel Tools

	Low-Carbon Steel	High-Carbon Steel Annealed	Alloy Steel Normalized	Aluminum Alloys	Cast Iron	Bronze
Speed (sfm)						
Roughing	90	50	45	200	70	100
Finishing	120	65	60	300	80	130
Feed (ipr)						
Roughing	0.010-0.202	0.010-0.020	0.010-0.020	0.015-0.030	0.010-0.020	0.010-0.020
Finishing	0.003-0.005	0.003-0.005	0.003-0.005	0.005-0.010	0.003-0.010	0.003-0.010

edge. Such an edge will break off if you use the tool on hard materials.

Clearance requirements change with almost every operation, but there are certain standards for all aspects of the cutting tool. You must not only provide clearance from the cutting edge; there must also be end and side clearance. To help the chip pass with minimum resistance across the top of the tool, it should often have top rake as well. You determine the shapes and rakes to which you'll grind your tools by the toolholder you use. The CB-1239 XL has a four-sided turret toolpost that accommodates four high-speed-steel (HSS), carbide-tipped, or indexable carbide turning tools.

7.2 Heat

The energy expended at the lathe's cutting point converts largely into heat, and because the energy expended is great, the heat is intense. Before the development of today's HSS, carbide, and ceramic tools, this heat created a serious machining problem. Machining could be done only under a steady flow of coolant, which kept the tool from heating to its annealing point, softening, and breaking down.

With HSS, you can usually cut dry unless you're using a small lathe at extremely high speeds on continuous, heavy-duty production work. HSS tools are self-hardening even when red hot (950° F). They do not dissipate the heat, however, or in any way prevent the workpiece from heating up. Because steel expands when heated, it is a good idea, especially when working on long shafts, to check the tightness of the lathe centers frequently and make sure workpiece expansion does not cause centers to bind.

In everyday lathe operations like thread cutting and knurling, always use a cutting oil or other lubricant. On such work, especially if the cut is light and lathe speed low, dipping a brush in oil occasionally and holding it against the work will provide sufficient lubrication. For continuous, high-speed, heavy-duty production work, however, especially on tough alloy steels, using a cutting oil or coolant will increase cutting efficiency. It's essential if you're using a non-HSS cutting tool. When you use coolant, direct it against the cutting point and cutter. Smithy offers various types of coolant systems, and you should seriously consider installing a coolant device.

Figure 7.3 lists cutting speeds and feeds for HSS cutters so you can set up safe rpm rates. The formula is as follows:

$$\text{rpm} = CS \times 4 / D''$$

where CS = cutting speed in surface feet per minute (sfm) and D″ = diameter of the workpiece in inches.

To use this formula, find the cutting speed you need on the chart and plug that number into the CS portion of the formula. After calculating the rpm, use the nearest or next-lower speed on the lathe and set the speed.

If you were to make a finish cut on a piece of aluminum 1″ in diameter, for example, you would see the desired sfm per *Figure* 7.3 is 300. Then rpm = 300 sfm × 4 / 1

rpm = 1200 / 1

rpm = 1200 or next slower speed.

For high-carbon steel, also 1″ in diameter,

rpm = 50 sfm × 4 / 1

rpm = 200 / 1

rpm = 200 or next slower speed.

The CB-1239 XL's four-turret toolpost lets you mount up to four different tools at the same time. You can install all standard-shaped turning and facing tools with 1/2″ or smaller shanks. Smithy also offers quick-change tool sets that greatly speed up lathe operations. Contact a Smithy technician for details.

SECTION EIGHT
GRINDING CUTTER BITS FOR LATHE TOOLS

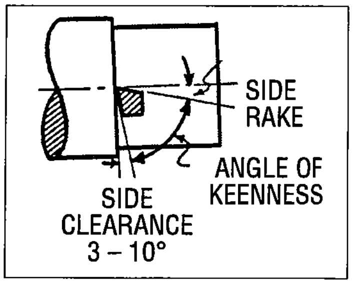
8.1 High-speed-steel cutters

The advantage of HSS cutter bits is you can shape them to exact specifications through grinding. This lets you grind a stock shape into any form. Stock shapes come in an assortment of types, from squares to flats to bevels. Many shops buy their cutters as ready-ground or ready-to-grind bits or blades.

Ready-to-grind bits and blades are of specially selected HSS, cut to length and properly heat-treated. They are fine tools in the rough and generally superior to HSS shapes sold by the pound.

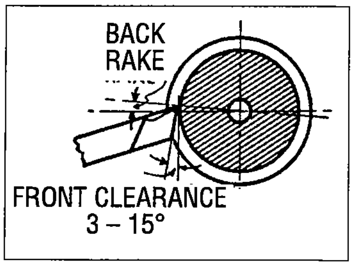
In grinding HSS cutter bits, you have five major goals:

- A strong, keen cutting edge or point
- The proper cutting form (the correct or most convenient shape for a specific operation)
- Front clearance away from the toolpoint
- Clearance away from the side of the tool (side rake)
- Free chip movement over the tool and away from the cutting edge.



8.1 Keenness angles vary from 60° to 90°.

Keenness angles can vary from 60° for mild softness to 90° for hard steels and castings (*Figure* 8.1).



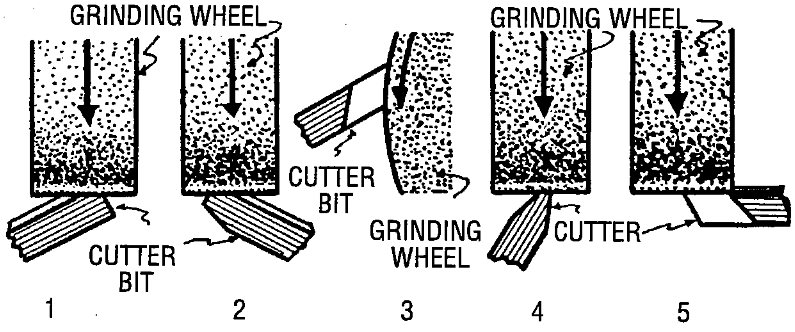
8.2 The edge weakens if front clearance is too great.

Front clearance must always be sufficient to clear the work. If it is too great, however, the edge weakens and breaks off (*Figure* 8.2). Side and back-rake requirements vary with the material used and operation performed. Back rake is important to smooth chip flow, which is needed for a uniform chip and good finish, especially in soft materials. Side rake directs the chip flow away from the point of cut.

Grind cutters on a true-surfaced, good-quality, medium-grit grinding wheel (preferably an 8″, 46 - 60A or 68A grit Carborundum wheel) at 6000 or 6500 rpm. When starting with an unground cutter bit, the procedure (*Figure* 8.3) is usually to (1) grind the left-side clearance, (2) grind the right-side clearance, (3) grind the end clearance, (4) grind the end form or radius, and (5) grind the top rake, touching in a chipbreaker. If honing the cutting edge (for fine finishing or machining soft materials), draw the cutter away from the cutting edge across the oilstone (*Figure* 8.4).

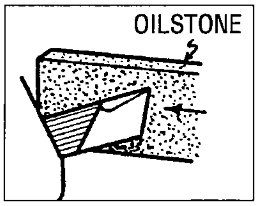
8.2 Materials other than steel

As pointed out earlier, when grinding HSS cutters, we determine cutting angles primarily by strength requirements, not keenness requirements. Angles and rakes for general industrial shop use are established. In machining steel, the softer the steel, the keener the angle of the cutting



8.3 With an unground cutter bit you grind (1) the left-side clearance, (2) right-side clearance, (3) end clearance, (4) radius, and (5) top rake.

edge. For soft steels, angles as acute as 61° are possible (*Figure* 8.5).



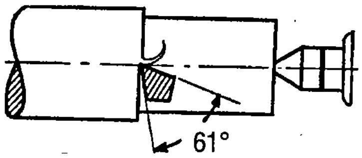
8.4 When honing, draw the cutter away from the cutting edge across the oilstone.

The same general rule applies to cast iron. Chilled or very hard cast iron requires tools with cutting-edge angles as great as 85°. For ordinary cast iron, you obtain greatest efficiency with a more acute cutting edge—approximately 71° (*Figure* 8.6).

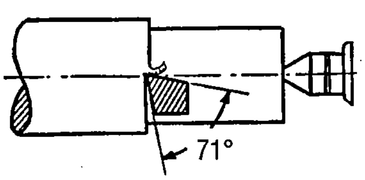
8.3 Bits for turning and machining brass

Brass tends to pull or drag when machined. It's best to machine it on dead center with the top rake in the horizontal plane of the lathe centers. Softer than steel, brass needs less support for the cutting edge. Brass cutters require an almost flat top angle and can gain greater angle keenness only in increased side and end rakes. It is often advisable to hone the cutting edges of cutters used to machine brass.

Note: All roundnose cutters are ground with flat tops and equal side rakes because they are fed across the work, to both right and left.



8.5 With soft steels, 61° angles are possible.



8.6 For cast iron, a 71° angle is most efficient.

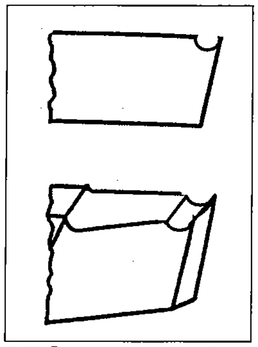
8.4 Special chip craters and chipbreakers

When grinding cut-off blades, and occasionally on other cutter bits where the material's extreme hardness or toughness makes it difficult to control the chip leaving the work, it sometimes helps to grind a smooth, round crater just behind the cutting edge. This serves as a chip guide and starts the chip curling smoothly (*Figure* 8.7).

8.5 Using a center gauge to check V-thread forms

It may be convenient to grind a standard cutter bit for thread cutting, especially for cutting standard 60° V-threads. When grinding an ordinary square cutter into a thread-cutting tool, take care to ensure a true thread form. The easiest way is to use an ordinary center gauge for a standard V-thread tool or a special thread gauge for special thread forms.

To grind a cutter for an ordinary V-thread, grind first the left side of the tool, then the right side, to 30°. Be careful to grind equally from both sides to center the toolpoint. Then test for true form by inserting the newly ground point in the closest-sized V in a standard center gauge (*Figure* 8.8). Examine the gauge and cutter before a light. When the cutter is ground perfectly, no light streak will show between tool and gauge. Use a grinding chart for other rakes.



8.7 A crater starts the chip curling smoothly.

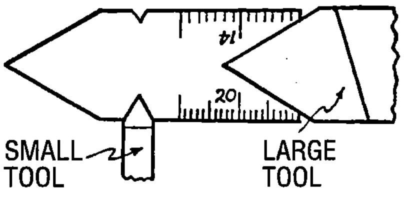
8.6 Acme or other special threads

Thread gauges are available for all standard threads. Before grinding such cutters, ascertain the correct pitch angle of the particular thread profile. For example, the pitch of an acme thread is 29° to a side, and the toolpoint is ground back square to an exact thread profile that requires a different end width for each thread size.

Thread forms must be accurate if threads are to fit snugly and smoothly. Every resharpening of this type of cutter requires regrinding the entire form. It is far better, when doing any amount of threading, to use a threading tool with a special form cutter. Sharpening such cutters requires only flat, top grinding, which does not alter the cutting profile.

8.7 Carbide-tipped cutters and cutter forms

Carbide is a compound of carbon and a metal. In cutting tools, it is usually carbon and tungsten. The hardness of carbide cutting materials approaches that of diamond. While carbides permit easy machining of chilled cast iron, hard and tough steels, hard rubber, Bakelite, glass, and other difficult or "unmachinable" materials, its primary use in metalworking industries is for long production runs on ordinary steels. On such work, carbide-tipped tools permit higher running speeds and much longer runs



8.8 Insert the point into the nearest-sized V in the center gauge.

between resharpenings. The cutting edge of carbide tools stands up 10 – 200 times as long as the edge of HSS tools (*Figure* 8.9).

The advantage of carbide is that it tolerates much higher heat than HSS or other alloys so you can run at higher speeds. The disadvantage is that it is more brittle than HSS and needs adequate support in the toolpost to prevent vibration and breakage.

8.9 Carbide Types and Cutting Tool Applications

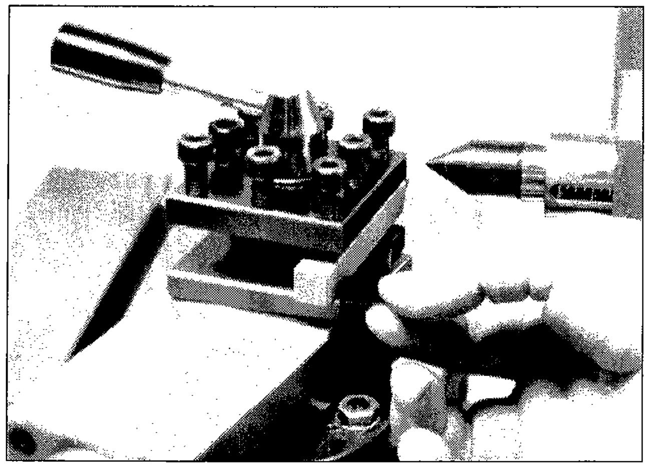
Application	Use	Grade
Cast irons	Roughing Cuts	C-1
Nonferrous, nonmetallic, high-temperature alloys	General Purpose	C-2*
200 and 300-series stainless steels	Light Finishing	C-3
	Precision Boring	C-4
	Roughing Cuts	C-5
	General Purpose	C-6*
Alloy steels	Finishing Cuts	C-7
400-series stainless, high-velocity	Precision Boring	C-8

* C-2 and C-6 are the most commonly used carbides.

SECTION NINE
SETTING UP LATHE TOOLS

After selecting a cutter, insert it in the toolholder. Allow the cutter bit to project just enough to provide the necessary clearance for the cutting point. Remember, the closer the cutter is to the toolpost, the more rigid the cutting edge.

Allen-head capscrews hold the tool in the toolpost. To assure maximum rigidity, don't let the tool extend too far beyond the end of the toolpost turret.



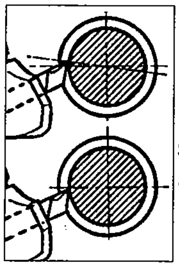
9.1 Placing shims under the tool can correct cutting-tool height.

9.1 Cutting tool height

After inserting the cutting tool into the toolpost, adjust the height of the cutting edge in relation to the lathe center. Insert a center in the tailstock. Then run the tool and center together. The cutting edge on the tool should meet the point on the center. It may be necessary to use shims, which can be of various thicknesses and materials (*Figure* 9.1). Many seasoned machinists use pieces of old hacksaw blades as shims. If the toolbit is too high, shim the back of the toolbit. If it's too low, shim the entire tool.

9.2 Turning tools

For general turning operations, set the point of the cutter bit slightly above the centerline of the work. In steel, the harder the material, the less above center (*Figure* 9.2, top). Exceptions are soft brass, aluminum, and materials that tend to pull or tear. When machining these materials, set the cutter on dead center (*Figure* 9.2, bottom).



9.2 The harder the steel (top), the less above center you set the cutter point. For soft brass and aluminum (bottom), set the cutter on dead center.

When cutting toward the headstock on most turning and threading operations, swing the compound rest to hold the shank of the toolholder at an angle. The angle should be approximately 29-1/2° left of perpendicular to the line of centers, except for extremely heavy, rough, forcing cuts close to the limits. For such work, use a straight-shanked tool held perpendicular to the line of lathe centers in the right side of the toolpost. The tool will tend to swing out of the cut rather than hog into the work if you reach a stalling point (*Figure* 9.3).

9.3 Threading tools

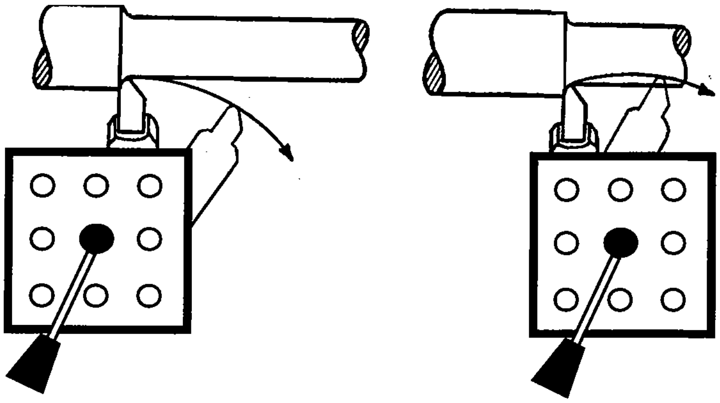
Threading tools should always engage the work on dead center. Any deviation above or below will affect the thread profile (*Figure* 9.4).

9.4 Cut-off, thread-cutting, and facing tools

For cutoff, thread cutting, and facing, feed the cutter to the work on dead center (*Figure* 9.5). For the beginner, the average feed should not exceed 0.002 inches per revolution (ipr).

9.5 Boring and inside threading tools

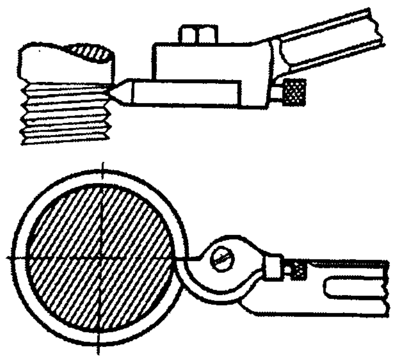
For boring and inside threading, the cutter point engages the work on dead center (*Figure* 9.6). For greater cutting efficiency, position the bar while



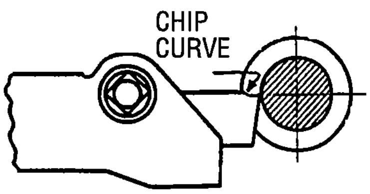
9.3 The tool will swing out of the cut (left) rather than hog into the work (right) if you reach a stalling point. Note the tool is in the right-hand side of the toolpost.

parallel to the line of lathe centers sufficiently below center to give the cutter a 14-1/2° approach angle. For internal threading, the top face of the cutter is ground to compensate for this angle, giving a flat, true-form top face.

Some machinists prefer to position the tool slightly above center when boring. With the bit above center, if a tool chatters it deflects down into empty space instead of into the workpiece.

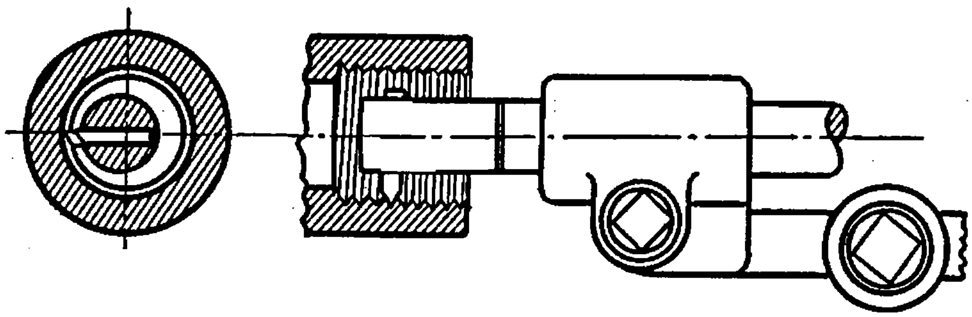


9.4 Threading tools engage the work on dead center.



9.5 Feed the cutter on dead center for cutoff, thread cutting, and facing.

9.6 For boring and inside threading, the cutter point is at dead center.

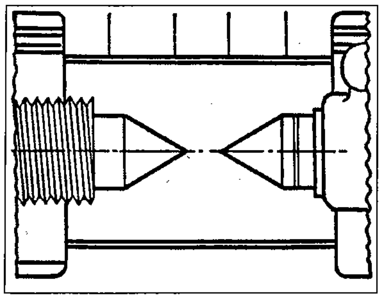


SECTION TEN

SETTING UP WITH CENTERS, COLLETS, AND CHUCKS

Before setting work up on centers, make sure the spindle and tailstock centers align accurately. Do this by inserting a center into the nose spindle and inserting the tailstock center into the tailstock ram. Then move the tailstock toward the headstock until the centers touch (*Figure* 10.1). You can correct any lateral alignment error by adjusting the tailstock set-over screws (*Figure* 10.2).

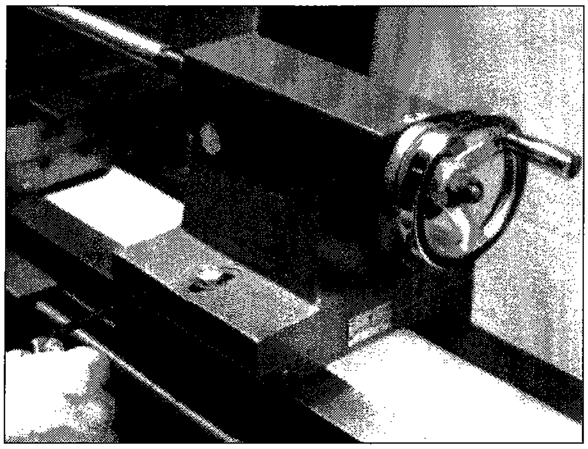
For most turning operations, work is held in the lathe between the lathe centers by means of holes drilled in the ends of the stock to be machined. Your machining accuracy depends primarily on how precisely you locate these holes at the center of the bar or block. Locating these holes is called centering.



10.1 When aligning spindle and tailstock centers, move the tailstock toward the headstock until the centers touch.

10.1 Centering

You can improve centering greatly by first squaring or facing the ends of the workpiece. This gives you a true cross section in which to locate the centering holes. Facing the ends in a chuck is your first lathe operation. (You can do it with a file, but using a lathe is good practice.) First, chuck the stock in the appropriate chuck. Let the stock protrude about an inch. Place a right-hand side tool (or a straight turning tool with a facing cutter) in the toolpost. Carefully adjust the cutting edge so it is exactly on center, then tighten it into the toolpost. If you don't do this, a small tit or projection will remain in the center of the stock, later causing the center drill to run off center.

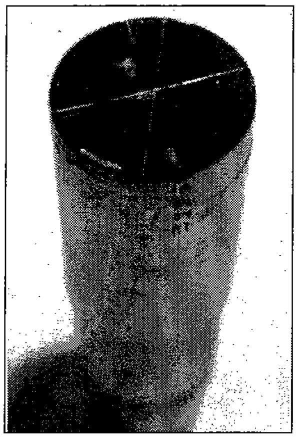


10.2 Adjusting the tailstock set-over screws corrects lateral misalignments.

Start your lathe on the slowest speed. Bring the tool into the cutting position against the center of the workpiece. Feed the tool from the center of the stock outward, toward yourself, using the hand crossfeed. One or two light cuts is usually enough to true up an end roughened by the hacksaw. After facing one end, reverse the work and face the opposite end.

You can center on round stock (*Figure* 10.3) with calipers, dividers, or special centering instruments (*Figure* 10.4). Centering square or rectangular stock is done by scribing lines from opposite corners. The intersection of these lines is the center (*Figure* 10.5). After locating the center of each end, drive a starting depression for the drill into the stock with a center punch.

After locating centers with a center punch, check their accuracy before drilling. Do this by placing the workpiece between the spindle and tailstock centers. Revolve the headstock slowly against the tip of a tool or a piece of rigidly held chalk. The chalk should touch just the high spots (*Figure* 10.6). If the center proves inaccurate (0.002″ or more off), correct the position of the center by repunching at an angle.



10.3 Centering on round stock.

10.2 Drilling and countersinking centers

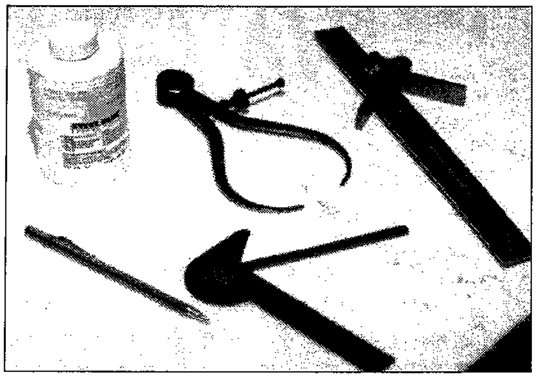
After accurately locating the centers, drill and countersink them to conform to the profile of the lathe centers. They now will take the lathe centers without play or chatter. This is best done with a combination center drill and countersink held in the tailstock arbor chuck.

If a combination drill is not available, you can drill centers with a small drill and countersink them with any drill of sufficient diameter that has been ground to a 60° point. A 60° taper is standard for lathe centerpoints. Correct center depth is given in *Figure* 10.7. Take care to get an accurate 60° countersink in the center (*Figure* 10.8).

10.3 Mounting work between centers

Remove the chuck from the lathe, bolt the faceplate to the spindle flange (*Figure* 4.7), and put in both headstock and tailstock centers. Fasten a lathe dog (*Figure* 10.9) to one end of the work. For ease of operation, use a live or rotating center in the tailstock end so you won't need lubrication. Mount the work as shown in *Figure* 10.10. Before starting the lathe, make sure the centers don't hold the workpiece too tightly. Heat may cause the workpiece to expand, so watch for binding. Adjust the tailstock center so the work turns freely but without end play.

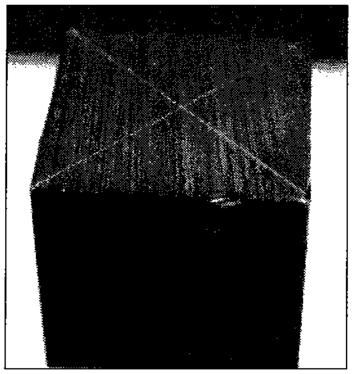
If, after partially machining the workpiece, you find you must machine the stock under the lathe dog, remove the workpiece from the lathe and place the lathe dog on the machined end. Then turn this new tailstock center end of the shaft down to the desired diameter or form.



10.4 You can use calipers, dividers, or other centering instruments.

10.4 Using a clamp dog

Standard lathe dogs drive round, or near-round, shapes. Rectangular or near-rectangular stock requires clamp dogs. In a properly made clamp dog, the underface of the heads of tightening screws are convex and fit into concave seats, while the holes in the upper bar are elongated. This design gives a firm grip of off-square shapes without bending the screws. Top and bottom bars should also have V-notches to permit a firm grip on triangular or other odd-shaped stock. You can also use clamp dogs or special V-jaw dogs to hold highly polished round bars.



10.5 Centering square or rectangular stock.

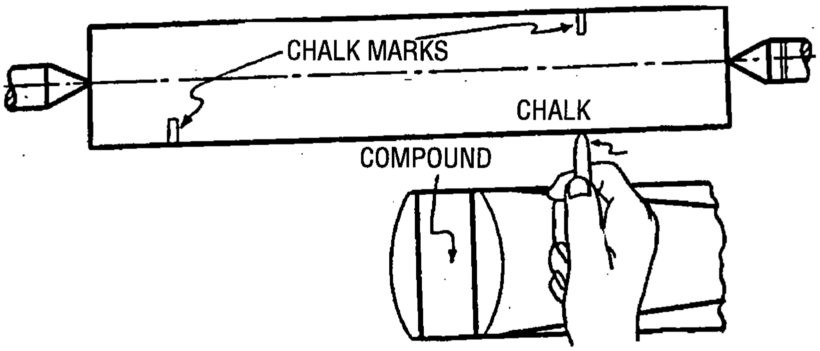
10.5 Using faceplates

To set up work, faceplates serve two purposes. They drive workpieces held between centers, and they hold workpieces shaped so you can't chuck them or mount them on centers.

Faceplates for driving workpieces on centers are generally small. They're notched and slotted to receive the tail of the lathe or clamp dog, bolt drive, or other driving tool (*Figure* 10.11). Faceplates for holding workpieces (irregularly shaped casting, machine, or die parts, for example) are usually larger and of varied design. They may be T-slotted, drilled all over, or slotted and drilled, depending largely on what they do. Workpieces mount on such faceplates with T-slot or standard bolts, strap clamps, angle plates, or other standard setup tools.

Note: Before starting to machine work set up on centers, check to see the lathe dog tail is free in the faceplate slot so it won't lift stock off its true line of centers, as in Figure *10.12. Also, be sure lathe centers fit closely into the center holes to eliminate side play but not so tightly they bind. If you're working on a long workpiece, check it frequently to be sure the center does not bind.*

Each job presents unique problems, so there are few general rules for setting up work on faceplates. Remember this, however: balance unbalanced setups with counterweights to overcome any "throw" as the work revolves (*Figure* 10.13).



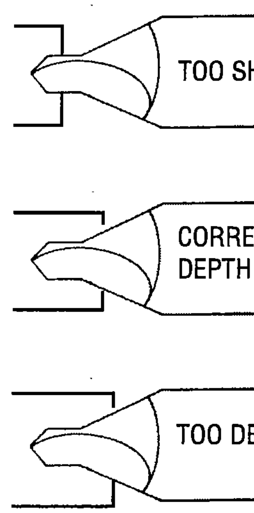
10.6 When you revolve the headstock against a piece of chalk, the chalk should touch just the high spots.

10.6 Setting up work on a mandrel

You can machine cylindrical or bored pipe or cored castings too long to fit in a chuck by mounting them first on a mandrel. Then mount them between centers (*Figure* 10.14). The solid mandrels, which are driven into the hole of the workpiece, must be tight enough to turn the workpiece against the tool without slippage. Oil them lightly before driving them into the workpiece. Otherwise, the workpiece may freeze to the mandrel, making it impossible to remove the mandrel without damaging both workpiece and mandrel.

When removing a mandrel, be sure to drive it back out of, instead of through, the hole. Drive it out in the opposite direction from the way you drove it in.

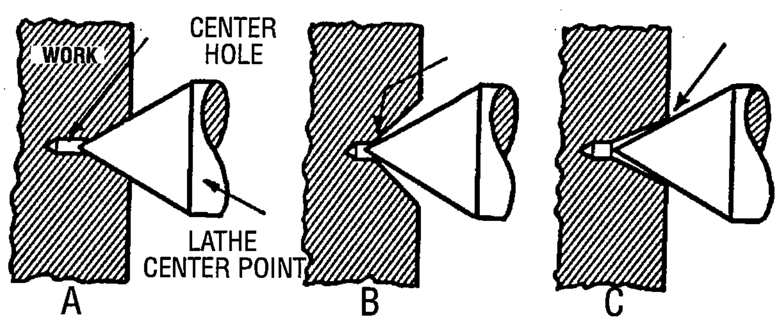
You can purchase hardened steel mandrels, which have a slight (0.003″) ground taper and an expanding collar, to facilitate mounting and demounting (*Figure* 10.15). Mandrels with compressible ends for holding single or ganged pieces are also available. Once it's mounted on a mandrel, machine the workpiece as you would a solid shaft. You can drill eccentric centers in mandrel ends to permit eccentric turning.



10.7 The correct depth of center is shown in the center. If it's too deep (bottom), only sharp outer edges will contact the center.

10.7 Steady rests and follow rests

Rests are for setting up (1) work that is relatively long in proportion to diameter or (2) work whose dead end must be left free for boring or other operations. You can also use rests to machine slender shafts that are apt to spring out of alignment from the thrust of the tool. The purpose of a rest is to support the workpiece and maintain it in accurate alignment for machining. Rests are classed as steady rests or follow rests.



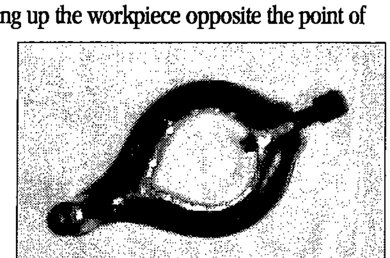
10.8 Counterbore centers with a drill to a 60° point so they fit lathe centers (A). Too obtuse (B) or too acute (C) a counterbore will give insufficient bearing, prevent accuracy, and destroy lathe centers.

Steady rests. Steady rests mount on the lathe bed. Clamped over the ways, they provide three bearing surfaces. These surfaces bear down lightly but rigidly against the surface of the shaft and keep it from moving out of the line of the lathe centers. You can place a steady rest anywhere along the bed where it will best support and steady the workpiece without interfering with the operation (*Figure* 10.16).

To set up a steady rest, first center the work in the chuck and true it up. Then slip the steady rest into position and tighten it to the bed. With the bearing jaws clearing the work, close the top of the rest and tighten the locking screw. Now, with the lathe running, adjust the three bearing jaws to touch, but not push, the workpiece. Finally, test again for alignment, making sure the axis of the workpiece coincides with the axis of the lathe. Otherwise, the end will not be square and the surfaces and boring will be untrue. The tips of the jaws are bronze and require lubrication.

Follow rests. Long or slender shafts that are apt to spring out of alignment by the thrust of the cutting tool often require a follow rest (*Figure* 10.17). The follow rest mounts on the carriage of the lathe and moves with the tool, backing up the workpiece opposite the point of the tool thrust. Follow rests have two adjustable supporting jaws, one holding the work down to keep it from climbing up on the tool and the other behind the work to counter the thrust of the tool.

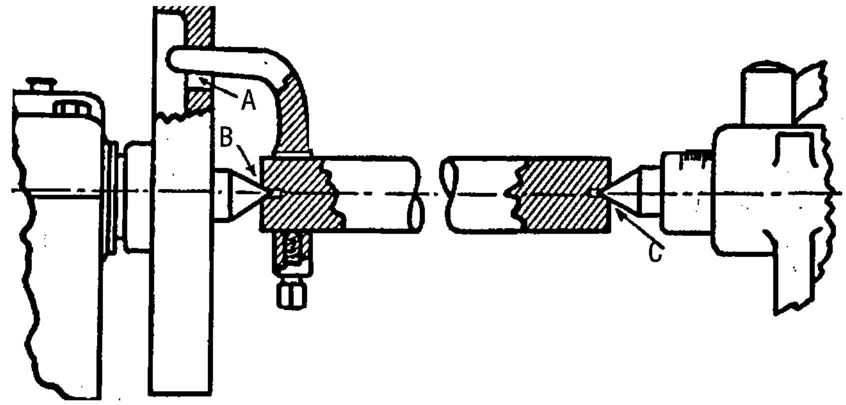
Note: Take great care in adjusting the jaws of rests, as they must form a true axial bearing for the work and let it turn freely but without play.



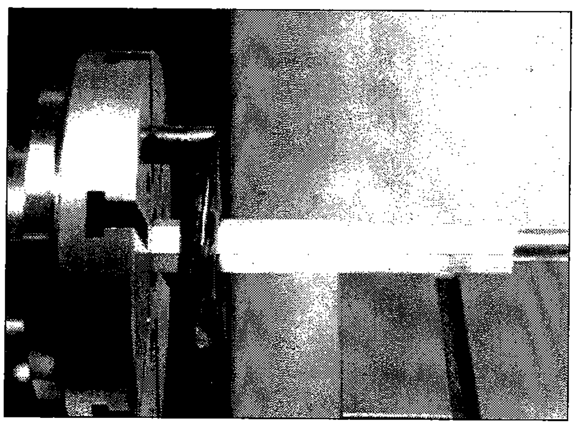
10.9 Lathe dogs drive round or near-round shapes.

10.8 Setting up work in a chuck

Chucks usually hold work that is too short to hold conveniently between centers or work



10.10 Mounting work between centers



10.11 Notched and slotted faceplates, used for driving work on centers, receive driving tools like the tail of a lathe dog.

requiring machining at, into (boring or inside threading), or across its end. While it is possible to set up such work on a faceplate, the convenience of chucks has made them part of every complete lathe. Lathe chucks come in sizes that hold work of diameters approaching the swing of the lathe, and are of many types, including automatic and high-speed, air-operated production chucks.

For ordinary use, there are two standard types of headstock chucks. The four-jaw independent lathe chuck has four holding jaws that can operate independently and adjust to hold round, square, eccentric, or odd-shaped work (*Figure* 10.18). The three-jaw universal geared scroll chuck holds only round or near-round work with three, six, nine, 12, or other multiple-numbered sides. It always holds work concentrically and has the advantage of being self-centering—all jaws move in or out together (*Figure* 10.19).

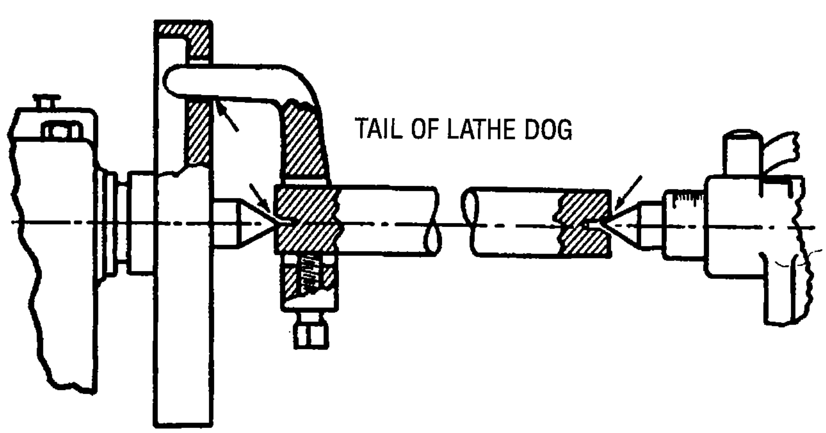
10.9 Mounting work in a four-jaw independent lathe chuck

For small-diameter, short work, insert jaws in the chuck with high ends to the center. This gives the maximum gripping and tool clearance (*Figure* 10.20). For large-diameter work, insert the jaws in the chuck slots with the high steps of the jaws to the outside of the chuck (*Figure* 10.21).

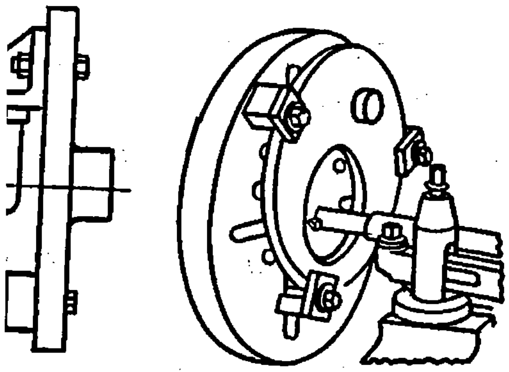
To place work in a chuck, follow these steps:

• Adjust the chuck jaws to the approximate opening to receive the work. Roughly center them by matching the nearest concentric ring on the chuck face with the corresponding mark on the jaws.

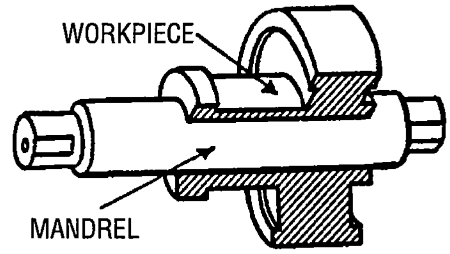
• Place the work in the chuck and grip it. Turn up the opposing jaws a uniform number of turns with the key provided. This will



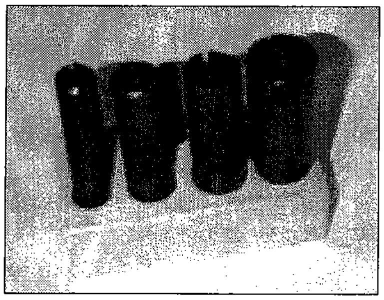
10.12 Make sure the tail of the lathe dog is free in the faceplate slot so it won't lift off the true line of centers.



10.13 **Counterweights can help with unbalanced setups.**



10.14 Mount workpieces too long for a chuck on a mandrel.



10.15 Hardened steel mandrels have a slight ground taper and expanding collar.

hold the work in position. Then bring in the other pair of opposing jaws the same way.

• Revolve the spindle slowly with your left hand while holding a piece of chalk until the chalk touches the high point (the nearest surface) of the work (*Figure* 10.6).

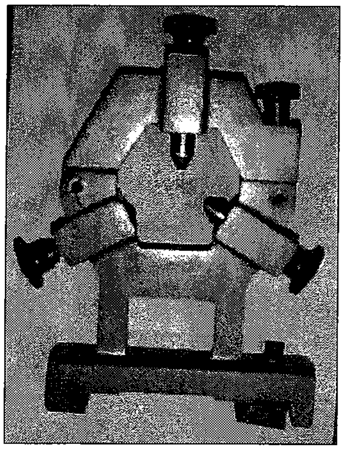
• Guided by the chalk marks, readjust the jaws until a chalk line will carry completely around the work. Then tighten all jaws securely. For greater accuracy, use a dial indicator instead of chalk. When making several identical pieces, after completing each workpiece release only two adjoining jaws, leaving the others to hold the center. The jaws of the four-jaw independent chuck are reversible. You can insert them with high steps to the inside or outside.

Never leave the chuck key (wrench) in the chuck while the chuck is on the spindle. Any movement of the spindle can crash the key into the ways, seriously damaging the ways, spindle, and chuck. Turning on the lathe with the key in the chuck can seriously damage your lathe. The key can also be thrown when the lathe starts, causing damage and/or injury. Never let your hand leave the chuck key unless you are picking it up or storing it.

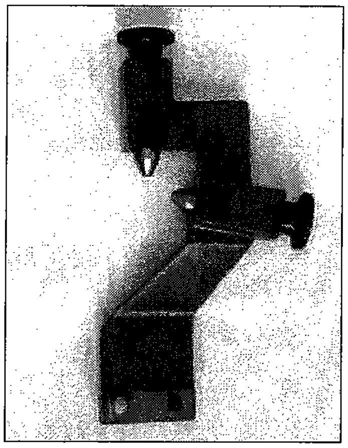
Never remove a chuck or heavy faceplate without first laying a board across the ways to protect them in case the chuck falls when it comes off the spindle nose. Or use a chuck cradle to ease chuck removal and installation.

10.10 Mounting work in a three-jaw universal chuck

Work is set up in a three-jaw universal chuck as in a four-jaw independent chuck, with these exceptions:



10.16 Steady rests mount on the lathe bed.



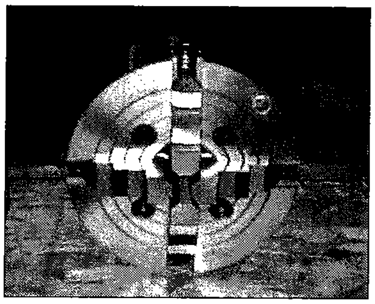
10.17 Follow rests mount on the lathe carriage.

• On three-jaw chucks, the key moves all jaws at once.

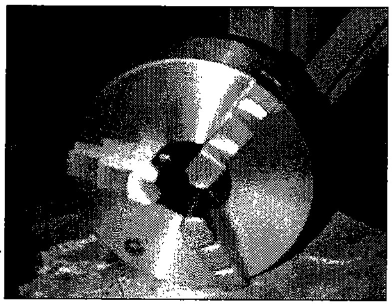
• There is no centering or checking of concentricity because these chucks self-center automatically.

• Jaws are not reversible. Each chuck comes with two sets of jaws. One is for setups with high steps toward the inside (inside jaws), the other for mounting in the chuck with high steps to the outside (outside jaws).

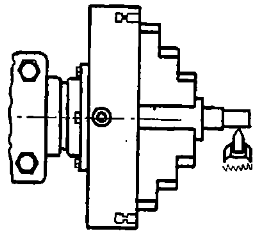
• When installing the chuck jaws on a three-jaw chuck, install them in numerical order and counterclockwise rotation.



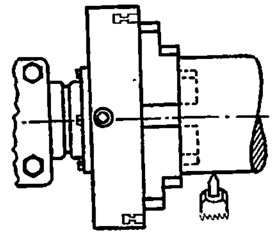
10.18 Four-jaw independent lathe chucks hold round, square, eccentric, or odd-shaped workpieces.



10.19 Three-jaw universal geared scroll chucks hold round or near-round work.



10.20 Mount short, small-diameter workpieces with high ends to the center.



10.21 Mount large-diameter workpieces with high steps of the jaws to the outside.

Each jaw is stamped with a serial number and jaw number (#1, #2, or #3), *Figure* 10.22. The slots in the chuck are not always numbered, but there is a serial number stamped at the #1 slot. With the #1 slot in the 12:00 position, the #2 slot is at 8:00 and the #3 slot at 4:00 (*Figure* 10.23).

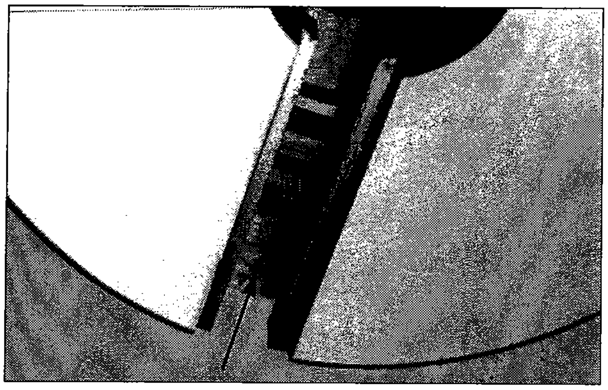
To install the jaws, first insert the #1 jaw into the #1 slot and turn the chuck key until the chuck scroll engages the teeth on the jaw. Then put on the #2 jaw and engage it, then the #3 jaw.

10.11 Collets and collet attachments

To hold small-diameter work, whether bar stock fed through the hole in the spindle or small pieces of semifinished parts, collet attachments are preferable to standard chucks (*Figure* 10.24) for several reasons:

- They have much faster release and grip actions.
- They center the work automatically and accurately.
- They grip firmly even small pieces and pieces with only a short hold.
- They are housed within the spindle nose for maximum tool clearance, making it possible to machine, thread, or cut off close to the spindle.

While chucks are universal tools that hold a range of stock sizes and shapes, collets are special tools. There is a collet for every size and shape of workpiece.



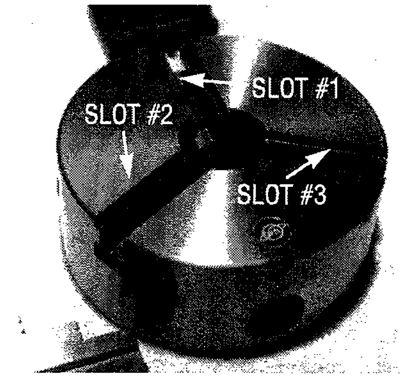
10.22 A serial number is stamped in the #1 slot of the three-jaw chuck.

Made with extreme accuracy, hardened, and ground, standard split collets are slotted so their jaw ends compress inwardly to grip the workpiece. This is done by pulling the collet jaw's externally tapered shoulder into a matching taper-bored adapter sleeve. The adapter sleeve connects the lathe spindle's MT5 taper to the collet's MT3 taper. A drawbar holds the collet in place.

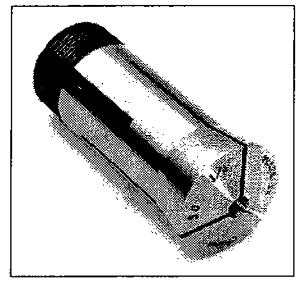
10.12 Toolpost grinders

A fully equipped lathe has a toolpost grinder, a small, independently operated grinding head with an integral electric motor that mounts as a unit in the toolpost T-slot of the compound rest. (For lighter work, some are held in the toolpost.) You can maneuver it as you would any other cutting tool.

Toolpost grinders come with wheels of different shapes, sizes, and grits for grinding different materials and surfaces. They also come with arbors and mounted wheels for grinding internal surfaces. You can use them to grind or polish surfaces; to grind lathe centers, arbors, taper sockets, leader pins, gauges, valve seats, and other close-fitting parts; and to sharpen milling cutters and similar tools.



10.23 With the slot for the #1 jaw in the 12:00 position, the slot for the #2 jaw is at 8:00 and the slot for the #3 jaw is at 4:00.



10.24 Collet attachments are best for small-diameter work.

SECTION ELEVEN
LATHE TURNING

11.1 Rough turning

In turning a shaft to size and shape where you have to cut away a lot of stock, take heavy, rough cuts to get the work done in the least time. Use a transverse powerfeed for heavy cuts—from right to left toward the headstock so the thrust is against the headstock or chuck. Use a right-hand turning or roundnose cutter.

After selecting a cutter, place it into the left side of the turret (*Figure* 11.1). The cutter's point should be just above or on the line of the centers. The greater the diameter of the work, the higher the cutter can be. Adjust the height by placing shims under the cutter and raising or lowering it.

With the tool properly positioned, tighten the Allen capscrews. Next, run the carriage to the right end of the workpiece with the handwheel on the apron. Make sure the lathe is set to feed toward the headstock. Now determine the depth of the cut. Move the tool to the desired depth till it just touches the stock and zero the cross-feed dial.

Start the lathe. Run the crossfeed in by hand to take as heavy a cut as is consistent with the power of the drive or the amount of metal to remove.

Say, for example, you need to reduce a diameter by a known number of thousandths of an inch. If you zero the collar and watch the movement of the dial, you'll know the depth of the feed from the zeroing point.

Note: The dial gives you a good approximation, but for exact measurements, use a measuring instrument.

To reduce the diameter, advance the tool only half as many thousandths on the dial. This is because the tool cuts an equal amount from both sides as it cuts a continuous strip around the work. For example, to reduce the diameter of a shaft 0.005″, you advance the tool only 0.0025″, or 2-1/2 calibrations. This cuts 0.0025″ off each side (0.0025 + 0.0025 = 0.005″ off the diameter of the shaft).

Engage the tool before setting the floating dial. The tool must be moving in the direction you want to go before you set the dial to zero.

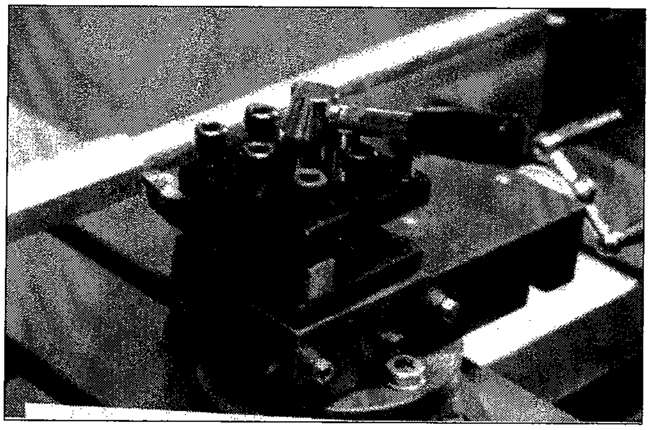
For a screw to move, there must be some play in the thread. When backing the cutting tool away from the cut, move the feedscrew enough to take up the backlash, or lost motion in the screw, before setting the collar or when drawing the tool from the cut. Normal backlash is 0.008 – 0.015″.

Engage the longitudinal feed by pushing down gently on the half-nut lever. *Do not force it.* Press the handle until the threads in the nut and screw line up and the lever drops into place. Always cut deeply enough to reach below the scale on oxidized bars or iron castings. Hard, oxidized surfaces dull tools rapidly.

11.2 Finish turning

After you've rough-turned the workpiece to within 1/32″, replace your cutter bit with a freshly ground, keen-edged cutter. Make one or more light finishing cuts across the machined surface. Check the diameters carefully with a caliper or micrometer to be sure you are working to proper dimensions. Remember, the diameter will reduce twice the thickness of the cut.

For rough turning, most machinists prefer a deep cut and a comparatively fine feed, but the reverse is true for finishing cuts. They usually use a very light crossfeed and a coarse transverse feed with a cutting edge wider than the feed per revolution. In *Figure* 11.2, the left-hand tool illustrates the



11.1 Place cutter into the left side of the turret.

is to be tapered.

When the taper extends the entire length of the workpiece, tailstock setover should equal half the difference between the finished diameters of the ends (*Figure* 11.8). When a taper extends only part of the length of the shaft to be machined, find the correct setoff for the tailstock by dividing the total length of the shaft by the length of the portion to be tapered. Then multiply the resulting quotient by half the difference between the extreme diameters of the finished taper.

Notes: (A) Most drawings give the taper in inches per foot of length. It may be easier to convert all dimensions to inches. (B) Be sure to zero the tailstock before resuming straight turning.

11.7 Boring a tapered hole

Boring a tapered hole involves setting the compound at the desired degree of taper and feeding the tool with the compound rest. Make sure the compound rest is set at half the degrees of the angle of the completed taper hole.

SECTION TWELVE

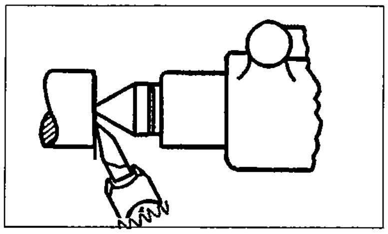
LATHE FACING, KNURLING, AND THREADING

Before removing your workpiece from the centers, face or square up the ends. On accurate work, especially where shoulders, bevels, and the like must be an accurate distance from the ends, do the facing before turning the shank. This also cleans the ends and machines the workpiece to accurate length.

When diameters are large, it's best to face with a special side tool that has a long, thin blade with a wide cutting edge. If you don't have one, use a right or left-hand facing cutter. Feed the tool from the center outward to avoid marring the lathe center (*Figure* 12.1).

12.1 Facing across the chuck

When facing a stub-end workpiece held in the headstock chuck, the same rules apply, except you must set the cutting edge on the exact line of center so there will be no tit. While a side tool is preferable for this type of work because of its smooth slicing cut, you can use an ordinary turning tool. A better choice is an offset turning tool because there will be no clearance problems.



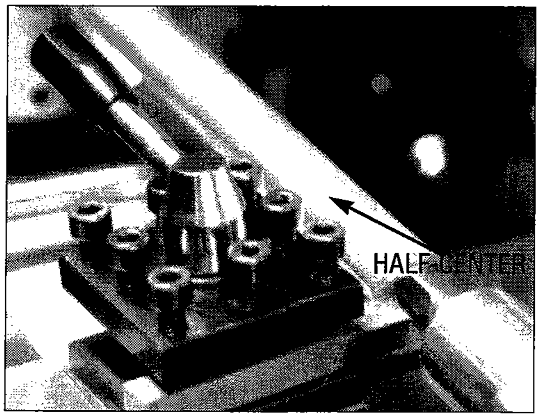
12.1 With a facing cutter, feed the tool from the center out.

When facing with a turning tool, cut from the center to the outside. Do not start with a heavy feed because the sfm increases rapidly as the cutter moves toward the outside.

If you must finish the ends of the shaft, use a half-center (*Figure* 12.2). This lets you extend the tool across the entire face of the work.

12.2 Knurling

Strictly speaking, knurling is not a machining operation because no metal is cut. It is a forming operation, in that the hard patterned knurls are pressed into the work, depressing and raising the surface of the metal into the knurl pattern. As with all other forming operations, your work can be no better than the pattern, your knurling no better than the rolls or knurls. Be sure the knurls are sharp and clean-cut (preferably hob-cut) and properly hardened.



12.2 Using a half-center lets you extend the tool across the entire face of the work.

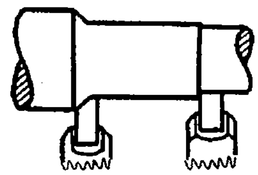
To make a true knurl, maintain uniform pressure on both knurls. Select a self-centering knurling tool that equalizes pressure on the knurls automatically and is strong enough to withstand end and side thrusts.

Before starting, remove the workpiece from the lathe and scribe lines on it indicating the area to be knurled. Operate the lathe at the slowest speed.

Knurling exerts extreme thrust against centers and bearings. You can lessen this thrust materially by feeding the knurling tool at a slight angle off from perpendicular to the line of the workpiece. This engages the right side of the knurl first (*Figure* 12.3).

Place a few drops of oil on the workpiece and knurling tool. Start the rolls of the knurling tool from the right-hand scribe line and feed them in until the knurl reaches a depth of 1/64″. Then stop the lathe and inspect the work. If the knurl is not clear-cut, adjust the tool in or out as needed.

Use plenty of oil, lubricating both knurl and workpiece. Then start the lathe and engage the automatic feed, moving the knurls across the portion to be knurled. When you reach the left scribe line, force the tool into the work another first roughing cut and the right-hand tool shows the following finishing cut.



11.2 Roughing (left) and finishing (right) cuts.

11.3 Turning to shapes

Other turning cuts, machining shapes, corners, fillets, etc., are done the same way. The main difference is in selecting cutter bits and maneuvering the cutting point by means of various cutting tools (*Figure* 11.3).

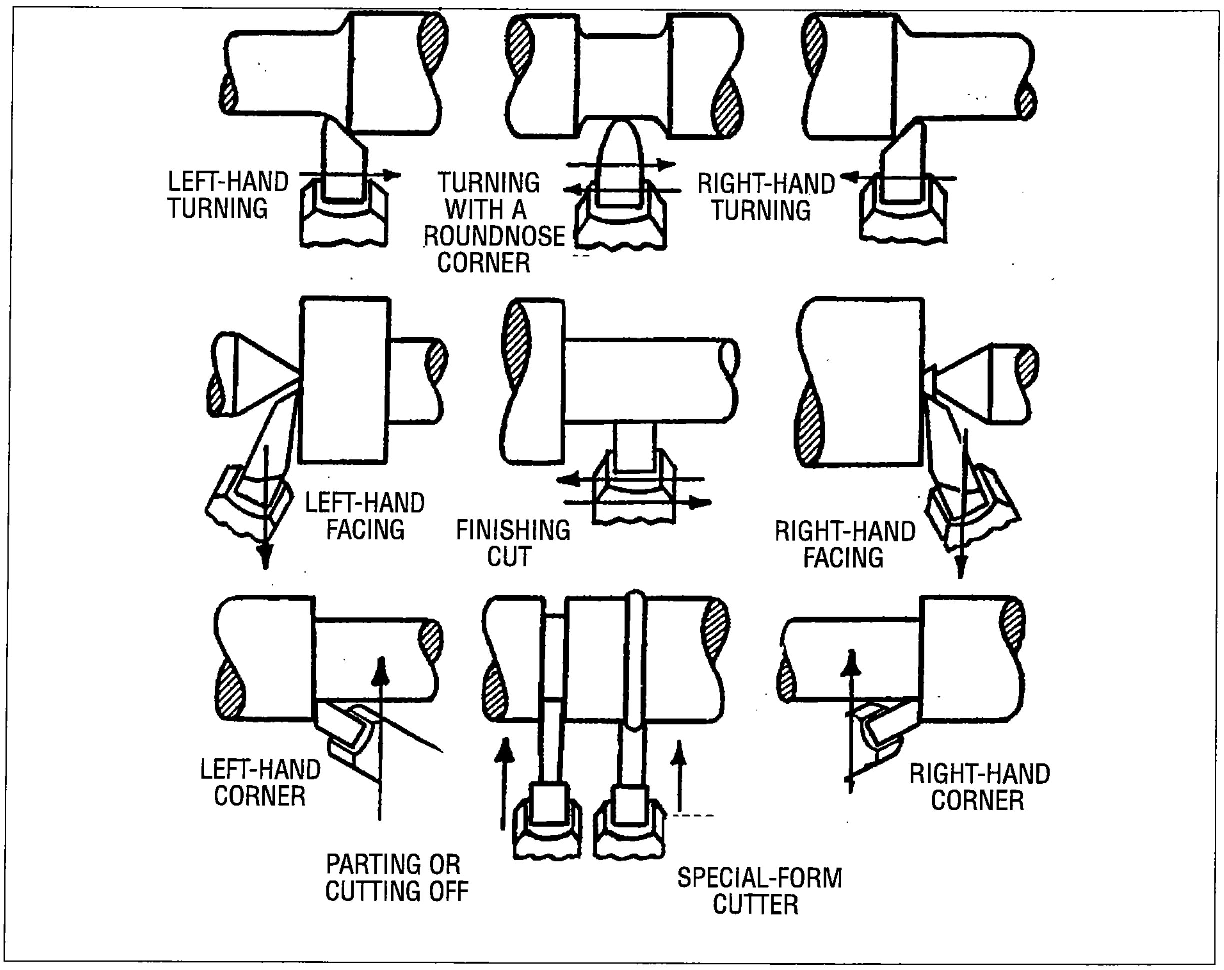
11.4 Machining square corners

To machine an accurate corner, follow these steps:

- Set the compound rest perpendicular to the line of the centers and insert a right or left-hand corner tool.
- Using the longitudinal feed, turn a small diameter to finish up to the shoulder.
- Then, using the compound rest, feed the tool the amount needed to finish the work to the length, taking the last facing cut across the shoulder away from the center.

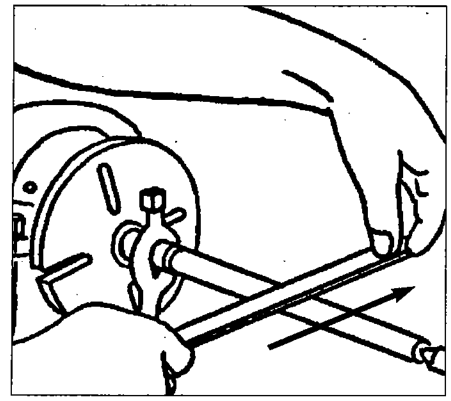
11.5 Finishing and polishing

After machining, you'll want a smooth, polished surface free of machine marks. The best results come from using a toolpost grinder. If you don't have one, use a file.



11.3 You can do other turning cuts with different cutter bits and cutting tools.

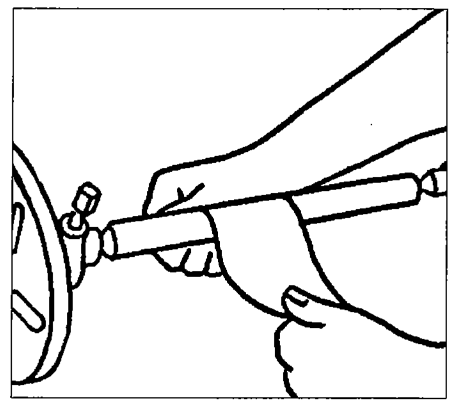
With a file, take full, biting strokes across the revolving workpiece at a slightly oblique angle. Do not drag the file back across the workpiece; instead, lift it clear for each return stroke. Use a clean, dry file and keep the workpiece clean, as well. Wipe the workpiece dry and clean if you've used coolant or cutting oil. Never hold the file stationary while the workpiece is revolving (*Figure* 11.4).



11.4 With a file, take full strokes at an oblique angle; never hold the file still.

For an even finer file finish, chalk the file by rubbing railroad chalk into its teeth. This provides additional lubrication and absorbs filings. Do not use blackboard chalk.

After filing off the machining marks, polish the workpiece with emery or other abrasive cloth. Keep the lathe turning at high speed and spread a few drops of oil on the workpiece. Don't stop moving the cloth (*Figure* 11.5).

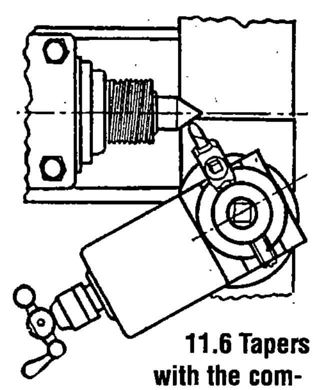


11.5 You can polish a workpiece with an abrasive cloth and oil.

11.6 Taper turning

There are two ways to turn a taper on the CB-1239 XL: with the compound rest and by setting over the tailstock. In both methods, the cutter must engage the work on dead center if the taper is to be accurate.

Compound rest. Tapers cut with the compound rest are usually short, abrupt angles, such as centers, bevel gear blanks, and die parts (*Figure* 11.6). In general, these are not considered taper turning, which applies to machining longer, more gradual tapers.



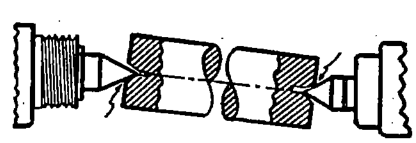
11.6 Tapers cut with the compound rest are usually short, abrupt angles.

Setting over the tailstock. Cutting tapers by setting over the lathe tailstock gives a misalignment of the lathe centers. The lathe centers move from their parallel position with the tool's transverse travel, allowing for the desired degree of taper (*Figure* 11.7). The tailstock of the CB-1239 XL has a set-over scale calibrated both forward and backward from the straight turning or zeroing point for measuring set-over distances.

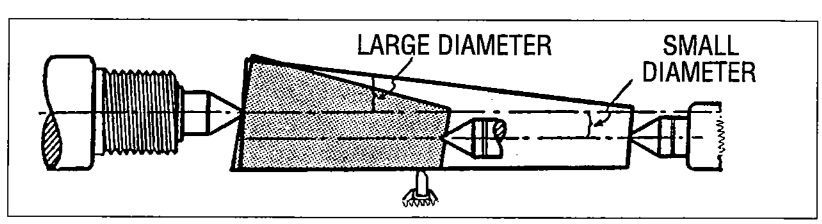
It is possible to turn long, gradual tapers by setting over the tailstock, but take care. Your computations must be nearly perfect, because an error will spoil your work.

The distance of tailstock setover needed to machine any given taper depends on three factors:

- The differential between the finished diameters of the extreme ends of the taper
- The length of the taper in relation to its extreme diameters, if the entire shaft is to be tapered
- The ratio between the length of the tapered portion to the entire length of the shaft (or work between centers) being machined, when only part of the shaft

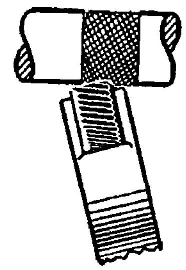


11.7 In setting over the tailstock, the lathe centers move from their parallel position with the tool's transverse travel.



11.8 Tailstock setover should be half the difference between the finished diameters of the ends.

1/64", reverse the lathe without removing the tool, and feed it back to the starting point. Feed both ways using the automatic longitudinal feed. Once across, each way, usually makes a good knurl.



12.3 Feed the knurling tool at a slight angle off from perpendicular to the line of the workpiece.

12.3 Screw threads

Because screw-thread cutting is so generally a part of machining, anyone interested in building things of metal should master it. Threading requires a bit of patience and skill that come through practice. Before attempting to cut a thread on anything you're actually making, cut a few practice threads on odd bits of steel, iron, and aluminum.

The CB-1239 XL is built for thread cutting and will cut standard internal and external threads, as well as special threads. You may cut coarse or fine threads in a great range of numbers of threads per inch, in V or square shapes, in established profiles like Unified National, acme, and metric. You can cut single threads or multiple threads that run concurrently along the shaft. You determine the type of thread by how you'll use the screw. Each thread form requires a different-shaped tool to cut or chase it.

For most work, the beginner will use the Unified National Standard (formerly the American National Coarse Thread) or American National Fine Thread (formerly SAE Standard). Both of these are V-form threads, slightly flat on top and at the root of the thread. Inch screw threads are usually referred to by pitch numbers, such as 18 or 24, meaning 18 or 24 threads per inch (tpi). The CB-1239 XL cuts standard threads in pitches from 11 to 40 tpi.

Because the lathe spindle, which carries the work, connects by gearing to the leadscrew, which moves the cutting tool along the lathe bed, a ratio exists between spindle speed in revolutions per minute and cutting tool movement in inches. When you change the gearing, you change this ratio. For this reason you can cut screw threads of various pitches by changing both the pick-off gears at the head of the lathe and the speed selection lever.

Thread charts on the CB-1239 XL show both inch and metric measures (*Figure* 6.3). The inch chart on the headstock shows the tpi from 11 to 40. The metric chart shows the distance from thread crest to crest from 0.12 to 3 mm.

With practice, you can grind cutters to almost any profile. It is difficult, however, to sharpen such cutters without altering the cutting form, and almost every resharpening requires a complete regrinding of profile and clearance angles.

12.4 Threading terms

In the early part of the century, rapid industrial development led to such a confusion of thread designs and standards that, in 1928, the US Congress authorized a National Screw Thread Commission to establish a standard system for screw threads. This led to the American National Screw Thread System and a uniformity in terminology.

- *Pitch*. Pitch is the distance from the center of a thread to the center of the next thread. To measure pitch in inches, measure an inch on a bolt and count the threads.

- *Pitch diameter*. This is the diameter of an imaginary cylinder superimposed on a straight screw thread, the surface of which would make an equal width of the thread and the spaces cut by the cylinder.

- *Lead*. The lead is the distance a screw thread advances axially (as through a nut) with one complete revolution. The lead and pitch of a single thread are identical, but they differ on multiple threads (the lead of a double thread is twice its pitch; of a triple thread, three times its pitch).

SECTION THIRTEEN
CHANGING GEARS

To change the gears on the CB-1239 XL, follow these steps.

Tools required: 17-mm wrench
10-mm wrench
Screwdriver (to remove C clips)
Pliers (to replace C clips)

- Remove all C clips and gears, starting with gear A and ending with gear D (*Figure* 13.1). With the 10-mm wrench, loosen the B & C gearshaft in its bracket by turning the gearshaft counterclockwise (*Figure* 13.2). This allows the shaft to slide freely along the bracket for easy gear removal and replacement.

- Select the proper A-D gear combination from the list outside the pulleybox door.

- Use the 17-mm wrench to loosen the nut at the bottom of the bracket assembly for full swing and easy gear replacement (*Figure* 13.3).

- Place the selected D gear on the D shaft, flange side in. Replace the C clip.

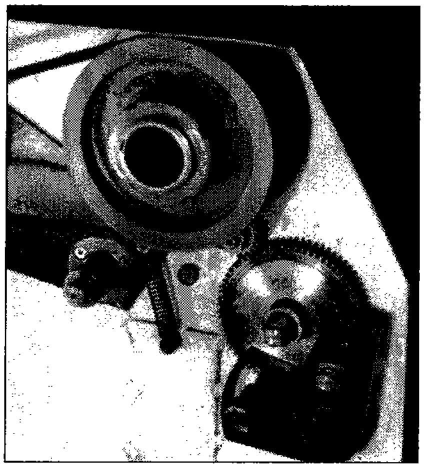
- Place the selected B gear, flange side in, on the B & C gearshaft.

- Place the selected C gear, flange side in, on the B & C gearshaft. Replace the C clip.

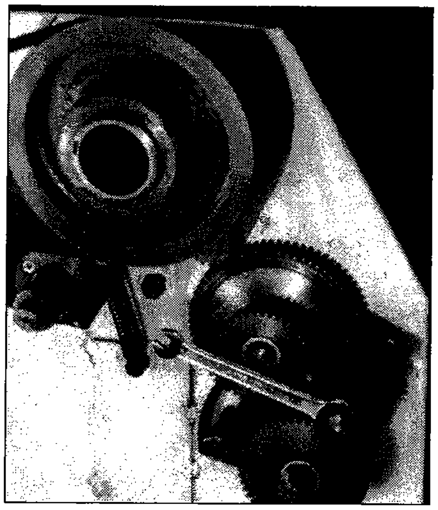
- Slide the B & C gearshaft until the C gear meshes properly with the D gear and tighten it with the 10-mm wrench (*Figure* 13.2).

- Place the selected A gear, flange side in, on the A gearshaft and replace the C clip.

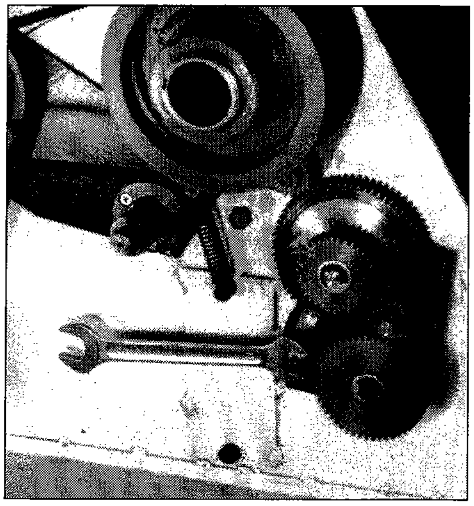
- Swing the bracket assembly until the A and B gears mesh. Hold the bracket assembly in place and tighten the nut (*Figure* 13.3). Make sure the gears turn smoothly while the half-nut is not engaged. You may need to make some adjustments.



13.1 Remove all C clips and gears, starting with gear A and ending with gear D.



13.2 Loosen the B & C gearshaft by turning it counterclockwise.



13.3 Loosen the nut at the bottom of the bracket assembly.

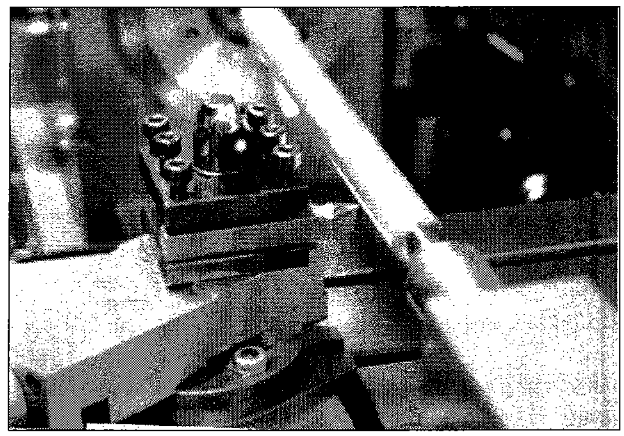
SECTION FOURTEEN

CUTTING SCREW THREADS

After turning the work to be threaded to the outside diameter of the thread and setting the gears for the desired thread, put a threading tool in the toolpost. Set it exactly on the dead center of the workpiece you'll be threading, using a center gauge to guide you.

To make sure your cutter is on dead center, place a credit card or shim between the cutter point and the workpiece. When the tool is on dead center, the credit card or shim will remain vertical. The advantage of a credit card is that it eliminates the possibility of chipping the cutter as the workpiece and cutter come together.

With the compound set perpendicular to the line of centers, rotate it 29-1/2° to the right (*Figure* 14.1). Place the center gauge on the point of the threading tool and feed the tool toward the workpiece (*Figure* 14.2). Adjust the tool so the edge of the gauge is exactly parallel to the workpiece. A slip of white paper held below the gauge will help check the parallel of the gauge to the shaft and the fit of the toolpoint in the V of the gauge. Placing the threading tool perpendicular to the surface of the workpiece assures a true-form thread

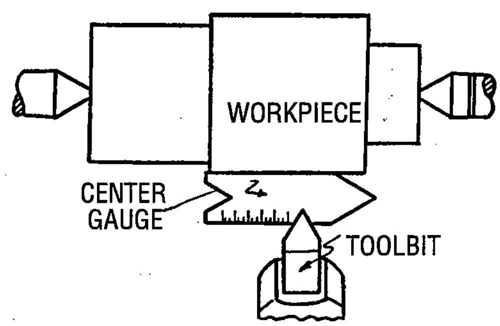


14.1 With the compound perpendicular to the line of centers, rotate it 29-1/2° to the right.

14.1 Cutting right-hand threads

Now you are ready to cut right-hand threads. First, advance the tool so it just touches the workpiece and turn the compound calibration back to zero. Then, using the compound feed, feed in the tool 0.002″. Next, turn on the lathe and engage the half-nut lever. *Do not disengage the half-nut until you are finished threading.*

It is best to take a light, scratch cut first without using cutting fluid. After the tool runs the desired length, turn off the lathe and back the tool out of the work. Then reverse the motor to return the tool to the starting position. Using a screw-pitch gauge, check the thread pitch. The benefit of taking the light cut is that you can correct any mistakes you might have made.

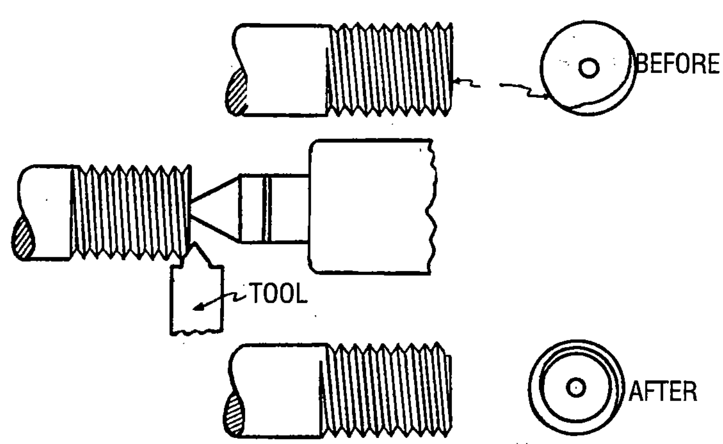


14.2 Using a center gauge, set the threading tool at exactly dead center on the workpiece.

It's time to take the real cut now, so apply the appropriate cutting fluid to the work. Feed the compound feed in 0.005 – 0.020″ for the first run, depending on the pitch of the thread you have to cut. If you are cutting a coarse thread, start by taking a few heavy cuts. Reduce the cut depth for each run until it is about 0.002″ at the final run.
Bring the cross-feed calibration to zero, then make the second cut.

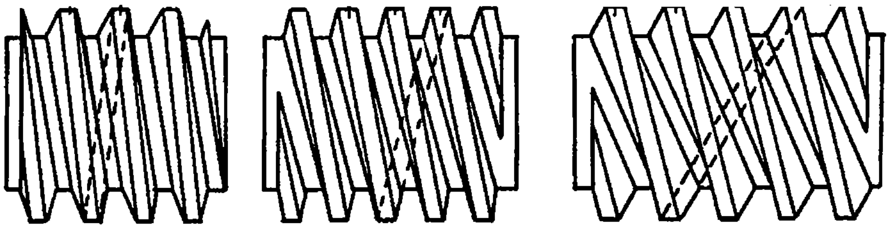
Continue this process until the tool is within 0.010″ of the finished depth. Brush the threads regularly to remove chips. After the second cut, check the thread fit using a ring gauge, a standard nut or mating part, or a screw thread micrometer. It is best to leave the piece in the chuck and not remove it for testing.

After returning the workpiece to the setup, continue taking 0.001 – 0.002″ cuts, checking the fit between each cut. When you thread the nut, it should go on easily but without end play. When you



14.3 Chamfer the end of the thread to protect it from damage.

have the desired fit, chamfer the end of the thread to protect it from damage. To chamfer is to take a 45° cut off the end of the bolt (*Figure* 14.3).



14.4 When cutting multiple threads, increase the lead to make room for succeeding threads.

14.2 What *not* to do when thread cutting

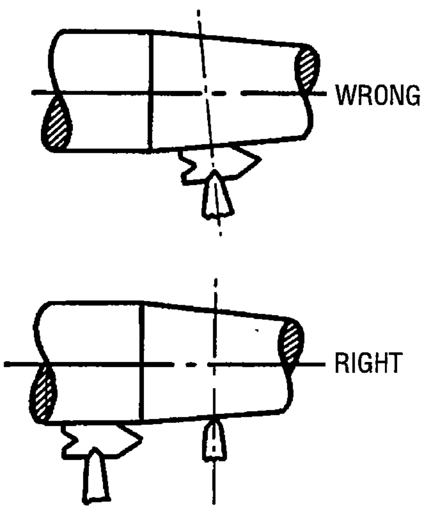
Do not disengage the half-nut. If you are cutting between centers, don't remove the lathe dog until the thread is finished and tested, and don't disturb the spindle while the work is off the centers. When you think the thread is about finished and ready for testing, and only if absolutely necessary, remove the workpiece from the center, leaving the lathe dog attached. Test the thread. If it does not fit properly and you have to remove another chip or two, place the workpiece back in the centers exactly as it was before. Then remove the chips and test again. Repeat until you are finished.

14.3 Finishing off a threaded end

After cutting a thread and before removing the threading tool, chamfer the end. This improves its appearance and removes sharp corners and burrs. It also aids the screw as it engages a nut or threaded hole.

14.4 Multiple threads

Cut multiple threads (*Figure* 14.4) one at a time exactly as you cut single threads, except increase the lead to make room for succeeding threads (a double lead for a double thread, a triple lead for a triple thread, etc.). After completing the first thread, remove the work from the centers without loosening the lathe dog. Then put it back in the lathe with the tail of the lathe dog in the correct slot to index the work for the next thread. This work requires a faceplate with accurately positioned slots, uniformly spaced and equal in number to the number of threads to be cut.



14.5 When cutting a thread on a taper, set the threading tool at 90° to the axis of the taper.

14.5 Cutting threads on a taper

Cut threads on a taper the same as on a straight shaft, except in the setup of the tool. Set the threading tool at 90° to the axis of the taper, rather than at 90° to its surface (*Figure* 14.5).

SECTION FIFTEEN
LATHE DRILLING AND BORING

You can lathe drill on the CB-1239 XL in two ways, holding the drill stationary and revolving the workpiece, or holding the workpiece stationary and revolving the drill. Holding the drill stationary in a tailstock chuck is the more accurate method; it gives a straighter hole (*Figure* 15.1). Without changing setup and recentering, the work is ready for any succeeding operations.

In lathe drilling, keep the drill sharp and properly ground. This is essential to get a straight, accurate hole.

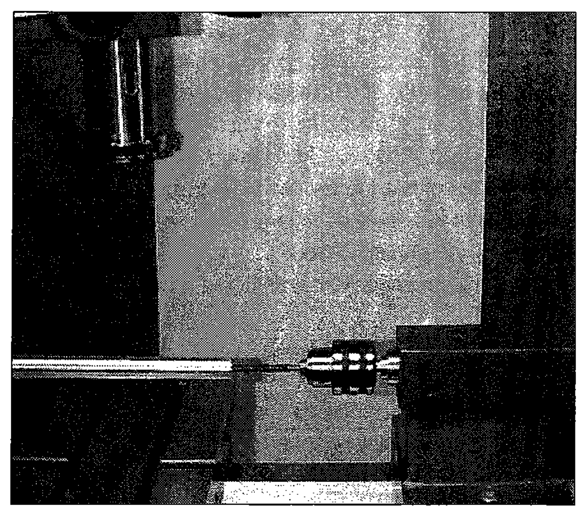
With HSS drills, operating speeds are not as critical as with carbon-steel drills. High speeds can quickly "burn" a carbon-steel drill. The number-of-feet-per-minute rule applies to drills even more than to other cutting edges because there is practically no air cooling of the point after it enters the hole. The larger the drill, the more peripheral feet cut per revolution. That is why you should use a slower drilling speed. If no drilling speed data are available, it's generally safe to run drills under 1/4″ diameter at up to 750 rpm and drills up to 1/2″ diameter at 500 rpm, with larger drills at proportionately slower speeds.

With the workpiece in the headstock and the drill in the tailstock chuck, feed the drill into the workpiece by advancing the tailstock ram. This is done by turning the tailstock handwheel. It is good to make a locating center for the drill point with a center punch, or even a countersunk center for large diameters, to keep the drill from creeping.

15.1 Reaming

When a hole must be accurate to 0.002″ or less, drill it slightly undersized (0.010 – 1/64″ on small diameters and 1/64 – 1/32″ on holes 1 – 2″ in diameter). Then ream it either by hand or in the lathe.

Lathe reaming is usually done with a solid reamer held in a tailstock chuck or with a taper shank that fits the tailstock ram in place of the tailstock center. At low speeds, feed the reamer into the workpiece slowly and evenly. Be sure the reamer teeth are free of burrs and chips.



15.1 Holding the drill in the tailstock gives a straighter hole.

15.2 Boring

Boring is internal turning, or turning from within. The diameter of the opening to be bored is often much smaller than its depth. Boring tools must therefore have relatively small diameters and still support a cutting edge projected at considerable distance from the toolpost or compound rest.

Boring tools consist of an extremely stiff, strong bar with a formed cutting end or a way to hold an HSS cutter or carbide insert. There are many sizes and types of boring bars. Choose the one that will give the stiffest possible bar at every depth and diameter and the greatest choice of cutters and cutter angles (ask a Smithy representative about the Smithy Boring Head Combo Package, #K99-125).

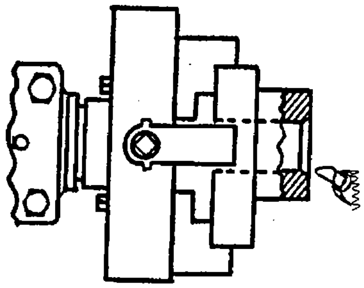
It is also wise to select tools with smooth-ended bars without a projecting nut or hardened edge that might mar the work (*Figure* 15.2). Most boring tools have only one cutting edge. There *are* double-end cutters, however, and they offer advantages in special instances. In grinding cutters, allow sufficient end rake to provide clearance from the internal diameter.

Except with cored castings, pipes, or tubing, boring starts with drilling a hole large enough to admit the end of the boring bar. Because the holes in cored castings often deflect boring bars off-center from their true axis, you may want to chamfer or turn out a starting cut in the opening of the hole with a turning tool (*Figure*

15.2 Tool with smooth-ended bar has no projecting nut or hardened edge.

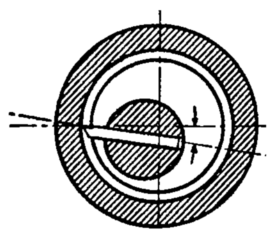
15.3) before introducing the boring tool.

With the boring toolholder set up in the toolpost or toolpost T-slot, select the largest diameter boring bar whose cutter the bore will accept. Extend the bar from the holder just enough to reach the full depth to be machined and still allow tool clearance. Except when using the adjustable boring tool (usually for very large-diameter work), feed the bar into the hole, parallel to the hole's axis. The cutter's cutting edge engages the work along a line in the mounted plane of the lathe centers with the bar positioned to give the cutter a top rake of approximately 14° from the radius at the cutting point. This takes into consideration the ground angle (top rake) of the cutter itself (*Figure* 15.4).

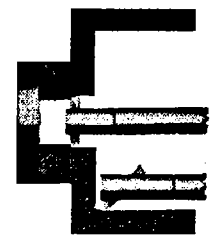


15.3 Chamfer a starting cut in the opening of the hole.

For straight longitudinal cuts, you can hold the cutter closer, therefore more rigidly, if it's at a 90° angle to the bar. For machining ends of a bar, however, you need a boring bar that holds the cutter at an angle or angles so the cutter extends beyond the end of the bar (*Figure* 15.5). For maximum visibility, position the cutting edge at the near side, parallel to the centerline.



15.4 The cutting edge engages the workpiece along a line in the mounted plane of the lathe center.



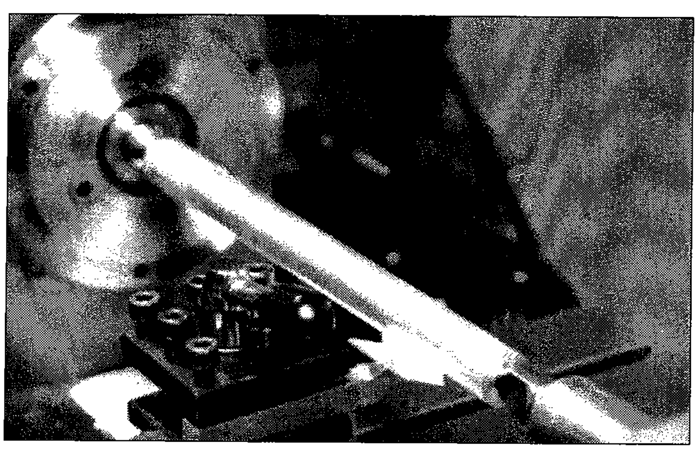
15.5 To machine ends of a bar, use a boring bar that angles the cutter so it extends beyond the bar.

The rules that apply to external turning apply to boring as well, except—as noted earlier—where the rake angles differ. They are governed by cutter type and bore diameter. Feeds are necessarily lighter to keep the tool from springing. This is especially true when enlarging an out-of-round hole, when you take several small cuts rather than one heavy cut. After the last finish cut, it is common to reverse the feed and take one last, fine cut with the tool coming out of the work. This last cut, taken without movement of the crossfeed, avoids a slightly undersized hole because you compensate for any spring in the bar.

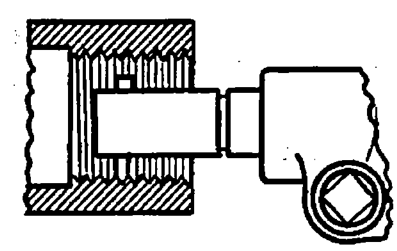
15.3 Cutting internal threads

The rules controlling internal thread cutting are much the same as for external thread cutting, except that in internal thread cutting, you have the clearance restrictions and tool problems of boring. You use the same toolholders, but the cutters have thread forms and are fed at thread-cutting ratios of feed to spindle revolutions. Another difference between boring and inside threading is the angle at which the cutter approaches the workpiece. As with external thread cutting, the internal threading tool must engage the work on dead center and be held so the cutter coincides with the center radius of the work.

In squaring the cutter with the work, use a center gauge (*Figure* 15.6) or thread gauge to correct any error in cutter alignment. Internal cutters require greater end and side clearance, and cutter length is also restricted because internal thread cutters need enough end clearance so the cutter lifts clear of the thread for removal (*Figure* 15.7). Before cutting an internal thread, bore the workpiece to the exact inside diameter.



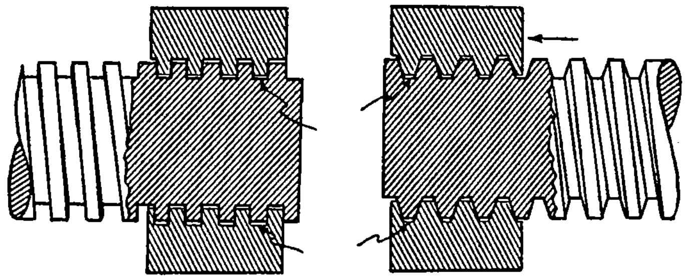
15.6 Use a center or thread gauge to correct cutter alignment error when squaring the cutter with the workpiece.



15.7 There must be enough end clearance for the cutter to lift clear of the thread.

Remember the feed of successive cuts is toward, not away from, the operator, and the thread-cutting set is therefore reversed. Also, cuts must be lighter, due to the extension of the cutter from the toolpost. Take an extra finishing cut without changing the setting of the compound rest.

15.4 Cutting special-form internal threads



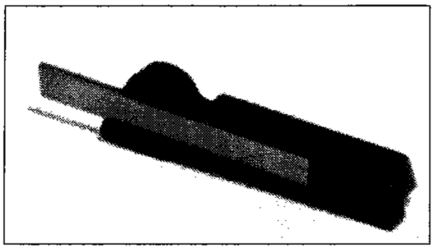
15.8 Use different clearances between nut and screw for different thread types.

You can cut internal forms in all thread forms used for external threads. There is only one factor that calls for special attention in cutting special-shaped internal threads: the difference of clearances between the nut and the screw recommended for different thread types (*Figure* 15.8). If these special recommended clearances are not available, it is safe to cut a nut thread (internal thread) 0.005—0.010″ per inch larger in the screw's outside diameter.

SECTION SIXTEEN

CUTTING OFF OR PARTING WITH A LATHE

You can cut off in a lathe only when one end of the work is held rigidly, as in a chuck. It is not practical for a long workpiece held between centers because the workpiece is not supported closely with a rest and the free section is long enough to sag and pinch the blade. Cutting off requires a tight lathe without excess play in the spindle, compound, carriage, or toolpost. Looseness will almost certainly cause chatter. It also requires a narrow cutting edge with ample (5 – 10°) side clearance, fed slowly into the workpiece to prevent hogging in. Once considered a difficult, costly operation, cutting off became much simpler with the development of narrow tools with special cut-off blades (*Figure* 16.1).



16.1 Narrow tools with special cut-off blades simplify cutoff.

The toolpost should hold the cut-off tool as close to the cutting end as possible, with the top of the blade on dead center and exactly perpendicular to the line of centers. Have the blade extend only far enough to pass through the workpiece, just beyond its center. The crossfeed should feed the tool slowly and evenly to the workpiece on exact center. If the tool hogs in and the spindle stops rotating, turn off the motor and reverse the spindle by hand before backing the tool out with the crossfeed.

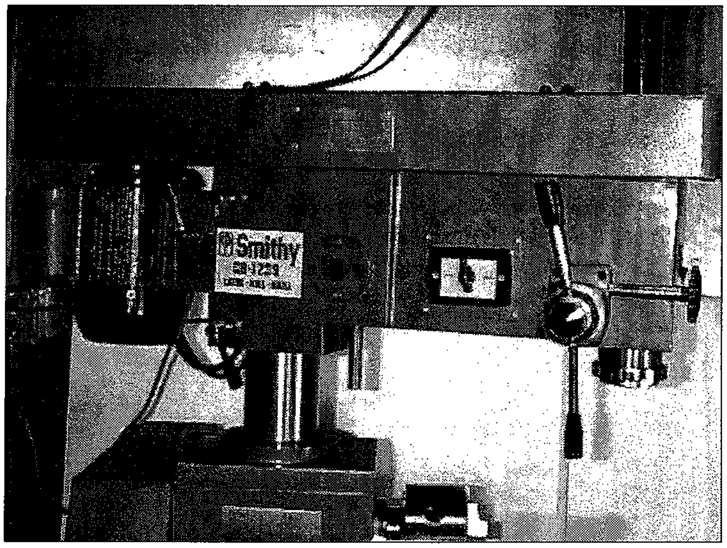
Always set up the workpiece to cut off as close to the cutting end as possible to the headstock. If you must make a parting cut on a long shaft or on work held between centers, never complete the cut in the lathe. Finish the parting with a hacksaw and return it to the lathe for facing. Slow the spindle speeds until you have a feel for cutting off. Although lubricants and coolants are not essential on small diameters, use them amply on deep cut-off work.

SECTION SEVENTEEN
MILLING

In milling, one or more rotating cutters shape a workpiece held by a vise or other fixture. The cutters mount on arbors or at the end of the spindle on collets or adapters.

Machinists use mills to machine flat surfaces, both horizontal and vertical, and to make shoulders, grooves, fillets, keyways, T-slots, and dovetails. They can also make curved and irregular surfaces and machine accurate holes. Its variety of machining operations and high metal-removal rates rank the mill in importance with the lathe.

The CB-1239 XL's millhead (*Figure* 17.1) rotates horizontally and moves vertically. A quill that moves in and out of the head carries the spindle.



17.1 The millhead rotates horizontally and moves vertically.

You can move the table horizontally in two directions by turning the cross-slide and long-feed handwheels located on the apron (*Figure* 4.2). The cross-slide handwheel turns the table longitudinally (at right angles to the spindle axis); the long-feed handwheel moves it transversely (parallel to the spindle axis).

To raise and lower the millhead, release the two lock levers and put the crank on the square-head gearshaft. To raise the millhead, turn it clockwise; to lower it, turn it counterclockwise.

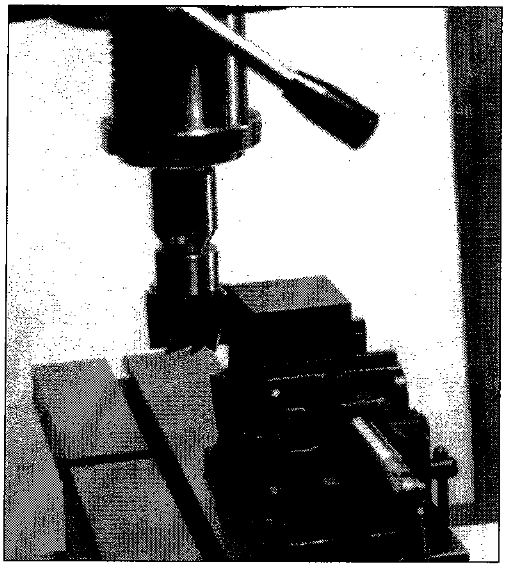
To rotate the millhead, release the two lock levers and push the millhead to the desired position.

SECTION EIGHTEEN
WORKHOLDING

The most common ways to hold a workpiece during milling are to secure it directly to the table via clamps or hold it in a vise (*Figure* 18.1). If you're making many similar workpieces, you may make a special fixture to hold them. Whatever method you use, hold the workpiece securely so it won't shift during machining and support it adequately to avoid swing.

18.1 Mounting to the table

If you need to align the workpiece to the table, place it against stops that exactly fit the table's T-slots. Another way is to measure in from the edge of the table to the workpiece. Be sure the table and workpiece are clean and free of burrs.



18.1 A common way to hold a workpiece during milling is to hold it in a vise.

18.2 Using a vise

Vise sizes are designated by the width of the vise jaw in inches or millimeters. *Plain* and *swivel* vises range from 3 to 10″ (76 to 254 mm). *Universal* and *tilting* vises range from 3 – 4″ to 5″ (76 – 102 mm to 127 mm).

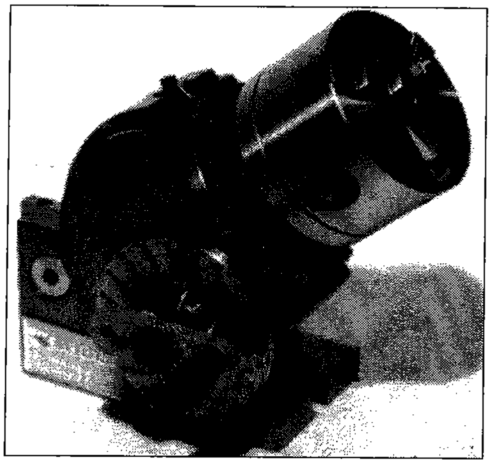
The bases of many vises are fitted with keys—small steel blocks that fit into the milling table T-slot for quick alignment of the vise. Before mounting a vise, make sure the bottom is clean and smooth. If there are any nicks or burrs, remove them with a honing stone. Set up the workpiece securely and correctly, and fasten the vise tightly to the table.

Plain vises have a flanged base with slots that lets them bolt to the table with the jaw faces either parallel to, or at 90° to, the longitudinal table travel. *Swivel vises* have a swivel base that bolts to the table. They're marked with degree graduations that let you position their jaws at any angle without moving the base. *Universal vises* tilt up or sideways, or swivel. They hold workpieces machined at a double or compound angle. *Tilting vises* are like universal vises except they do not tilt sideways.

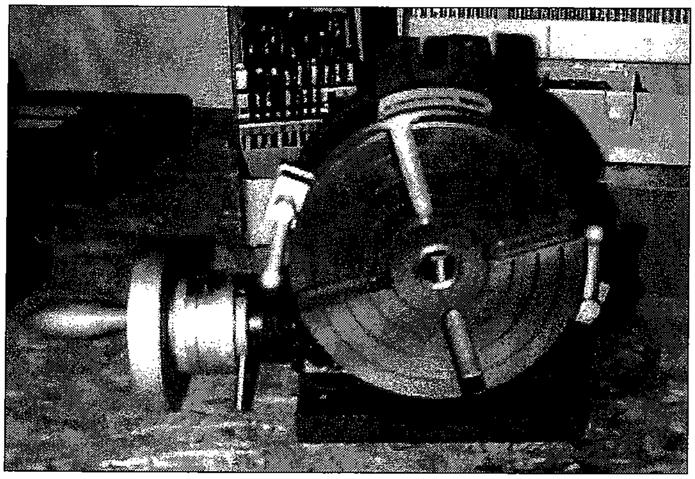
Using special fixtures. Clamp both workpiece and fixture securely in place. Be sure they are clean. Watch them carefully during machining; a loose fixture or workpiece can be disastrous.

18.3 Dividing heads

Also called indexing heads, dividing heads attach to the table to hold workpieces between centers, in chucks or on faceplates for



18.2 A chuck attaches to the spindle face of a dividing head.



18.3 Rotary tables let you cut gears, precision holes, and curved slots.

machining surfaces, grooves, or gear teeth at precise distances apart. The main parts of a dividing head are its head, tailstock, and indexing plates. The tailstock can hold the outer end of the workpiece (a shaft, for example). The head is more complex. When you turn its handle, a spindle rotates through a precisely machined gearing system. A chuck can be attached to the spindle face, which is set at 90° to the handle (*Figure* 18.2). An indexing plate is set in from the handle. By counting how many turns of the handle it takes to turn the workpiece a certain number of degrees, you can make cuts at different angles. This is how to cut gears.

18.4 Rotary tables

A rotary table (*Figure* 18.3) is a precision worm and wheel unit that lets you cut gears, precision holes, and curved slots. Rotary tables mount vertically or horizontally to the table. T-slots secure the workpiece. A typical rotary table is graduated in degrees and fractions.

The index plate on the rotary table has several circles of equally spaced holes into which the index crankpin fits. Although the hole circles are spaced equally, the number of holes varies in different circles, so you can get many different numbers of circumference divisions. You can buy sets of index plates for even more circumference divisions.

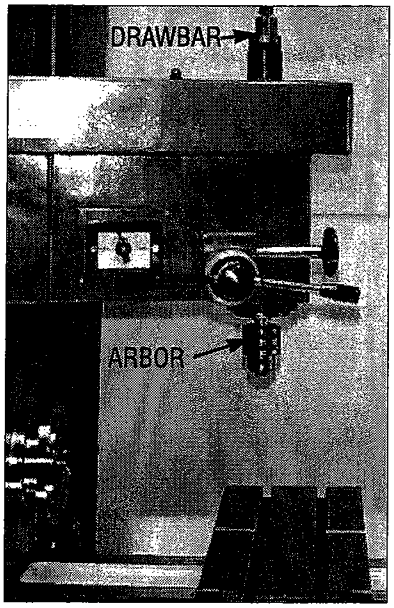
SECTION NINETEEN
HOLDING MILLING CUTTERS

There are several ways to hold milling cutters: in arbors, with collets and special holders, and in adapters.

19.1 Arbors

Arbors come in different sizes and lengths, with one end tapered to fit the bore in the end of the machine spindle. The arbor of the CB-1239 XL, which has an MT3 taper, is driven by friction between arbor and spindle. The arbor stays in place by means of a drawbar screwed into the end of the arbor from the top of the spindle (*Figure* 19.1).

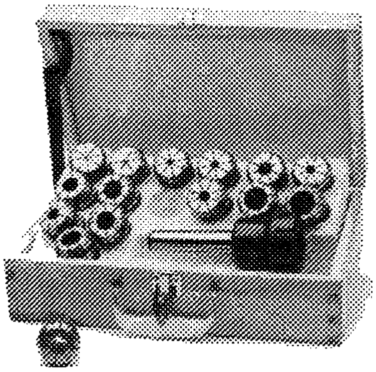
Take good care of your arbors. Store them in a rack or bin. If you won't be using them for several days or longer, oil them to prevent rusting, especially in damp weather.

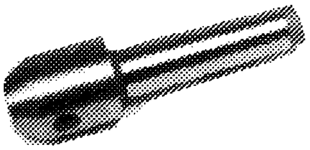


19.1 A drawbar holds the arbor in place.

19.2 Collets and holders

Straight-shank end mills fit into spring collets (*Figure* 19.2) or end mill holders (*Figure* 19.3). Their precision-ground shanks go into the mill spindle. When you tighten a spring collet, its hole reduces in size and the collet grips the end mill shank evenly. To use an end mill holder, tighten the end mill securely with the setscrew against the flat surface of the end mill, or it may slip out and damage the workpiece, the cutter, or you.


19.2 Spring collets, which fit into the mill spindle, hold straight-shanked end mills.


19.3 End mill holders also receive straight-shanked end mills.

19.3 Adapters

Adapters mount various types and sizes of cutters on the spindle. Arbor adapters mount face mills on the spindle. Collet adapters mount end mills on the spindle. Taper-shank end mills mount in adapters that have holes with matching tapers. If the taper shank on the tool is smaller than the hole in the adapter, put a reducing sleeve into the adapter (you may have to modify the sleeve). Shell end mill adapters come in different sizes to accept different-sized shell end mills.

To remove arbors or adapters held with a drawbar, follow these steps:

- Loosen the locknut on the drawbar about two turns.
- Hit the end of the drawbar with a dead blow hammer, releasing the arbor or adapter from the spindle hole.
- Hold the arbor or adapter so it won't fall out of the spindle when the drawbar is removed.
- Unscrew the drawbar and remove the arbor or adapter.

Your machines includes a tapered drift for removing tapers. Follow these steps:

- Remove the drawbar
- Extend the mill spindle to expose the outer taper drift slot.
- Rotate the spindle to align outer and inner taper drift slots. You will be able to see the end of the adapter through both slots.
- Insert the drift in the slot.

• Holding the adapter with one hand, use a nonmarring hammer (rubber, dead-blow, or brass) to drive the drift into the slot. The taper on the tool will release and the adapter drop out.

Cutters mounted in the spindle must fit accurately. Fit the shank of the arbor that carries the cutter directly into the taper hole at the end of the spindle. A drawbar holds the arbor in place.

SECTION TWENTY
MILLING CUTTERS

Choose milling cutters for the type of cut, the number of parts, and the material. Rake angles depend on both cutter and work material. Clearance angles range from 3° to 6° for hard or tough materials to 6° to 12° for soft materials.

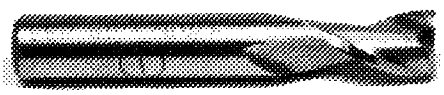
To determine the number of teeth you want, consider the following:

- There should not be so many teeth that they reduce the free flow of chips.
- The chip space should be smooth so chips don't clog.
- Don't engage more than two teeth at a time in the cut.

20.1 End mill cutters

End mill cutters cut on their ends and sides. They are either solid (cut from a single piece of material) or shell (separate cutter body and shank). They have two, three, four, or more teeth and may do right or left-handed cutting. Their flute twist or helix may also be right or left-handed. Solid end mills have straight or tapered shanks; shell end mill adapters have tapered shanks. End mills machine horizontal, vertical, angular, or irregular surfaces in making slots, keyways, pockets, shoulders, and flat surfaces.

- *Two-flute*, or center-cutting, end mills (*Figure* 20.1) have two teeth that cut to the center of the mill. They may feed into the work like a drill (called plunge milling), then go lengthwise to form a slot. Teeth may be on one end (single-end) or both ends (double-end).

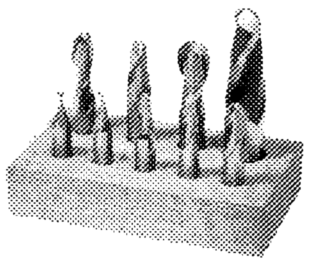


20.1 Two-flute end mills have two teeth that cut to the center of the mill.

- *Multiple-flute end mills* have three, four, six, or eight flutes and may be single or double-ended. Multiple-flute mills are center-cutting or non-center-cutting. *Don't* use noncenter-cutting end mills for plunge milling.

- *Geometry-forming end mills* form particular geometries. They include ball end mills, roughing end mills, T-slot cutters, keyseat cutters, and shell end mills.

Ball end mills (*Figure* 20.2) cut slots or fillets with a radius bottom, round out pockets and bottoms of holes, and do diesinking and diemaking. Four-fluted ball end mills with center cutting lips are available.



20.2 Ball end mills cut slots or fillets with a radius bottom.

Roughing end mills remove large amounts of metal rapidly with minimum horsepower. They have three to eight flutes. Also called hogging end mills, they have wavy teeth on their periphery that provide many cutting edges, minimizing chatter.

T-slot cutters cut T-slots. After machining a groove for the narrow part of the T-slot with an end or side mill, finish up with the T-slot cutter.

Keyseat cutters cut keyseats for Woodruff keys (shaped like a half circle).

Shell end mills (*Figure* 20.3), which mill wide, flat surfaces, have a hole for mounting on a short arbor. The center of the shell is recessed to provide space for the screw or nut that fastens the cutter to the arbor. The teeth are usually helical, and diameters go up to 6″ (152 mm).

Insert-type end mills use replaceable HSS or carbide inserts. Small end mills use two inserts; larger end mills, three or more.

Face milling cutters start in size at 2″ (51 mm) and have inserted teeth on the periphery and face. Most of the cutting takes place on the periphery. They are similar to, but larger than, shell end mills.

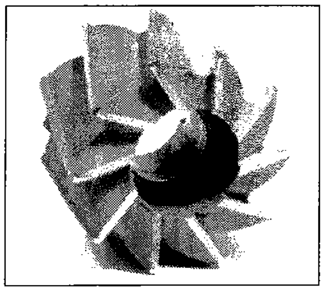
20.2 Plain milling cutters

Plain milling cutters have teeth only on their periphery. Used to mill plain, flat surfaces, they may combine with other cutters to produce various shapes. They are cylindrical and come in many widths and diameters.

• *Light-duty plain cutters* for light cuts and fine feeds come in two forms. Narrow ones have straight teeth parallel to the cutter axis. Wide ones have helical teeth at a 25° angle. Features include ease of starting cuts, little chatter, and good surface finishes.

• *Heavy-duty plain cutters*, or coarse-tooth cutters, come in larger widths and have larger and fewer teeth. Strongly supported cutting edges and wide flutes provide strength and space for heavy chip removal. The helix angle of their teeth is 25 – 45°.

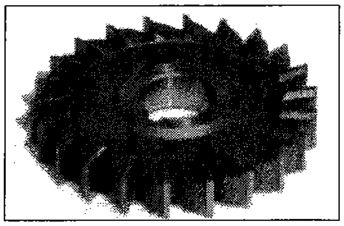
• *Helical plain milling cutters* have even fewer and coarser teeth with a helix angle of 45 – 60° or greater. These cutters are for wide, shallow profiling cuts on brass or soft steel.



20.3 Shell end mills mill wide, flat surfaces and mount on arbors.

20.3 Side milling cutters

Similar to plain milling cutters, they also have teeth on one or both sides (*Figure* 20.4). The teeth on the periphery do most of the cutting; those on the sides finish the side of the cut to size. They cut grooves or slots, and often work with other cutters to mill special shapes in one operation.

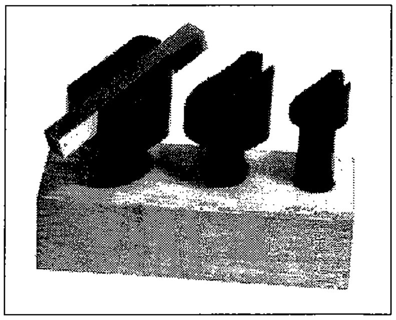


20.4 Side milling cutters are similar to plain milling cutters, but they have teeth on one or both sides.

• *Plain side milling cutters* have straight teeth on the periphery and both sides. Side teeth taper toward the center of the cutter, giving side relief or clearance.

• *Half side milling cutters* have helical teeth on the periphery and one side. These cutters do heavy-duty face milling and straddle milling where teeth are needed on only one side. The side teeth are deeper and longer for more chip clearance.

• *Staggered-tooth side milling cutters* are narrow cutters with teeth alternating on opposite sides. There is less dragging and scoring and more space for chip removal. These cutters do heavy-duty keyway and slotting cuts.



20.5 Flycutters take light face cuts from large surface areas.

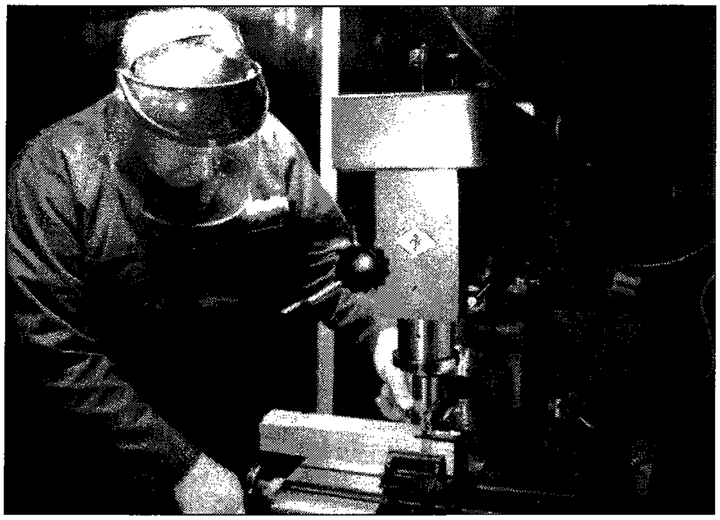
20.4 Slitting saws

Slitting saws do narrow slotting and cut-off operations.

• *Plain slitting saws* are thin plain milling cutters with only peripheral teeth. The teeth are fine, and the sides taper slightly toward the hole, giving side relief.

• *Slitting saws with side teeth* are like side milling cutters and are for deeper slotting and cut-off operations normally done with plain slitting saws.

• *Staggered-tooth slitting saws* have peripheral teeth with alternate right and left-hand helix and alternate side teeth. They are for 0.2″ (4.8-mm) and wider cuts and may do deeper cuts with standard feeds.



20.6 Protect yourself with face mask and protective clothing when using compressed air.

• *Screw-slotting cutters* are plain slitting saws with fine-pitch teeth that cut slots in screwheads. Their sides are straight and parallel and offer no side relief.

20.5 Angle milling cutters

Angle milling cutters, for such operations as cutting V-grooves, dovetails, and reamer teeth, come as single and double-angle cutters.

• *Single-angle cutters* have one angular surface. Teeth are on the angular surface and the straight side, and they usually have 45° or 60° angles.

• *Double-angle cutters* machine V-grooves. Those with equal angles on both faces usually have an included angle of 45°, 60°, or 90°.

20.6 Form-relieved cutters

Formed-tooth cutters machine surfaces with curved outlines. You can sharpen them without changing the tooth outline. Concave cutters mill convex half-circles; convex cutters cut concave surfaces.

• *Corner-rounding cutters* round outside corners.

• *Gear cutters* cut gear teeth.

• *Fluting cutters* cut flutes in reamers and milling cutters.

• *Formed-tooth cutters* come in right and left-hand styles and various special shapes.

20.7 Flycutters

With one or more single-point toolbits or cutters, flycutters (*Figure* 20.5) do end milling even though they're not end mills. They take light face cuts from large surface areas. You must grind the toolbit properly to get correct rake and clearance angles. Grind toolbits for flycutters as you grind lathe tools (see *Section* Seven). You can also use flycutters for boring. When the tool revolves, the cutting tool becomes almost invisible, so be careful.

20.8 Using cutting fluid

Cutting fluids get rid of heat generated by the friction of the milling cutter against the workpiece. They also lubricate the interface between the cutting edge and the workpiece and flush chips away. You can apply fluid in a stream (flood) or as a mist.

We recommend cutting fluids for steel, aluminum, and copper alloys. They tend to reduce the life of carbide tools in milling cast iron and steel, however, leaving tiny cracks along the cutting edge. Follow the advice of tool manufacturers. Materials such as cast iron, brass, and plastics are often machined dry. You can use compressed air to cool tools and clear chips away. When doing so, wear protective clothing (*Figure* 20.6), and keep cast-iron dust from between the lathe and carriage ways.

20.9 Tool grinding

Sharpen cutting tools when they become dull, or extreme forces may build up at the cutting edge of the teeth, causing chipping or fracture. Dull cutters are also inefficient, and regrinding very dull cutters shortens their life considerably.

The form of the cutting edge and the clearance back of the cutting edge (land) affect cutter operation significantly. The angle formed by the land and a line tangent to the cutter at the tooth tip is the primary clearance. The angle between the back of the land and the heel of the tooth is the secondary clearance. Check both clearances and the rake.

Some cutters are sharpened on the periphery by grinding the land at a suitable angle. They include cutters with straight or spiral teeth, angular cutters, side milling cutters, face mills, end mills, and reamers. Others are sharpened by grinding the front faces of their teeth. Formed or relieved cutters, for example, have profiles that must be preserved. This category includes all sorts of formed cutters as well as cutters used for milling various regular and irregular shapes.

SECTION TWENTY-ONE
SPEEDS AND FEEDS FOR MILLING

21.1 Speeds

Milling cutting rates vary according to the machinability of the material being cut; whether cutting fluid is used and, if so, what kind; the type, size, and material of the cutter and the coarseness of its teeth; and the amount of metal being removed. Cutting speed for milling is the distance the cutting edge of a tooth travels in one minute. If cutting speed is too high, the cutter overheats and dulls. If it's too low, production is inefficient and rough.

There is no exact right cutting speed for milling a particular material (*Figure* 21.1). Machinists usually start with an average speed, then increase or decrease it as appropriate. For light cuts, use the upper end. Use the lower end for heavy cuts and when

21.1 Recommended Cutting Speeds for Milling (fpm)

Material	Brinell Hardness	High-Speed-Steel Cutters	Carbide Cutters
Free-machining low-carbon steel resulphurized 1111	100-150	120-160	400-600
1112	150-200	120-180	400-900
Free-machining low-carbon steel leaded 10L18	100-150	100-225	250-500
12L14	150-220	110-250	250-600
Plain low-carbon steels 1006	100-125	80-150	300-600
1026	125-175	80-140	250-500
Plain medium-carbon steels 1030	125-175	80-140	250-500
1052	175-225	70-130	225-400
Plain high-carbon steels 1060	125-175	70-120	250-450
1095	175-225	60-110	225-400
Tool steels, W1-W7	150-200	80-120	300-350
Tool steels, H20-H43	200-250	40-85	175-300
Tool steels, D1-D7	200-250	30-60	100-200
Stainless steel 302	135-185	70-100	225-350
Stainless steel 430F	135-185	100-140	350-450
Gray cast iron, ASTM class 20 Through scale	110-160	140-200	350-700
Under scale	———	130-225	400-800
Aluminum, cold-drawn wrought alloys	———	500-800	1000-1800
Aluminum, casting alloy (as cast)	———	600-1000	1200-2000
Brass, 360 free-cutting, cold-drawn	———	300-500	600-1000
Bronze, 220 commercial, annealed	———	80-140	180-275

you don't use cutting fluid.

Determining rpm. To set the spindle speed, you have to know the cutter rpm (revolutions per minute). For inch measurement, use the following formula:

$$rpm = 12 \times CS\ (fpm) / D'' \times \pi$$

where CS = cutting speed, fpm = feet per minute, D″ = diameter of the cutter in inches, and $\pi = 3.14$. For metric measurement, use this formula:

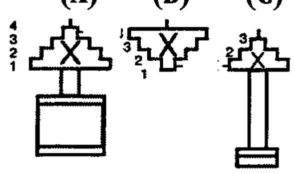
$$rpm = CS\ (mpm) \times 1000 / D\ (mm) \times \pi$$

where CS = cutting speed, mpm = meters per minute, D (mm) = diameter of the cutter in millimeters, and $\pi = 3.14$. You can use an rpm chart for selected diameters of cutting tools at different cutting speeds.

To change speeds, set the belts according to *Figure* 21.2.

21.2 Gearing for Various Milling Speeds

Motor (A) Idler (B) Spindle (C)

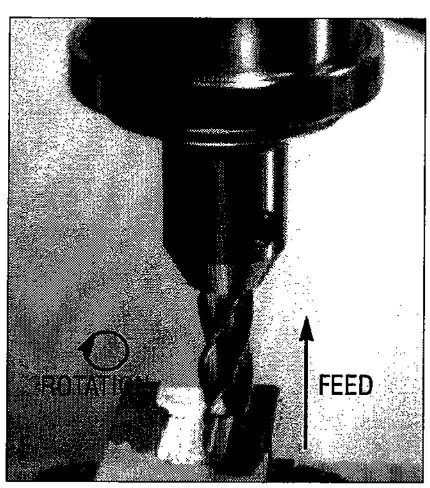


RPM					
$\frac{A4}{B4} \times \frac{B1}{C1}$	315	$\frac{A3}{B3} \times \frac{B1}{C1}$	630	$\frac{A2}{B2} \times \frac{B3}{C3}$	1250
$\frac{A4}{B4} \times \frac{B2}{C2}$	400	$\frac{A2}{B2} \times \frac{B1}{C1}$	800	$\frac{A1}{B1} \times \frac{B2}{C2}$	1600
$\frac{A4}{B4} \times \frac{B3}{C3}$	500	$\frac{A3}{B3} \times \frac{B2}{C2}$	1000	$\frac{A1}{B1} \times \frac{B3}{C3}$	2000

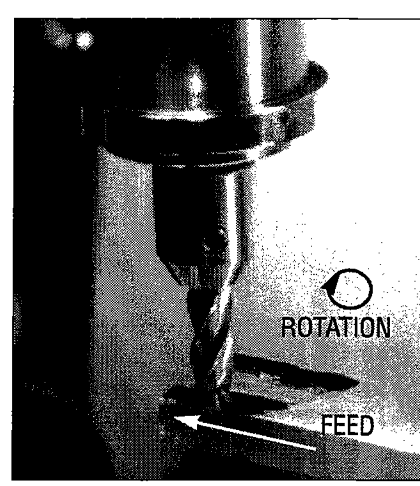
21.2 Feeds

Set the direction of feed before you begin milling. Set the crossfeed by hand. To set the powerfeed, remove the lathe chuck and turn on the lathe. You can now go forward or in reverse. Be careful to keep your clothes out of the lathe spindle.

Up milling, or conventional milling, is when the direction of feed is opposite to the direction of cutter rotation (*Figure* 21.3). Down milling, or climb milling, is when the direction of feed is the same as the direction of cutter rotation (*Figure* 21.4).



21.3 In up milling, the direction of feed is opposite the direction of cutter rotation.



21.4 In down milling, the direction of feed is the same as the direction of cutter rotation.

Up milling. In up milling, forces on the workpiece tend to pull it out of the vise or fixture holding it, so fasten it securely. These forces also push the workpiece away from the cutter, which eliminates backlash. Up milling is advised for milling cast iron, softer steels, and other ductile materials. In general, it's how you should perform milling operations.

Down milling. Down milling usually produces good surface finishes because chips do not sweep back into the cut. Setups are more rigid, an advantage when cutting thin workpieces held in a vise or workpieces held in a magnetic chuck. Down milling also produces straighter cuts. We recommend down milling when using carbide cutters because there is less wear on the cutting tool. In general, however, avoid it because of the backlash problems associated with it.

Feed rates. Your feed rate should be as high as your machine, cutting tool, workholding method, and workpiece can tolerate while giving a good finish. Feed rate is usually given in inches per minute (ipm).

You determine feed rate by the speed of the cutter in rpm and the number of teeth in the cutter. There are many factors to consider in selecting the feed per tooth, and there is no easy formula to follow. Here are several principles to guide you:

- Use the highest feed rate conditions allow.
- Avoid using a feed rate below 0.001″ per tooth.
- Harder materials require lower feed rates than softer materials.
- Feed wider, deeper cuts more slowly than narrow, shallow cuts.
- Slower feed rates gives a better surface finish.
- Never stop the feed before finishing the cut..

If you know the feed in inches per tooth, use the following formula to calculate table feed rate in inches per minute (ipm):

$$ipm = ipt \times N \times rpm$$

where ipt = inches per tooth, N = number of teeth in the milling cutter, and rpm = spindle speed of the milling machine.

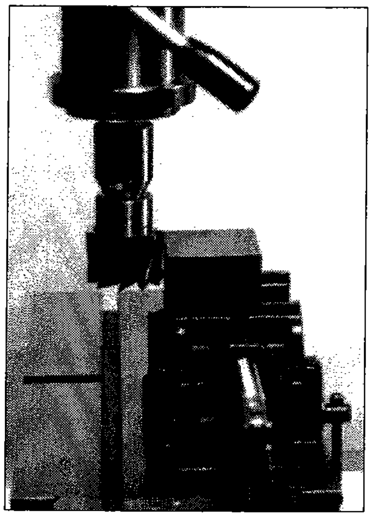
SECTION TWENTY-TWO
COMMON MILLING OPERATIONS

22.1 Milling flat surfaces

One way to mill a flat surface is by plane milling (*Figure* 22.1). Adjust the milling cutter vertically to give the needed depth of cut while the workpiece is held on the table and slowly feed it horizontally. Every tooth on the periphery of the cutter removes a chip every revolution. Milling wide, flat surfaces this way is called slab milling.

Another way to mill flat surfaces is by face milling. In this method, the cutter teeth operate at right angles to the cutter axis. Inserted-tooth face-milling cutters (*Figure* 22.2) face-mill large surfaces.

Bevels and chamfers are cut at an angle to the main workpiece surface. A bevel cut (*Figure* 22.3) goes from side to side, completely removing the perpendicular edge. A chamfer (*Figure* 14.3) removes only part of the perpendicular edge.



22.1 In plane milling, the workpiece feeds horizontally to the stationary cutter.

22.2 Inserted-tooth face-milling cutters face-mill large surfaces.

To cut bevels and chamfers, either move the workpiece into an angular cutter or hold the workpiece at the desired angle while moving it into a plain cutter or end mill. You may hold the workpiece in a vise or in a fixture held in a vise.

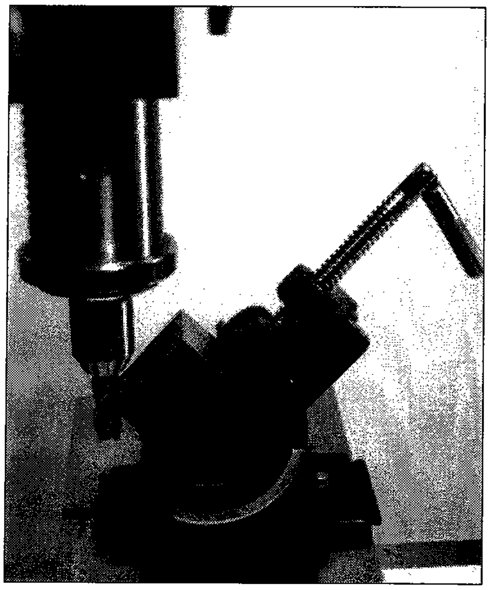
22.2 Squaring a workpiece

To square the ends of a workpiece, use the peripheral teeth of an end mill. If you want to remove a lot of material, use a roughing end mill first, then finish to size with a regular end mill.

Plunge cutting is efficient for removing material quickly on low horsepower. Plunge the end mill a predetermined width and depth, retract it, then advance and plunge it again repeatedly. The maximum cutting force is in the machine's strongest (axial) direction.

22.3 Milling a cavity

After laying out the outline of the cavity to cut, rough it out to within 0.030″ of the finished size before making finish cuts. Use a center-cutting end mill for the starting hole.



22.3 Bevel cuts go from side to side, removing the perpendicular edge.

22.4 Tapping

Drill a hole. Then remove the drill bit and put a tap into the chuck. By turning the chuck slowly by hand with slight downward pressure, you can get a perfectly threaded hole.

SECTION TWENTY-THREE
TROUBLESHOOTING

23.1 Powerfeed and thread cutting

Powerfeed does not move carriage

Cause	Solution
• Carriage locked	Unlock carriage
• Speed selector not engaged	Select Speed I or II
• Sheared pin (see *Section* 24.5, *part* 35)	Replace pin
• Gears not meshing or teeth missing	Check gears and mesh

Cut is not smooth

Cause	Solution
• Tool dull	Sharpen or replace tool
• Tool not on center	Center tool (shim if needed)
• Tools not mounted tight in post	Remount tools
• Cross-slide gibs to bed and base loose	Adjust gibs
• Gibs in toolpost loose	Adjust gibs
• Tool turret not tight	Tighten toolpost
• Feed rate too fast	Install correct gears
• Gears loose	Tighten gears and posts

Thread is not smooth

Cause	Solution
• Tool dull	Sharpen tool
• Tool not centered	Center tool
• Tools not mounted tight in post	Remount tools
• Cross-slide gibs to bed and base loose	Adjust gibs
• Gibs in compound loose	Adjust gibs
• Tool turret not tight	Tighten toolpost
• Gears loose	Tighten gears and posts

Tool not cutting "on thread"

Cause	Solution
• Half-nut not fully engaged	Keep half-nut engaged

23.2 Carriage/milling table

Table won't move

Cause	Solution
• Table locks engaged	Loosen locks
• Gibs too tight	Loosen gibs

Horizontal movement in cross-slide table

Cause	Solution
• Carriage gib improperly adjusted	Adjust carriage gib
• Table gib improperly adjusted	Adjust table gib

Vertical movement in cross-slide table

Cause	Solution
• Carriage gib improperly adjusted	Adjust carriage gib
• Table gib improperly adjusted	Adjust table gib

Carriage moves smoothly in only one direction

Cause	Solution
• Debris on way or gib	Remove debris
• Burr on gib	Remove burr with fine file
• Gib improperly tensioned	Loosen gib and re-tension
• One or more wipers mounted too low	Reposition wiper(s)

Cross-slide handwheel turns during cutting operations

Cause	Solution
• Cross-slide brass nut worn	Tighten or replace brass nut
• Carriage locks not tight	Tighten carriage locks
• Gibs too loose	Readjust gibs

Too much backlash in the cross slide

Cause	Solution
• Loose screw (see *Section* 24.4, *part* 36)	Tighten screw, review how to eliminate backlash
• Loose brass nut	Put a shim between the stud on the nut and the side of the hole
• Worn brass nut	Replace brass nut or adjust screw at end of nut
• Excessive space between bearing and dial	Add shim washers

23.3 Lathe turning

Cut is rough

Cause	Solution
• Tool dull	Sharpen or replace tool
• Tool not ground properly	Regrind tool
• Tool at wrong angle	Correct tool position
• Tools not held tightly	Tighten toolholder
• Wrong cutter for material	Use correct cutter
• Cutting speed incorrect	Increase or reduce speed

Work has unwanted taper

Cause	Solution
• Work improperly aligned	Realign centers on work
• Debris in spindle, setup, or tools	Clean and reset setup, work, or tool
• Offset tailstock incorrectly positioned	Correct position of tailstock
• Spindle out of alignment	Tighten taper bearings to return to alignment, replace spindle bearing

Machine vibrates

Cause	Solution
• Work mounted wrong	Remount work
• Speed too high	Reduce speed
• Too much pressure at tailstock	Reduce pressure and increase lubrication

Work stops turning but machine continues to run

Cause	Solution
• Work not mounted securely	Remount work
• Tools forced into work	Reduce force on tools
• Belts slipping	Tension belts, use belt dressing, or replace belts

Diameter of work not consistent

Cause	Solution
• Too much flex in workpiece	Use a follow rest
• Too much flex in compound rest, cross slide, or carriage	Tighten gibs, clean ways

Too much backlash in the compound

Cause	Solution
• Loose spanner nuts (see *Section* 24.12, *part* 11)	Tighten spanner nuts
• Worn nut	Replace nut

Slinging oil from behind the chuck or in belt box

Cause	Solution
• Oil reservoir overfilled	Check oil level
• Worn oil seal	Replace felt in seal

23.4 Milling

Tool chatters

Cause	Solution
• Gibs too loose on cross slide, compound,	Readjust gibs

Cause	Solution
or carriage	
• Unused feeds not locked	Lock all axes but the one moving
• Millhead not locked	Lock millhead
• Quill too loose	Tighten quill lock
• Tool not on center	Center tool
• Improper tool shape, tool dull	Reshape, sharpen, or replace tool

Depth of cut not consistent

Cause	Solution
• Quill moving	Lock quill
• Setup wrong	Make sure setup is parallel to table

23.5 Drilling

Hole off center or bit wanders

Cause	Solution
• Bit dull	Use sharp bits
• Bit not mounted correctly in chuck	Remount tool
• Bit bent	Replace bit
• Chuck loose in spindle	Remount chuck on arbor
• Drawbar not secured	Tighten drawbar
• Debris on spindle	Clean debris and arbor and remount tool
• Bearings loose or worn	Tighten or replace bearings
• Cutting too fast	Reduce speed
• Incorrect bit	Use correct bit

Entrance hole out of round

Cause	Solution
• Bit dull	Use sharp bits
• Incorrect drill bit	Use correct bit

Bit turns erratically or stops

Cause	Solution
• Bit fed into work too fast	Reduce feed rate
• Belts slipping	Reduce feed rate, re-tension belts

Chuck difficult to tighten or loosen

Cause	Solution
• Chuck sticking	Apply lubricant
• Debris in chuck	Clean chuck

Chuck wobbles

Cause	Solution
• Chuck loose on arbor	Clean arbor and remount

Cause	Solution
• Drawbar not tight	Clean spindle and replace drawbar

23.6 Drive system

Turn on machine and nothing happens

Cause	Solution
• Machine unplugged	Plug in machine
• Loose electrical connections	Tighten wiring connections

Notes

SECTION TWENTY-FOUR
PARTS LIST WITH SCHEMATICS

24.1 Mill/drillhead

Diagram 1

24.2 Mill fine-feed assembly

Diagram 2

24.3 Adjustable tailstock

Diagram 3

24.4 Bed and cross-slide table

Diagram 4

24.5 Leadscrew assembly

Diagram 5

24.6 Lathe head

Diagram 6

24.7 Gearbox

Diagram 7

24.8 Mill motor

Diagram 8

24.9 Lathe motor

Diagram 9

24.10 Lathe rest

Diagram 10

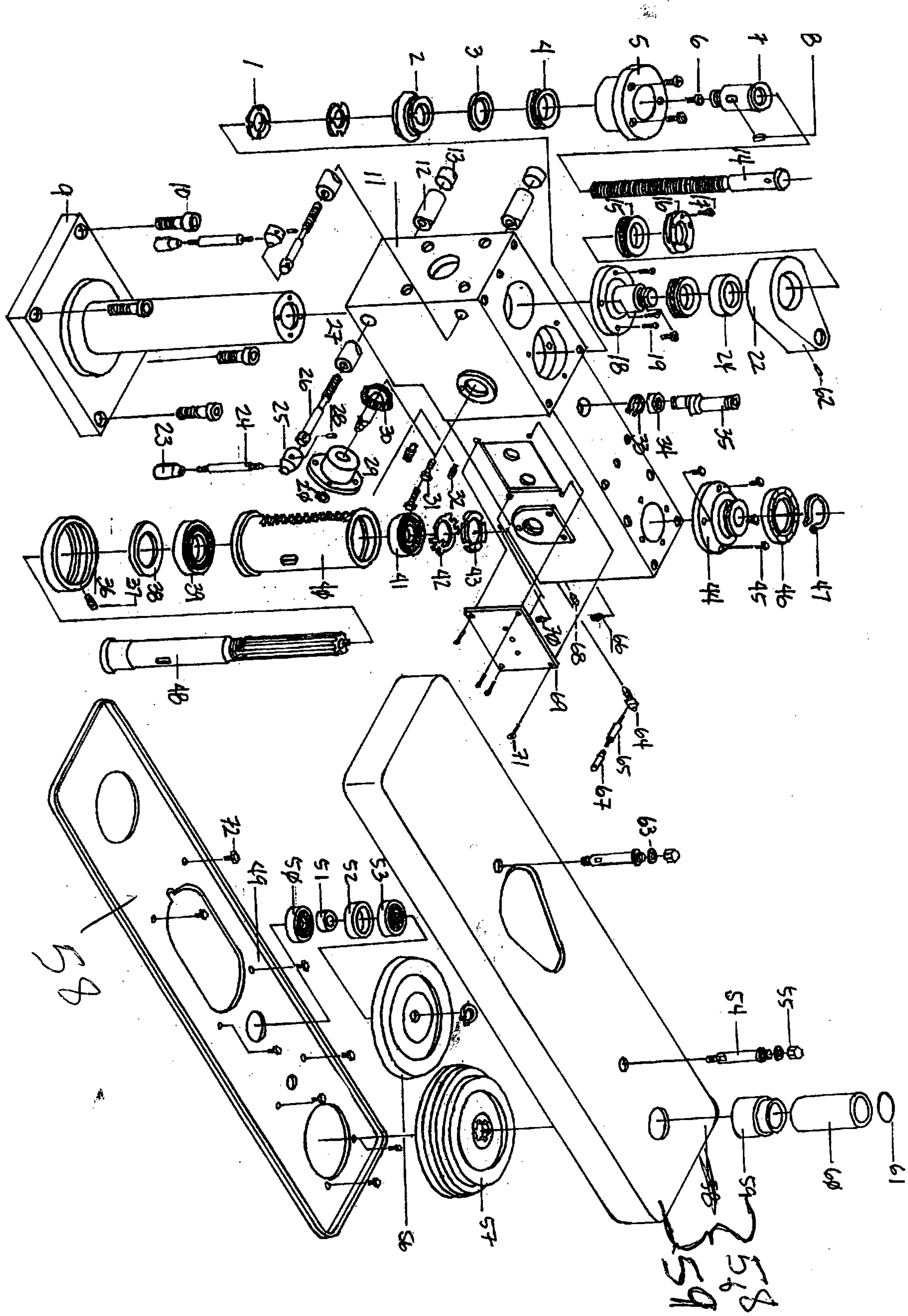
24.11 Follow rest

Diagram 11

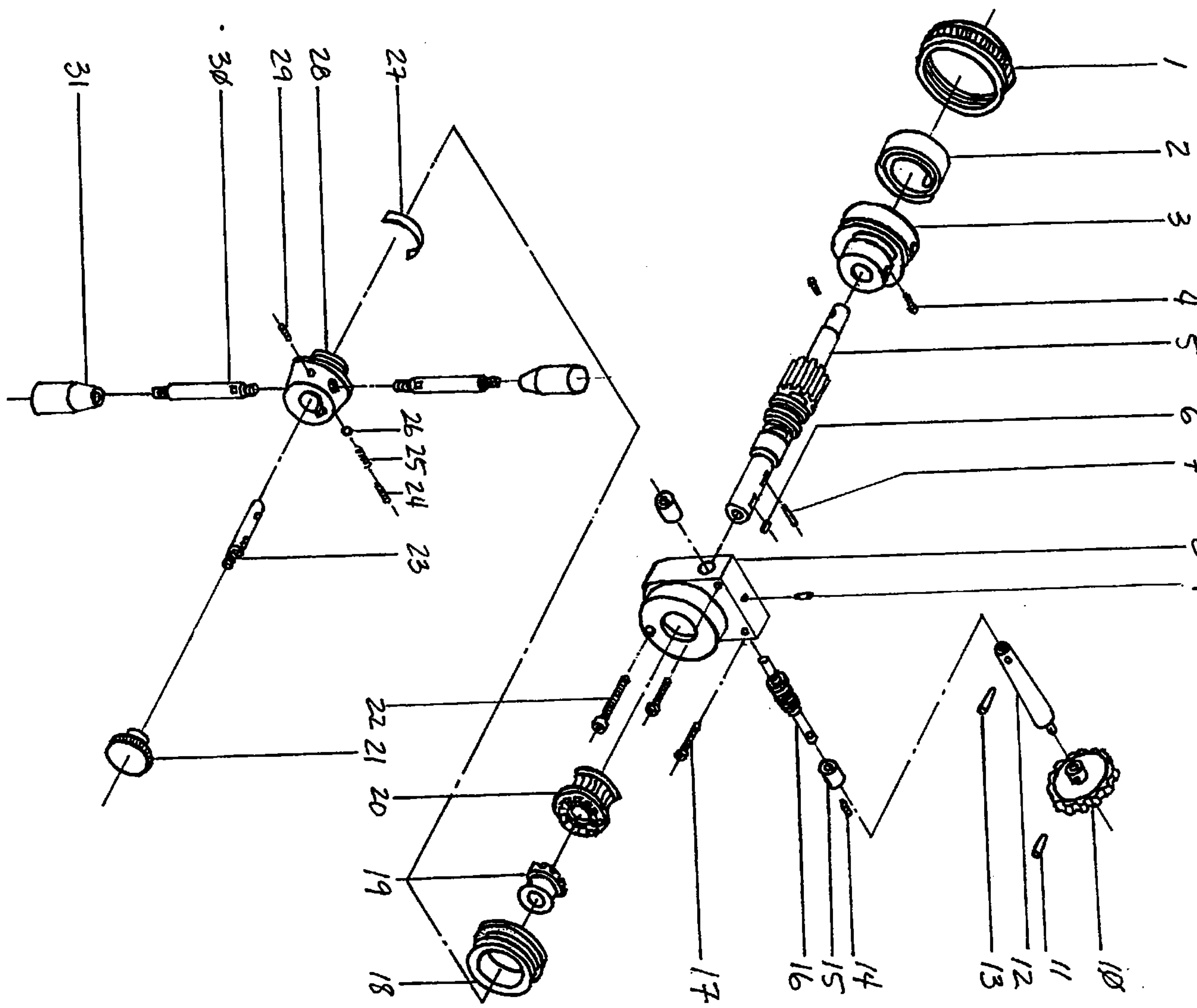
24.11 Compound angle toolpost

Diagram 12

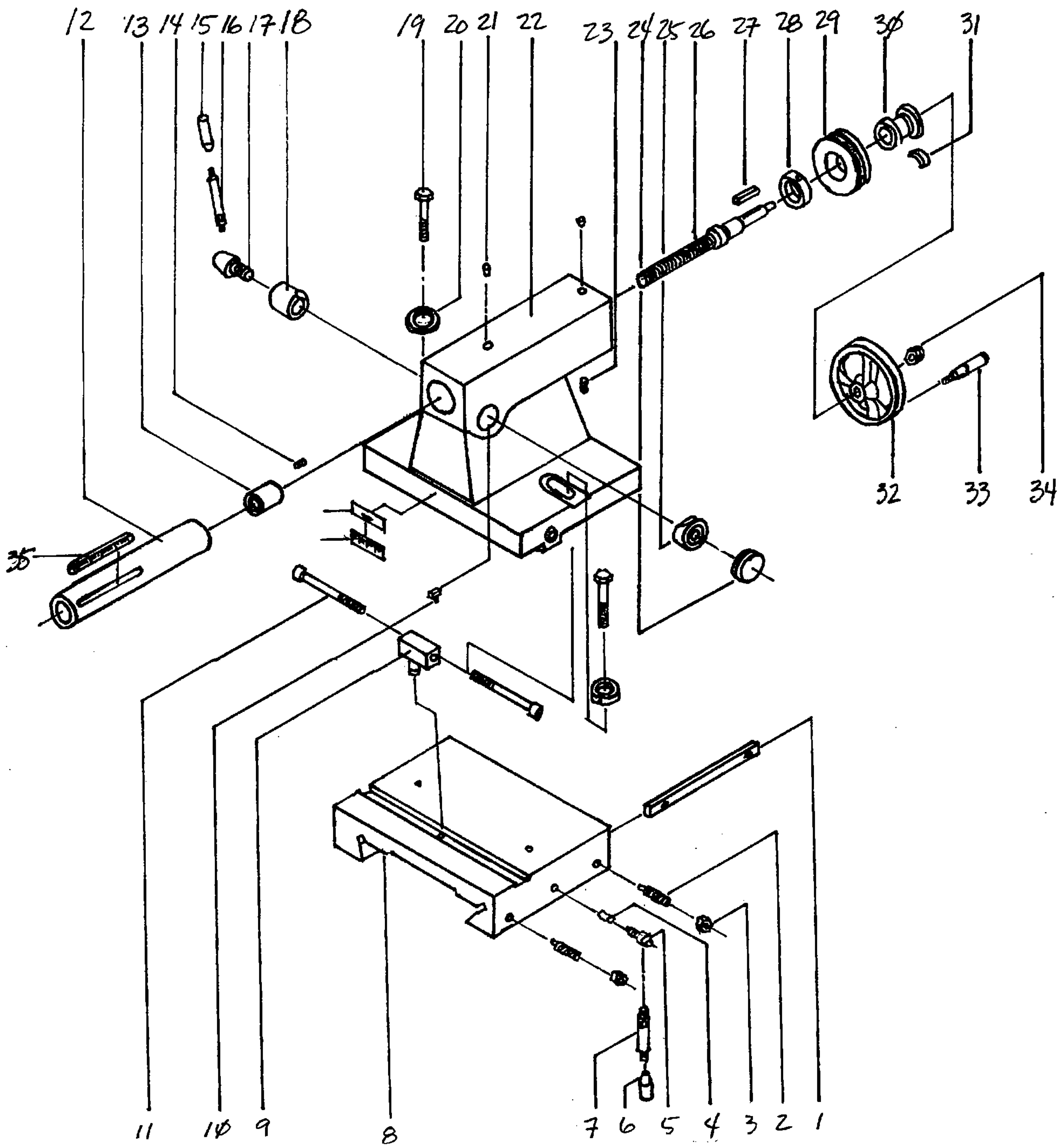
1	C30211	M30x1.5 GB810-86	Spanner Nut
2	B30227		Bevel Gear
3	B30228		Adjustment Washer
4	S20490	8107	Bearing
5	B30229		Nut Base
6	S12321	M8x30 GB70-85	Capscrew
7	B30230		Height-Adjustment Nut
8	S21770	8x15 GB1096-79	Plain Parallel Key
9	B30231		Mill Column
10	S13231	M16x35 GB70-85	Capscrew
11	B30232		Mill/Drillhead Box
12	B30233		Locking Nut
13	B30234		Plug
14	B30235		Head Elevation Leadscrew
15	S20510	8108	Bearing
16	B30236	M36x1.5 J14-5CJB/	Bearing Retainer
17	S11633	M5x8 GB68-85	Slotted Countersunk Screw
18	B30237		Column Head
19	S12311	M8x25 GB70-85	Capscrew
20	S23210	18 GB894.1-86	External Snap Ring
21	B30238		Bushing
22	B30239		Head Carriage
23	C30054	M8x40 GB4141.15-84	Knob
24	C30053	M8x40 GB4141.15-84	Handle Shaft
25	B30240		Handle Seat
26	B30241		Screw Arbor
27	B30242		Locking Sleeve
28	S22156	4x28 GB117-86	Taper Pin
29	B30243		Sleeve
30	B30244		Bevel Gearshaft
31	S11990	M6x20 GB5783-86	Hexagon-Head Bolt
32	S11938	M6x8 GB73-85	Flat-Point Slotted Setscrew
33	S23215	20 GB8941-86	External Snap Ring
34	B30245		Bushing
35	B30246		Middle Shaft
36	B30247		Bearing Cap
37	S11602	M5x5 GB71-85	Cone-Point Setscrew
38	C30014		Felt Collar
39	S20550	D2007108 GB297-84	Tapered Roller Bearing
40	B30248		Spindle Sleeve
41	S20500	D2007108 GB297-84	Single-Row Tapered Roller Bearing
42	C30217	35 GB858-88	Lock Washer
43	C30218	M35x1.5 GB812-88	Spanner Nut
44	B30249		Bearing Seat
45	S11670	M5x16 GB5783-86	Hexagon-Head Bolt
46	S20610	109 GB276-82	Roller Bearing
47	S23360	45 GB894.1-85	External Snap Ring
48	B30250		Mill/Drill Spindle
49	B30251		Pulley Guard
50	S20280	204 GB276-82	Roller Bearing
51	B30252		Inner Ring Spacer
52	B30253		Outer Ring Spacer
53	S20280	204 GB276-82	Roller Bearing
54	B30254		Pole Support
55	S18160	M10 GB923-88	Acorn Nut
56	B30255		Middle Pulley
57	B30256		Mill Pulley
58	B30251		Pulley Guard
59	~~B30258~~ 30257		~~Spindle Cap - Base~~
60	B30259		Spindle Cap - Upper
61	~~B30260~~ 30259		Spindle Cap - Lid
62	S12252	M8x12 GB871-85	Cone-Point Slotted Setscrew
63	S18170	10 GB97.1-85	Flat Washer
64	B30261		Handle Seat
65	C30032	M6x40 GB4141.15-84	Handle Lever
66	S12288	M8x18 GB75-85	Dog-Point Setscrew
67	C30033	M6x35 GB4141.14-84	Knob
68	S12278	M8x16 GB75-85	Dog-Point Slotted Setscrew
69	B30262		Switch Mounting Plate
70	S12282	M8x18 GB78-85	Cone-Point Setscrew
71	S11395	M4x20 GB67-85	Slotted Pan Head Screw
72	S11644	M5x10 GB65-85	Flat Fillister Screw



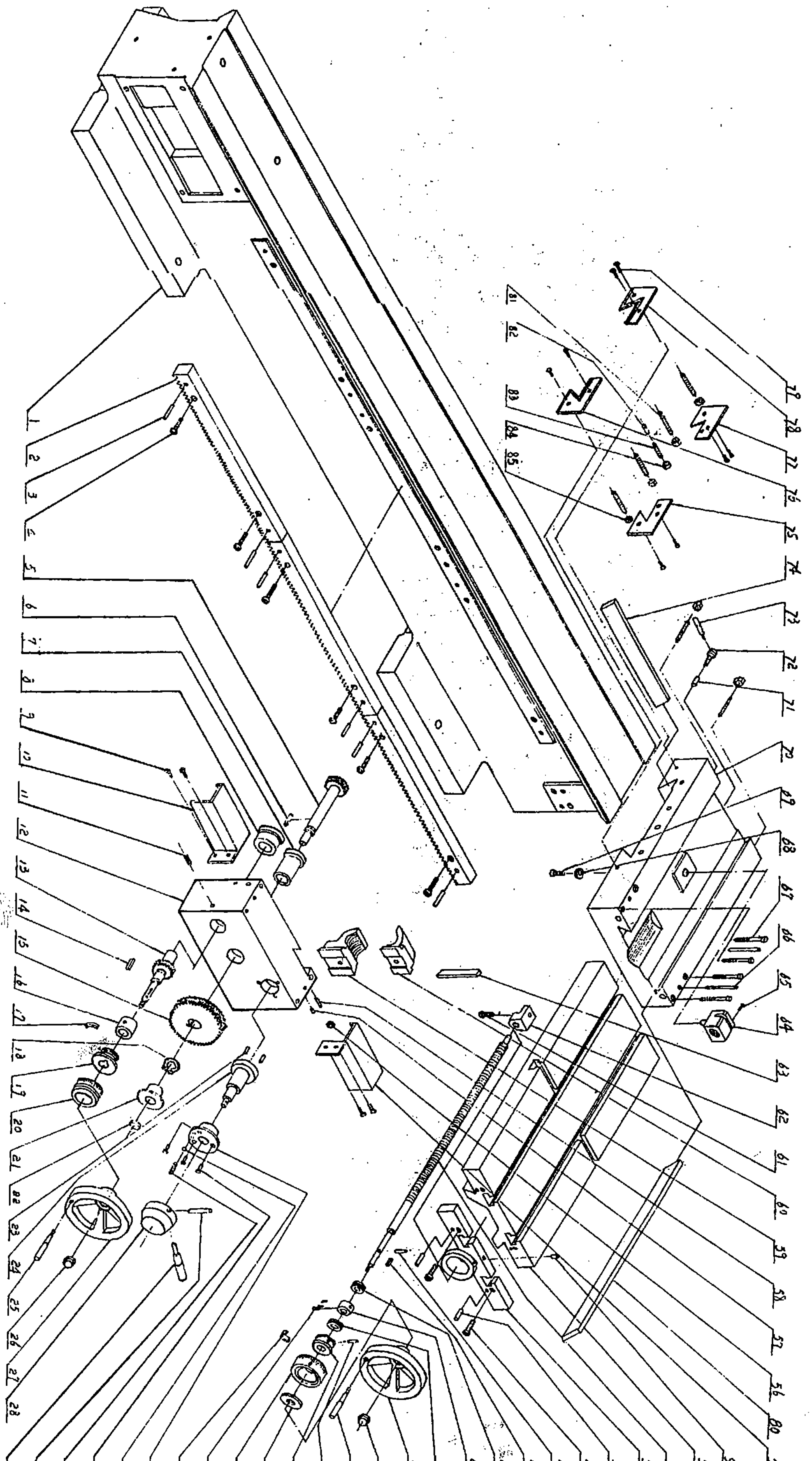
1	C30023		Spring Box Cover
2	B30263	0.8x20x1000	Spring
3	C30025		Spring Seat
4	S11315	M4x6 GB67	Slotted Pan Head Screw
5	B30264		Gearshaft
6	S21560	6x15 GB1096	Plain Parallel Key
7	S22180	5x28 GB119	Tapered Pin
8	C30036		Worm Seat
9	C30035	8 GB1155	Oil Cup
10	C30039		Handwheel Handle
11	S22130	3x18 GB117	Taper Pin
12	B30265		Extension
13	S22110	3x16 GB117	Taper Pin
14	S11632	M5x8 GB71	Cone-Point Slotted Setscrew
15	C30034		Sleeve
16	C30037		Wormscrew
17	S12021	M6x30 GB70	Capscrew
18	C30021		Calibrated Dial
19	C30028		Clutch Sleeve
20	C30027		Wormgear
21	C30030		Cone Knob
22	S11761	M5x50 GB70	Capscrew
23	C30029		Shaft
24	S11958	M6x12 GB73	Flat-Point Slotted Setscrew
25	C30200	0.6x5x15	Spring
26	C30199		Steel Ball
27	C30201	5 GB308	Dial Tension Spring
28	C30022		Handle Seat
29	S11952	M6x12 GB71	Cone-Point Slotted Setscrew
30	C30020	M10x100 GB4141.15	Handle Shaft
31	C30019	M10x50 GB4141.14	Knob


1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
29
30
31

1	B30266		Gib
2	S12328	M8x30 GB79-85	Dog-Point Setscrew
3	S18125	M8 GB6172-86	Hexagon Nut
4	S22420		Locking Pin
5	~~C30161~~ C30031		Locking Handle Seat
6	C30033	M6x36 GB4141.14-84	Knob
7	C30032	M6x40 GB4141.15-84	Handle Shaft
8	B30267		Tailstock Carriage
9	B30268		Nut
10	S21040		T-Key
11	S12441	M8x90 GB70-85	Capscrew
12	B30269		Tailstock Barrel - MT3
13	C30166		Tailstock Nut
14	S11642	M5x10 GB71-85	Cone-Point Setscrew
15	C30054	M8x40 GB4141.14-84	Knob
16	C30053	M8x50 GB4141.15-84	Handle Shaft
17	C30165		Taper Knob
18	B30270		Locking Sleeve
19	S12650	M10x45 GB5782-86	Capscrew
20	S18170	10 GB97.2-85	Flat Washer
21	C30050	6 GB1155-79	Lubricating Nipples
22	B30271		Tailstock Casting
23	S11952	M6x12 GB71-85	Cone-Point Setscrew
24	C30170		Cover
25	B30272		Locking Nut
26	B30273		Tailstock Screw
27	S21110	4x18 GB1096-79	Key
28	B30274		Bushing
29	B30275		Calibrated Dial
30	B30276		Dial Sleeve
31	C30173		Spring
32	C30059	M12x100 GB4141.22	Handwheel
33	C30060	M6x50 GB4141.15-84	Handwheel Handle
34	S18160	M10 GB923-88	Acorn Nut
35	B30279		Calibrated Label
36	B30280		Zero Line Calibration
37	B30281		Tailstock Label


12 13 14 15 16 17 18
19 20 21 22
23 24 25 26 27 28 29 30 31
32 33 34
35
11 10 9 8 7 6 5 4 3 2 1

1	B30282		Bed
2	B30283		Rack
3	S22190	6x30 GB117-86	Taper Pin
4	S12291	M8x20 GB70-85	Capscrew
5	B30284		Gearwheel Shaft
6	S21500	6x12 GB1096-79	Plain Parallel Key
7	B30285		Bearing Sleeve
8	B30286		Bearing Sleeve
9	S11635		Slotted Pan Head Screw
10	B30287		Leadscrew Cover - Left
11	S11672	M5x16 GB71-85	Cone-Point Slotted Setscrew
12	B30288		Apron
13	B30289		Gearwheel Shaft
14	S21180	4x32 GB1096-79	Plain Parallel Key
15	B30290		Gear
16	B30291		Bearing Sleeve
17	C30173		Flat Spring
18	S23200	20 GB894.1-86	External Snap Ring
19	B30292		Sleeve
20	B30293		Dial
21	B30294		Bearing Sleeve
22	B30295		Plug
23	B30296		Half-Nut Shaft Pin
24	B30297		Half-Nut Shaft
25	C30278	M8x65 GB4141.5-84	Handwheel Knob
26	S18160	M10 GB923-88	Acorn Nut
27	C30277	M12x125 GB4141.22	Handwheel
28	C30124	12x50 GB4141.19-84	Handle Seat
29A	C30053	M8x63 GB4141.5-84	Half-Nut Handle
29B	C30054		Knob
30	S22185	5x45 GB117-86	Taper Pin
31	C30125	0.8x5x25	Spring
32	S11353	M4x12 GB68-85	Slotted Countersunk Headscrew
33	C30126	6.5 GB308-84	Steel Ball
34	B30298		Locating Sleeve
35	C30173		Flat Spring
36	S11952	M6x12 GB71-85	Cone-Point Slotted Setscrew
37	C30299		Dial
38	C30300		Washer
39	C30301		Dial Sleeve
40	S22154		Taper Pin
41	C30060	M6x65 GB4141.15-84	Handle
42	S18160	M10 GB923-88	Acorn Nut
43	C30059	M12x100 GB4141.22	Handwheel
44	S20090	8201 GB302-64	Roller Bearing
45	C30302		Sleeve
46	S20090	8201 GB302-64	Roller Bearing
47	S21130	4x14 GB1096-79	Plain Parallel Key
48	S22151	4x25 GB117-86	Taper Pin
49	S12271	M8x16 GB70-85	Capscrew
50	S22151	4x25 GB117-86	Taper Pin
51	B30303		Cross-feed Screw Seat
52	B30304		Cross-feed Screw
53	B30305		Bench
54	C30035	GB115-79	Oil Cup
55	B30306		Leadscrew Cover - Right
56	S18065	M5 GB52-85	Nut
57	C30050	6 GB1155-79	Oil Cup
58	S11718	M5x25 GB75-85	Dog-Point Slotted Setscrew
59	B30307		Half-Nut
60	B30307		Half-Nut
61	S11971	M6x16 GB70-85	Capscrew
62	B30309		Trestle
63	B30310		Gib
64	B30311		Cross-feed Nut
65	S11371	M4x16 GB70-85	Capscrew
66	S22190	6x30 GB117-86	Taper Pin
67	S12041	M6x40 GB70-85	Capscrew
68	S18110	6 GB96-85	Flat Washer
69	S11971	M6x16 GB70-85	Capscrew
70	B30312		Carriage
71	S22420		Locking Pin
72	~~C30161~~ B30261		Locking Handle Seat
73A	C30032		Handle Lever
73B	C30033		Knob
74	B30313		Gib
75	B30314		Way Wipe - Right Back
76	B30315		Way Wipe - Left Back
77	B30316		Way Wipe - Right Front
78	B30317		Way Wipe - Left Front
79	S11635	M5x8 GB818-85	Slotted Pan Headscrew
80	B30318		Long Gib
81	S12348	M8x40 GB79-85	Dog-Point Setscrew
82	S22420		Locking Pin
83	S12648	M10x24 GB79-85	Dog-Point Setscrew
84	S18155	M10 GB6172-86	Hexagon Nut
85	S18125	M8 GB6172-86	Hexagon Nut



Chester Drake

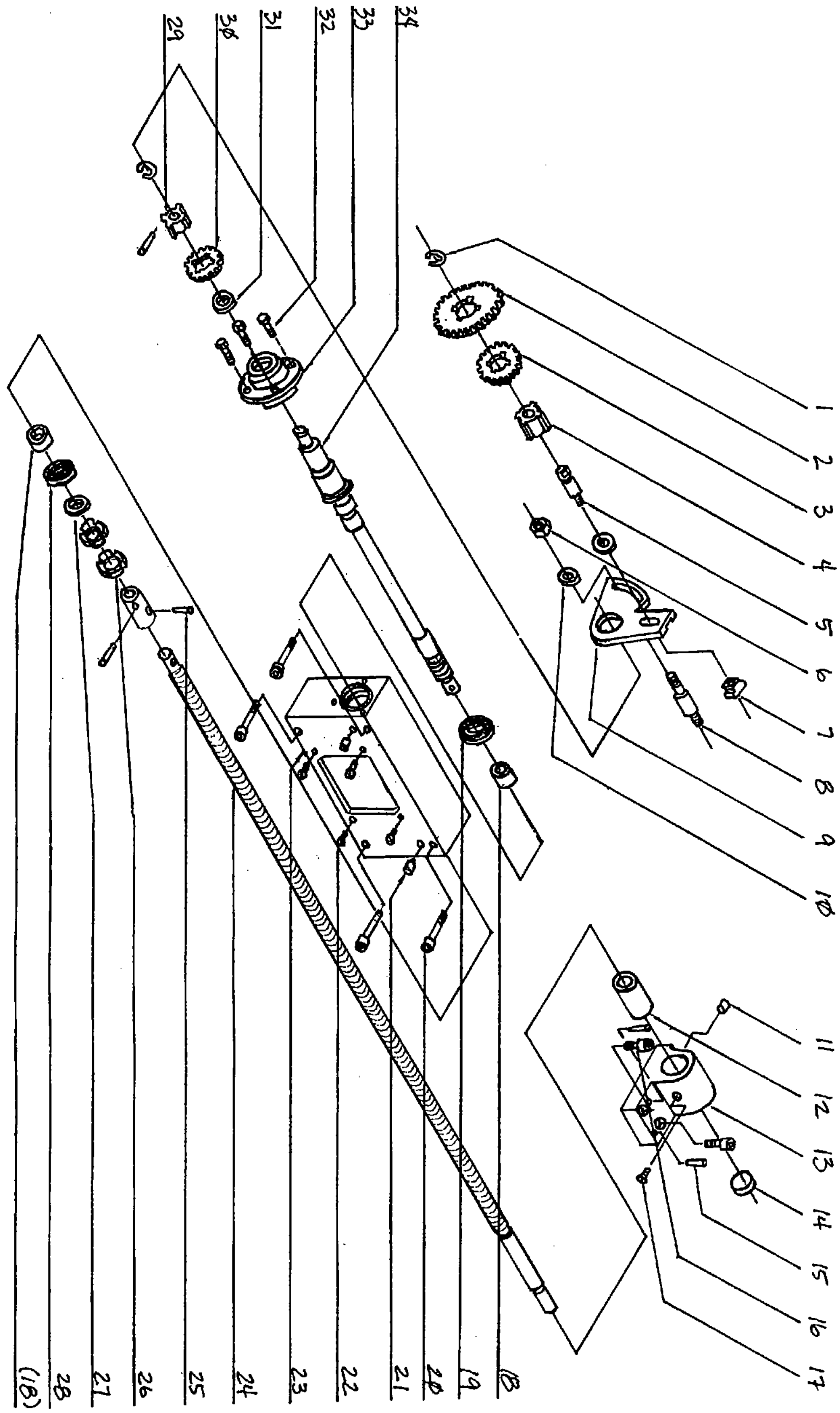
P .1835

H .1625

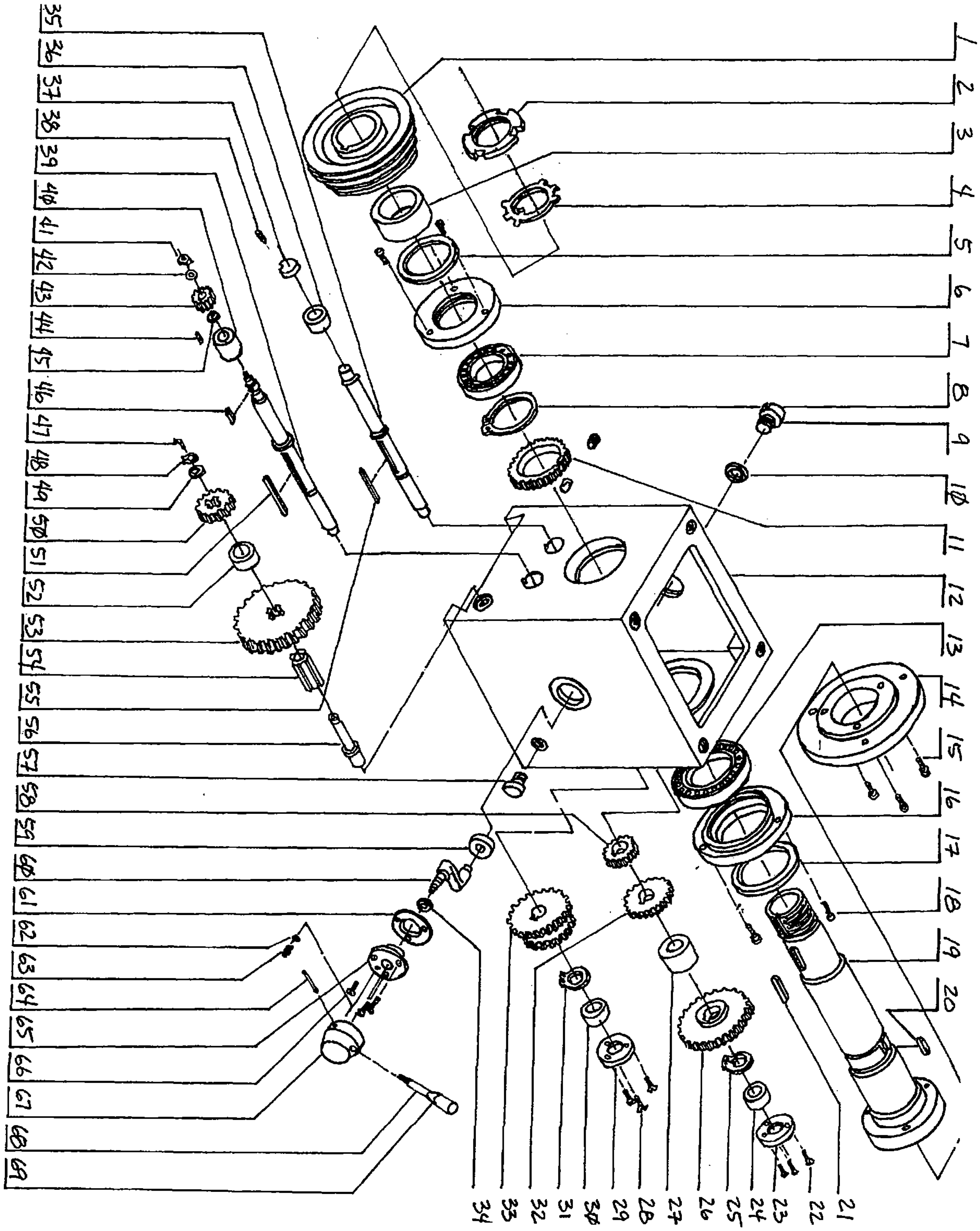
1	S23080	9 GB896-86	E-Ring
2			Change Gear
3			Change Gear
4	B30319		Spline Housing
5	B30320		Shaft
6	S18155	M10 GB6172-86	Hexagon Nut
7	B30321		T-Nut
8	B30322		Shaft
9	B30323		Change Gear Plate
10	S18170	10 GB96-85	Washer
11	S30050	6 GB1155-89	Oil Cup
12	B30324		Bush Bearing
13	B30325		Leadscrew Seat
14	B30326		Plug
15	S22192	6x32 GB117-86	Taper Pin
16	S12321	M8x30 GB70-85	Capscrew
17	S11642	M5x10 GB71-85	Cone-Point Slotted Setscrew
18	B30327		Shaft Sleeve
19	S20230	8105 GB301-64	Unidirectional Ball Bearing
20	S12081	M6x60 GB70-85	Capscrew
21	S30050	6 GB115-89	Oil Cup
22	S11335	M4x8 GB67-76	Slotted Round Headscrew
23	B30328		Left Trestle
24	B30329		Longitudinal Feedscrew
25	S22192	6x32 GB117-86	Taper Pin
26	C30209	M20x1.5 GB812-88	Spanner Nut
27	B30330		Sleeve
28	S20230	8104 GB301-64	Unidirectional Ball Bearing
29	B30457		Spline Housing
30			Gear
31	S18200	12 GB 96-85	Washer
32	S11394	M4x20 GB5780-86	Hexagon Head Bolt
33	B30331		Pedestal
34	B30332		Change Gearshaft
35	S22130	3x18	Taper Pin

CHANGE GEARS

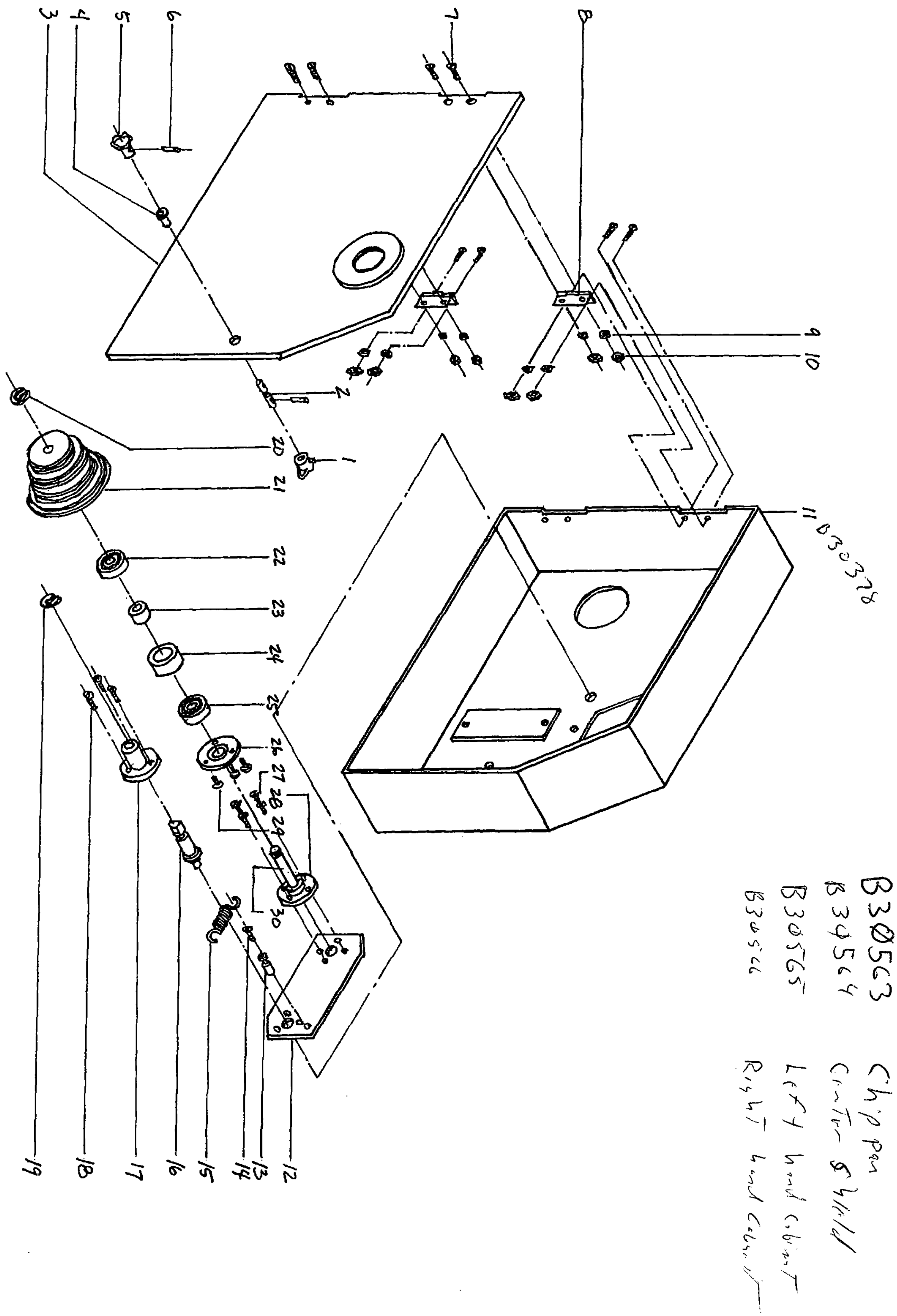
20T	B30333	Gear
22T	B30334	Gear
24T	B30335	Gear
25T	B30336	Gear
26T	B30337	Gear
27T	B30338	Gear
28T	B30339	Gear
30T	B30340	Gear
34T	B30341	Gear
35T	B30342	Gear
38T	B30343	Gear
40T	B30344	Gear
50T	B30345	Gear

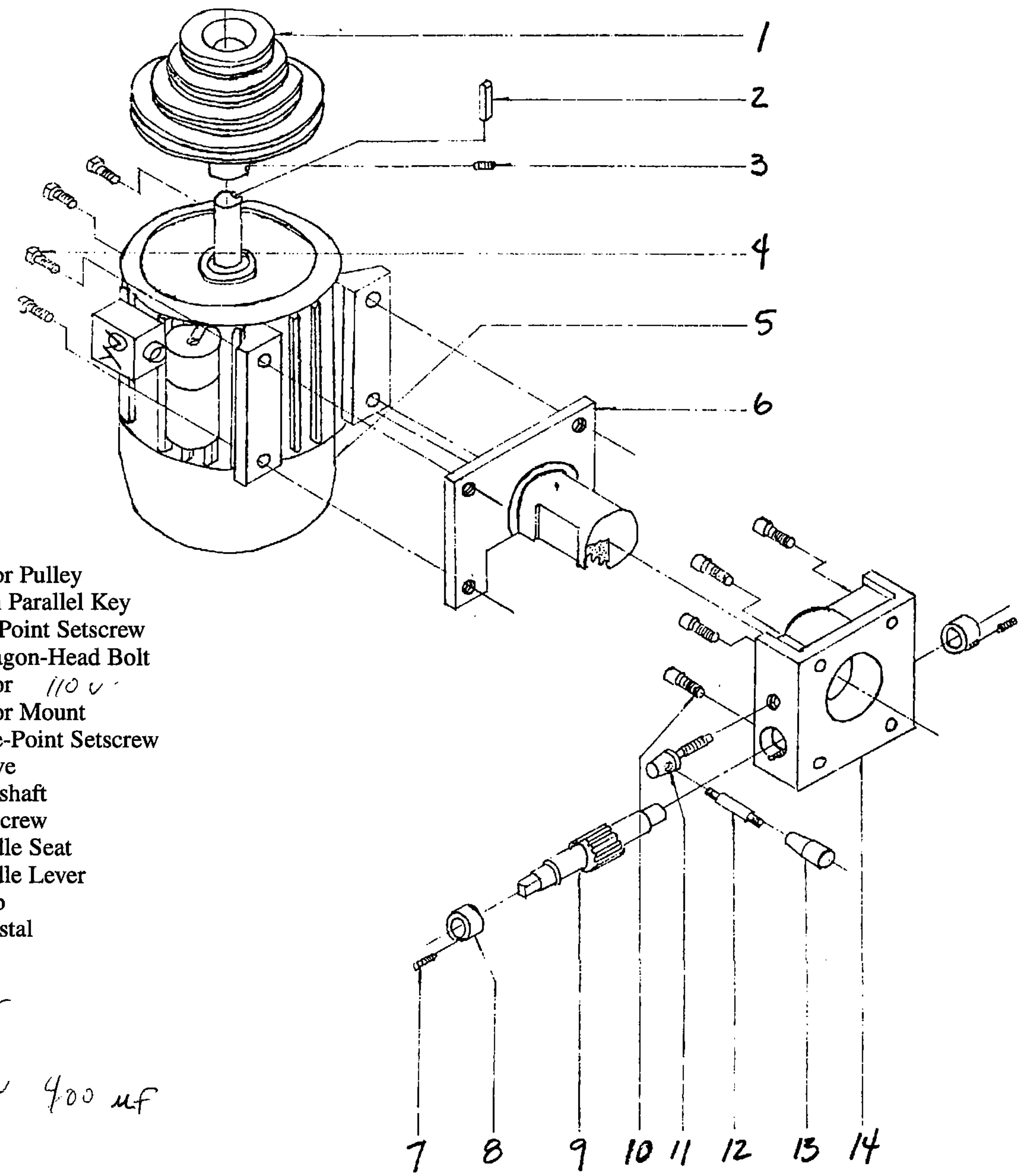


1	B30346		Spindle Pulley	38	S11302	M4x5 GB71	Cone-Point Slotted Setscrew
2	B30347	M50x1.5 GB812	Spanner Nut	39	B30362		Shaft
3	B30348		Shaft Sleeve	40	B30354		Oil-Impregnated Bearing
4	B30349	50 GB858	Lock Washer for Spanner Nut	41	S18155	M10 GB6170	Hexagon Nut
5	C30014	50 JB/GQ0324	Felt Collar	42	S18165	10 GB93	Spring Washer
6	B30350		Bearing Cap	43	B30363		Pinion – GEAR
7	S20640	2007110 GB297	Tapered Roller Bearing	44	S11302	M4x5 GB71	Cone-Point Slotted Setscrew
8	S23390	55 GB894.1	External Snap Ring	45	S24300	16x2.4 GB1235	O-Ring
9	C30103	M20x1.5 Q/ZB220	Oil-Port Plug	46	S21115	4x10 GB1096	Plain Parallel Key
10	C24400	30x20x2	Rubber Ring	47	C30050	6 GB1155	Oil Cup
11	B30351		Gear	48	S23080	9 GB896	E-Ring
12	B30352		Lathe Head	49	S18200	12 GB97.1	Flat Washer
13	S20660	2007112 GB297	Tapered Roller Bearing	50	?		Change Gear ?(Which one)
14	B30376		Ring Flange	51	S21745	6x70 GB 1096	Plain Parallel Key
15	S12621	M10x30 GB70	Capscrew	52	B30364		Sleeve
16	B30375		Main Shaft Bearing Oil Seal	53	B30365		Gear
17	C30014	68 JB/GQ 0324	Felt Collar	54	B30366		Spline
18	S11691	M5x20 GB70	Capscrew	55	S21742	6x65 GB1096	Plain Parallel Key
19	B30353		Lathe Spindle	56	B30367		Shaft
20	S21940	12x12 GB1567	Flat Key	57	B30368	M16x1.5 GB1160.2	Sight Glass
21	S21970	12x28 GB1567	Flat Key	58	B30369		Gear
22	S11373	M4x16 68	Slotted Countersunk Capscrew	59	B30370		Locator Bushing
23	B30377		Plug	60	B30371		Rocker Arm
24	B30354		Oil-Impregnated Bearing	61	B30372		Oil Seal
25	S23200	20 GB894.1	External Snap Ring	62	C30126	6.5 GB308	Steel Ball
26	B30355		Gear	63	C30125	0.8x5x25 GB2089	Spring
27	B30356		Shaft Sleeve	64	S22185	5x50 GB117	Taper Pin
28	S11373	M4x16 GB68	Slotted Countersunk Capscrew	65	~~C30127~~ C30561		Locating Sleeve
29	B30377		Plug	66	S11653	M5x12 GB68	Slotted Countersunk Capscrew
30	B30354		Oil-Impregnated Bearing	67	C30124		Position-Fixing Handle Seat
31	S23200	20 GB894.1	External Snap Ring	68	C30053	BM8x50 GB4141.15	Handle
32	B30357		Gear	69	C30054	BM8x40 GB4141.14	Handle Sleeve
33	B30358		Double Gear				
34	B30359		Rubber Ring				
35	B30360		Shaft				
36	B30354		Oil-Impregnated Bearing				
37	B30361		Cap				


1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
29
30
31
32
33
34
35
36
37
38
39
40
41
42
43
44
45
46
47
48
49
50
51
52
53
54
55
56
57
58
59
60
61
62
63
64
65
66
67
68
69

1	C30093		Locking Cam
2	C30092		Shaft
3	B30373		Gearbox
4	C30374		Sleeve
5	C30091	8x32 GB4141.29	Knob
6	S22130	3x18 GB117	Taper Pin
7	S11633	M5x8 GB68	Slotted Countersunk Capscrew
8	C30141		Hinge
9	S18075	5 GB97.1	Lock Washer
10	S18065	M5 GB6170	Hexagon Nut
11	B30378		Gearbox
12	B30379		Mounting Plate
13	B30380		Sleeve
14	S11653	M5x12 GB68	Slotted Countersunk Cap Screw
15	C30381	2x12x61.5 GB4142	Tension Spring
16	B30382		Shaft
17	B30383		Shaft Seat
18	S11653	M5x12 GB68	Slotted Countersunk Capscrew
19	S23140	12 GB896	E-Ring
20	S23205	15 GB896	E-Ring
21	B30384		Idler Pulley
22	S20260	203 GB276	Ball Bearing
23	C30198		Inner Sleeve
24	C30197		Outer Sleeve
25	S20260	203 GB276	Ball Bearing
26	B30385		Bearing Apron
27	S11633	M5x8 GB68	Slotted Countersunk Capscrew
28	B30386		Axle Location Plate
29	S11013	M3x6 GB67	Slotted Pan Screw
30	B30387		Pulley Shaft


1
2
3
4
5
6
7
8
9
10
11 B30378
12
13
14
15
16
17
18
19
20
21
22
23
24
25
26
27
28
29
30
B30563 Chip pan
B34564 Center Shield
B30565 Left hand cabinet
B30566 Right hand cabinet



1	B30388		Motor Pulley
2	S21560	B6x15 GB1096-79	Plain Parallel Key
3	S11648	M5x10 GB73-85	Flat-Point Setscrew
4	S12280	M8x18 GB70-85	Hexagon-Head Bolt
5	B30389	0.55kW/110v 60Hz	Motor 110v
6	B30390		Motor Mount
7	S11632	M5x8 GB71-85	Cone-Point Setscrew
8	B30391		Sleeve
9	B30392		Gearshaft
10	S12281	M8x18 GB70-85	Capscrew
11	B30393		Handle Seat
12	C30032	BM6x25 GB4141.15	Handle Lever
13	C30033	BM6x36 GB4141.14	Knob
14	B30394		Pedestal

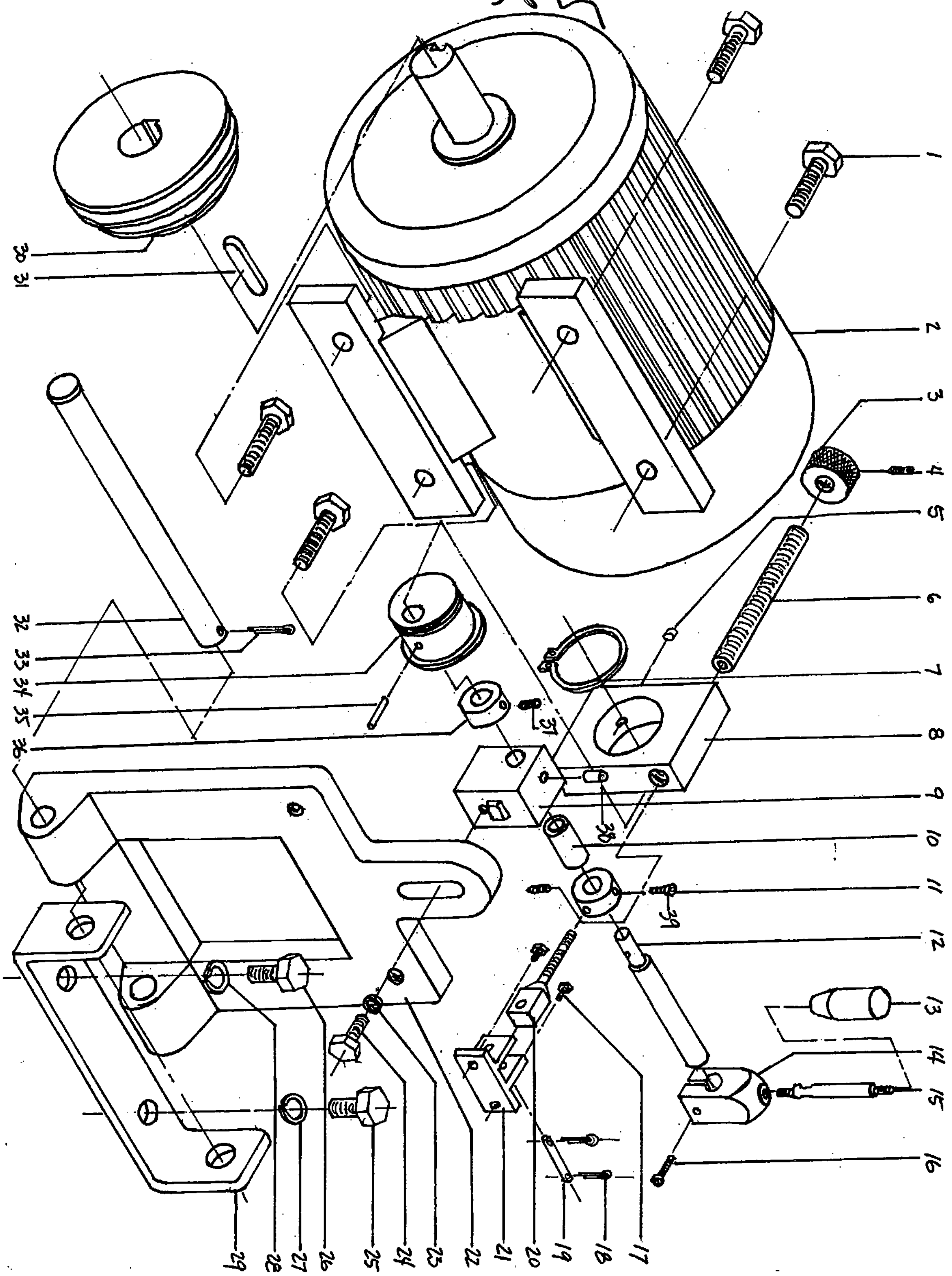
B30554 220v motor

S25101 capacitor 110v 400 uf

B30555 220 V motor

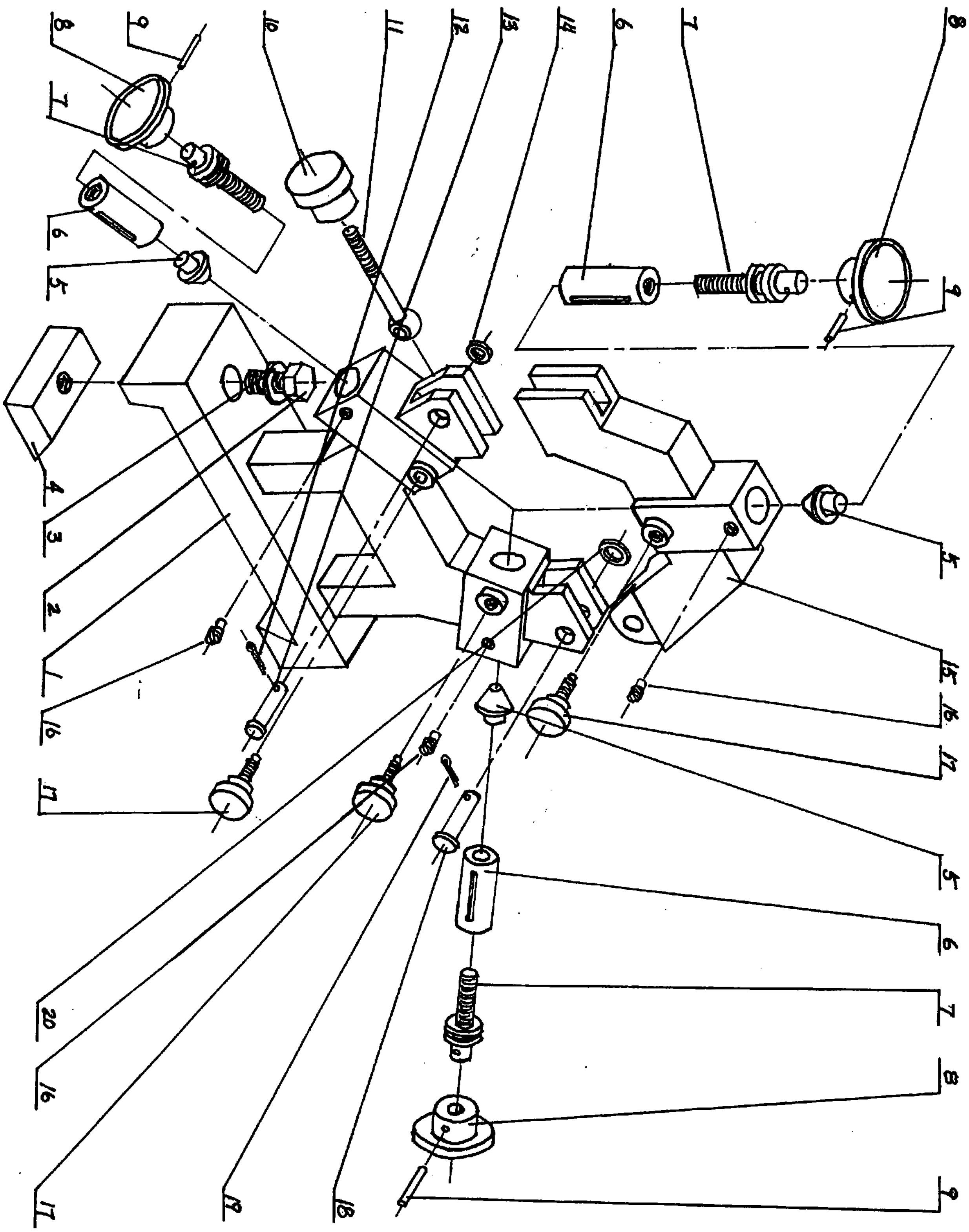
1	S12310	M8x25 GB5780-86	Hexagon-Head Bolt
2	B30395	1.1kW /110V 60Hz	Motor 110 V
3	B30397	M12	Screw Arbor with Knurled Knob
4	< Updated >		
5	C30050	6 GB1155-79	Oil Cup
6	B30397	M12	Screw Arbor with Knurled Knob
7	S23320	40 GB894.1-86	Circlips for Shaft - Type A
8	C30398		Set
9	C30399		Locating Piece
10	C30400		Sleeve
11	C30401		Locating Sleeve
12	C30402		Shaft
13	C30033	M6x36 GB4141.14	Knob
14	C30403		Handle Seat
15	C30032	M6x40 GB4141.15	Handle Lever
16	S11991	M6x20 GB70-85	Capscrew
17	S11670	M5x15 GB5-76	Hexagon-Head Bolt
18	S22520	2x12 GB91-86	Split Pin
19	C30404	6x28 GB119-86	Cotter Pin
20	C30405		Attachment Screw Arbor
21	C30406		Motor Tension Plate
22	B30407		Motor Mount
23	S18135	8 GB93-87	Lock Washer
24	S12310	M8x25 GB5781-86	Hexagon-Head Bolt
25	S12920	M12x30 GB30-76	Hexagon-Head Bolt
26	S12920	M12x30 GB30-76	Hexagon-Head Bolt
27	S18195	12 GB93-87	Lock Washer
28	S18195	12 GB93-87	Lock Washer
29	B30408		Pedestal
30	B30409		Motor Pulley
31	S21820	8x40 GB1567-79	Plain Parallel Key
32	B30410		Straight Pin
33	S22535	3x20 GB91-86	Cotter Pin
34	C30411		Eccentric Wheel
35	S22152	4x24 GB117-86	Taper Pin
36	C30401		Locating Sleeve
37	S11608	M5x5 GB79-85	Dog-Point Setscrew
38	C30050	6 GB1155-79	Oil Cup
39	S11611	M5x6 GB70-85	Capscrew

B30395
M070K

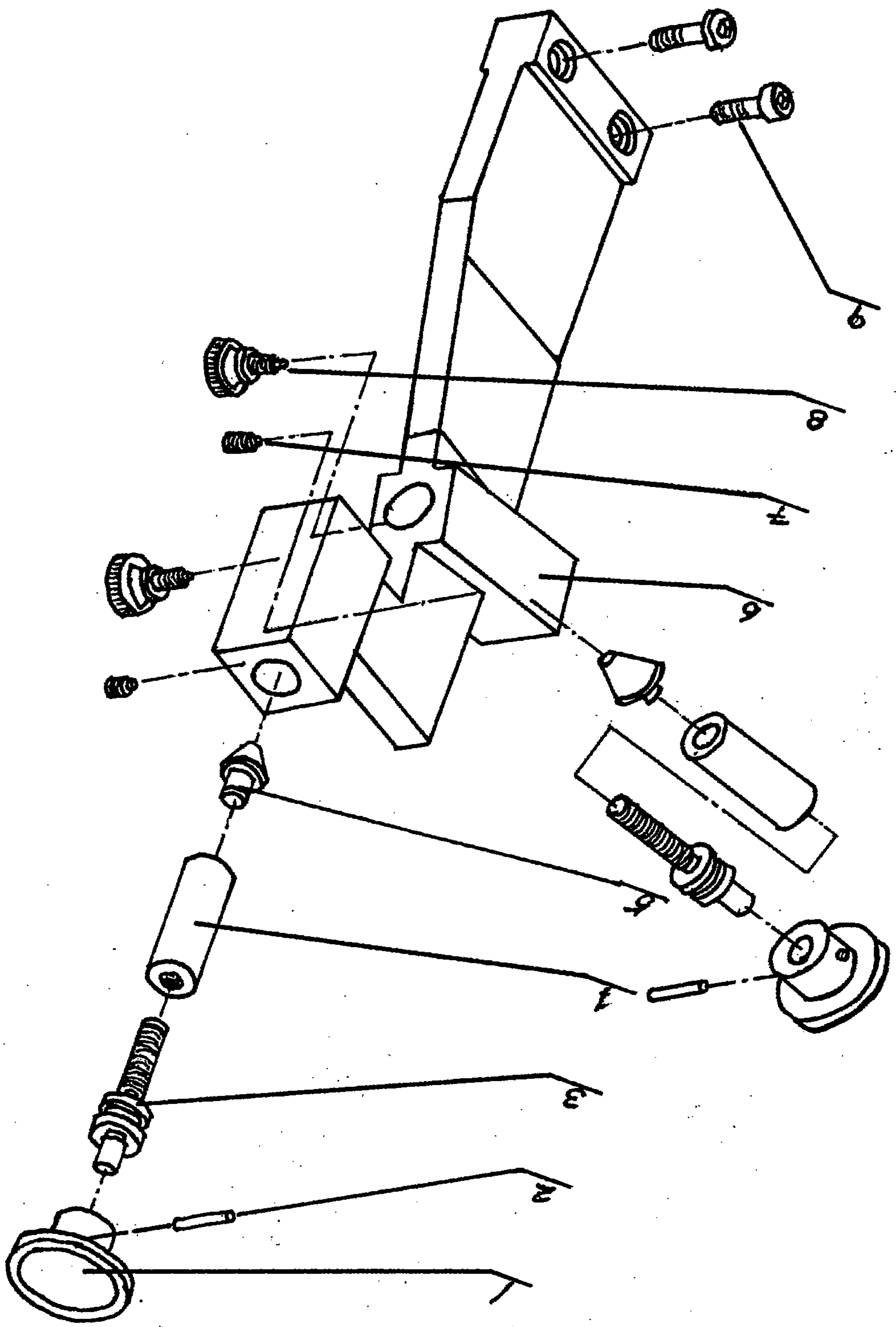


1	B30412		Steady Rest Frame Base
2	S12651	M10x45 GB30	Hexagon Head Bolt
3	S18170	10 GB97	Washer
4	~~C30187~~ B30562		Gib
5	C30188		Pressure Head
6	C30189		Core Sleeve
7	C30190		Adjusting Bolt
8	C30191	10x40 GB4141.27	Knob
9	S22140	3x22 GB117	Taper Pin
10	C30192		Knob
11	C30193	M8x65 GB798	Dog Bolt
12	S22510	2x10 GB91	Cotter Pin
13	S22620	6x30 GB882	Pivot Pin
14	S18110	6 GB97	Washer
15	C30194		Steady Rest Frame Top
16	S11948	M6x10 GB75	Full Dog-Point Setscrew
17	C30195		Knurled Screw
18	S22620	8x32 GB882	Pivot Pin
19	S22510	3.2x12 GB91	Cotter Pin
20	S18140	8 GB97	Washer

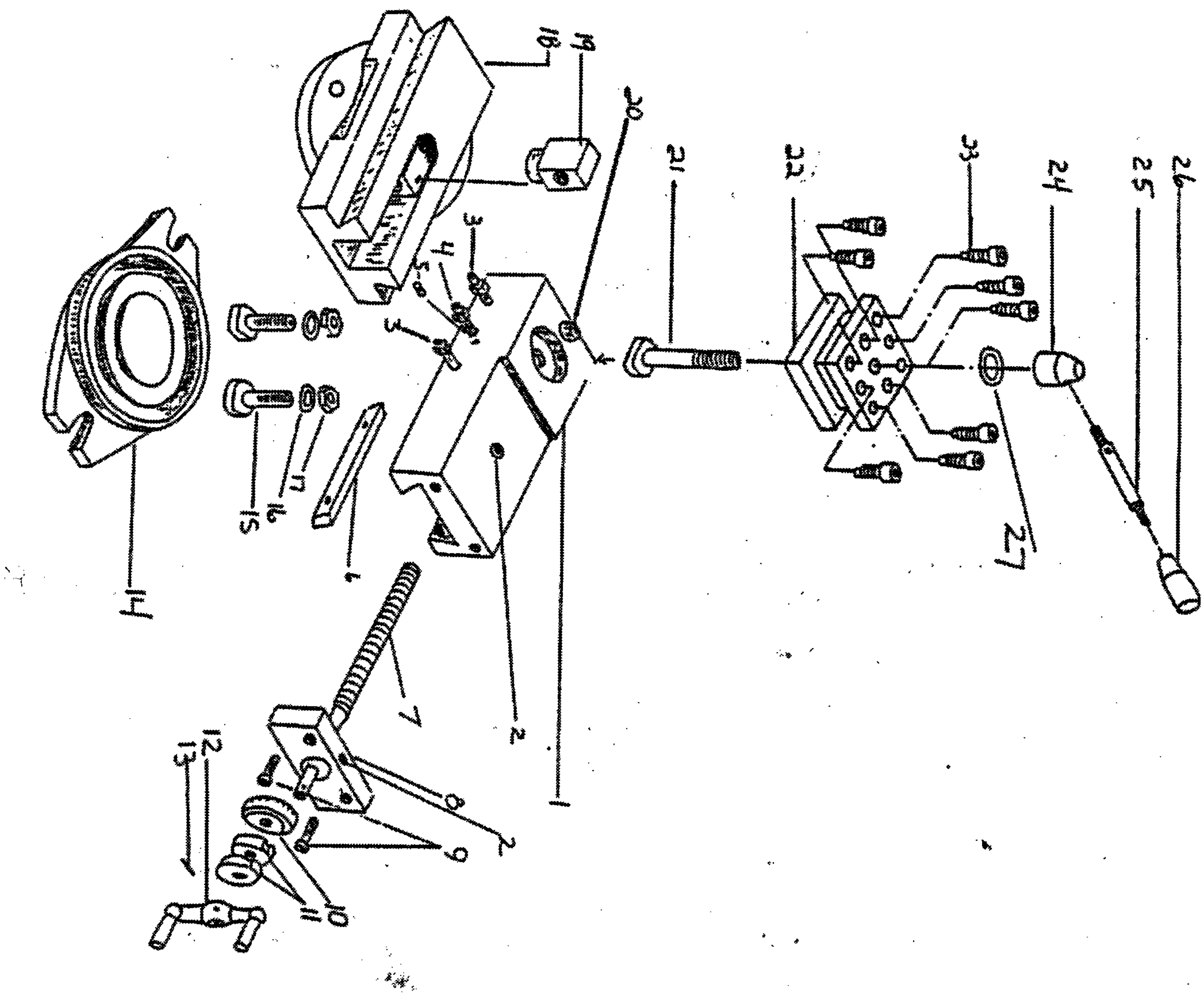
S25100 Capacitor Run 250v ~~5mF~~ 50mF
S25103 Capacitor Start 125V 600uF



1	C30191	10x40 GB4141.27	Knob
2	S22140	3x22 GB117	Taper Pin
3	C30190		Adjusting Bolt
4	C30189		Core Sleeve
5	C30188		Pressure Head
6	C30196		Mount
7	S11948	M6x10 GB75	Full Dog-Point Setscrew
8	C30195		Knurled Screw
9	S12291	M8x25 GB70	Capscrew



1	C1330175		Sliding Carriage
2	CB30050		Oil Cup
3	SP11672	M5x16	Cone-Point Hex Screw with Hex Nut
4	SP11640	M5x10	Setscrew with Hex Nut
5	SP22410		Locking Pin
6	CB30176		Gib
7	CB30177		Screw with Bearing
8	CB30178		Hanger
9	SP11971	M6x16	Hex Socket Setscrew
10	CB31015		Dial
11	CB30179		Spanner Nut
12	CB30179		Handel
13	SP22110		Taper Pin
14	CB30180		Revolving Bed
15	SP12296	M8x20	Base-Locking Bolt
16	SP18140	M8	Washer
17	SP18125	M8x1.25	Hex Nut
18	CB30181		Carriage Base
19	CB30182		Carriage Nut
20	CB30183		Ball with Spring
21	SP12696	M10x65	Turret Bolt
22	CB30184		Toolpost
23	SP12311	M8x25	Hex Socket Headscrew
24	CB30185		Handle Seat
25	CB30053		Toolpost Handle
26	CB30054		Long Sleeve Knob
27	SP18170	M10	Washer
31	SP12626	M10x30	Bolt
32	SP18170	M10	Washer
33	SP18155	M10	Nut



Notes

CB-1239

Product	Smithy Number	Price	
Air Mask	15-020	$	1.50
Goggles	15-015	$	2.50
Ear Plugs	15-025	$	1.95
Cutting/Tapping Fluid	49-101	$	1.00
Manual Cover	83-942	$	1.50
Owners' Manuals	83-94X	$	5.00
Carbide Bits	43-01(1-5)	$	11.45
Drill Chuck, 1\2"	* C30545	$	17.97
MT5 Dead Center	41-005	$	24.00
MT3, Dead Center	41-003	$	8.95
Key, Lathe Chuck	C30532	$	12.75
Key, Drill Chuck	C30533	$	5.25
Oil Gun	80-100	$	6.00
17/19 mm Open End Wrench	C30535	$	3.00
8/10 mm Open End Wrench	C30539	$	3.00
Jaws (3)	9-10	$	15.00
Allen Wrench, 4mm	C30540	$	1.50
Allen Wrench, 5 mm	C30542	$	1.50
Allen Wrench, 6 mm	C30537	$	1.50
Allen Wrench, 8 mm	C30536	$	1.50
Gear, 20 Teeth	B30333	$	5.55
Gear, 22 Teeth	B30334	$	5.55
Gear, 24 Teeth	B30335	$	5.55
Gear, 26 Teeth	B30337	$	5.55
Gear, 27 Teeth	B30338	$	5.55
Gear, 28 Teeth	B30339	$	5.55
Gear, 30 Teeth	B30340	$	5.55
Gear, 34 Teeth	~~B30340~~ – (B30341)	$	8.25
Gear, 35 Teeth	B30342	$	8.25
Center Shield with Screws	B30564	$	10.00
Arbor, MT2 to MT3	C30546	$	10.20
Compound Angle Toolpost	45-100	$	79.95
3 Jaw Chuck 6" with jaws	40-006	$	120.06
1/4 End Mill	50-402	$	3.89
3/8 End Mill	50-406	$	3.89
1/2 End Mill	50-410	$	5.09
Plug	S12898	$	1.23
Vise	32-110	$	24.95
End Mill Adaptor	65-010	$	22.95
Draw Bar Kit (14") (Inch) (3/8")	K99-163	$	4.95
Draw Bar Kit (14") (Metric) (12 mm)	K99-166	$	4.95