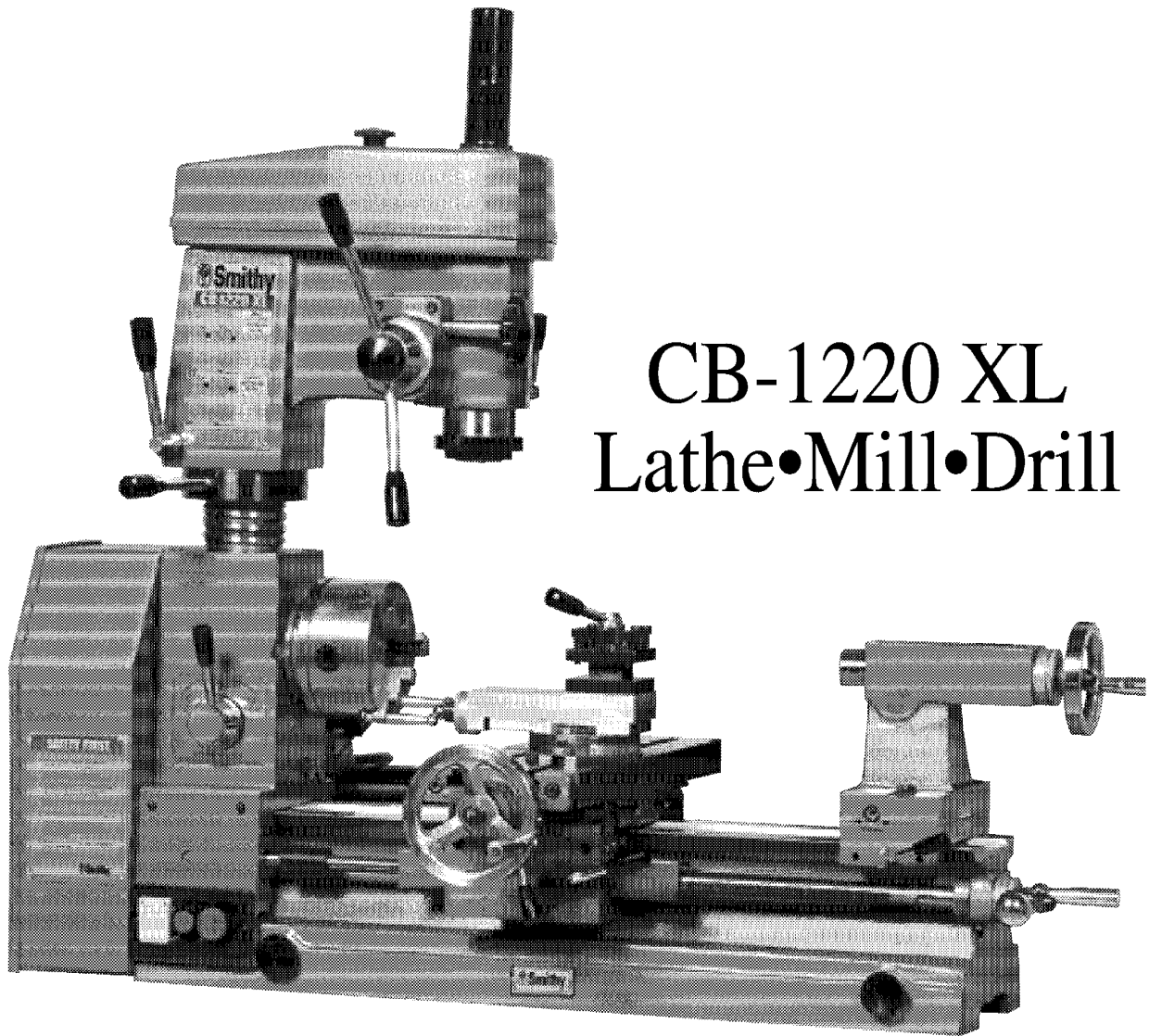


 **Smithy®**

**Operator's Manual**



**CB-1220 XL  
Lathe•Mill•Drill**

170 Aprill Drive • PO Box 1517  
Ann Arbor, Michigan 48106-1517

© 1997 by Smithy Co.

1-734-913-9063

## Contents

Section	Page
One: Introduction . . . . .	5
1.1 Customer information	
1.2 Specifications	
Two: Safety . . . . .	7
Three: Knowing Your Machine . . . . .	8
Four: Caring for Your Machine . . . . .	9
Five: Basic Parts of the CB-1220 XL . . . . .	10
Six: Uncrating and Setting Up Your Machine . . . . .	14
6.1 Moving the machine	
6.2 Uncrating and positioning the machine	
6.3 Selecting a location	
6.4 Cleaning and lubricating the machine	
6.5 Setting up your CB-1220 XL	
6.6 Setting lathe and mill speeds	
6.7 Adjusting belt tension	
6.8 Adjusting the gibs	
6.9 Reducing backlash	
6.10 Running in	
Seven: Turning . . . . .	22
7.1 Turning speeds	
7.2 Gear ratios	
Eight: Metalcutting Theory . . . . .	25
8.1 Tool sharpness	
8.2 Heat	
Nine: Grinding Cutter Bits for Lathe Tools . . . . .	28
9.1 High-speed-steel cutters	
9.2 Materials other than steel	
9.3 Bits for turning and machining brass	
9.4 Special chip craters and chipbreakers	
9.5 Using a center gauge to check V-thread forms	
9.6 Acme or other special threads	
9.7 Carbide-tipped cutters and cutter forms	

Ten: Setting Up Lathe Tools . . . . .	31
10.1 Cutting tool height	
10.2 Turning tools	
10.3 Threading tools	
10.4 Cut-off, thread-cutting, and facing tools	
10.5 Boring and inside threading tools	
Eleven: Setting Up with Centers, Collets, and Chucks . . . . .	33
11.1 Centering	
11.2 Mounting work between centers	
11.3 Using a clamp dog	
11.4 Using faceplates	
11.5 Setting up work on a mandrel	
11.6 Steady rests and follow rests	
11.7 Setting up work in a chuck	
11.8 Mounting work in a four-jaw independent lathe chuck	
11.9 Mounting work in a three-jaw universal chuck	
11.10 Collets and collet attachments	
11.11 Toolpost grinders	
Twelve: Lathe Turning . . . . .	40
12.1 Rough turning	
12.2 Finish turning	
12.3 Turning to shapes	
12.4 Machining square corners	
12.5 Finishing and polishing	
12.6 Taper turning	
12.7 Boring a tapered hole	
Thirteen: Lathe Facing and Knurling . . . . .	44
13.1 Facing across the chuck	
13.2 Knurling	
Fourteen: Changing Gears . . . . .	46
Fifteen: Cutting Screw Threads . . . . .	47
15.1 Threading terms	
15.2 Cutting right-hand threads	
15.3 Cutting left-hand threads	
15.4 Cutting multiple threads	
15.5 What <i>not</i> to do when thread cutting	
15.6 Finishing off a threaded end	
15.7 Cutting threads on a taper	

Sixteen: Lathe Drilling and Boring . . . . .	51
16.1 Reaming	
16.2 Boring	
16.3 Cutting internal threads	
16.4 Cutting special-form internal threads	
Seventeen: Cutting Off or Parting with a Lathe . . . . .	54
Eighteen: Milling . . . . .	55
Nineteen: Workholding . . . . .	56
19.1 Mounting to the table	
19.2 Using a vise	
19.3 Using a dividing head	
19.4 Using a rotary table	
Twenty: Holding Milling Cutters . . . . .	58
20.1 Arbors	
20.2 Collets and holders	
20.3 Adapters	
Twenty-One: Milling Cutters . . . . .	60
21.1 End mill cutters	
21.2 Plain cutters	
21.3 Side cutters	
21.4 Slitting saws	
21.5 Angle cutters	
21.6 Form-relieved cutters	
21.7 Flycutters	
21.8 Using cutting fluid	
21.9 Tool grinding	
Twenty-Two: Feeds and Speeds for Milling . . . . .	64
22.1 Speeds	
22.2 Feeds	
Twenty-Three: Common Milling Operations . . . . .	67
23.1 Milling flat surfaces	
23.2 Squaring a workpiece	
23.3 Milling a cavity	
23.4 Tapping	

Twenty-Four: Common Adjustments . . . . .68

- 24.1 Bevel gear adjustment
- 24.2 Millhead T-key
- 24.3 Lathe / Mill clutch

Twenty-Five: Troubleshooting . . . . .70

- 25.1 Powerfeed and thread cutting
- 25.2 Carriage and milling table
- 25.3 Lathe turning
- 25.4 Milling
- 25.5 Drilling
- 25.6 Drive system

Twenty-Six: Parts Lists with Schematics . . . . .77

# SECTION ONE INTRODUCTION

Congratulations on purchasing a Smithy 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-1220 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 lathe tools, hold workpieces, and do all basic machining operations.

Please read this operator's manual carefully. If you don't understand how your machine works, you may damage it, 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 operator's manual, please call Smithy. Our trained technicians will help you with any machining problems you may have. You can reach them by dialing—1-734-913-9063—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 this operator's manual, or if you have a project you'd like to share with other Smithy owners, contact the Communications Director, Smithy Co., 170 April Drive, PO Box 1517, Ann Arbor, MI 48106-1517.

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

## 1.1 Customer information

This manual should remain with your Smithy machine. If ownership changes, please include the owner's manual with the machine.

**Model #** \_\_\_\_\_

**Serial #** \_\_\_\_\_  
(on the back of the lathe bed)

**Purchase date** \_\_\_\_\_

**Delivery date** \_\_\_\_\_

**Sales technician** \_\_\_\_\_

## 1.2 Specifications

General dimensions: 42" long x 20" wide x 34" high

Crate size: 44" long x 23" wide x 38" high

Weight: shipping 550 lb, machine 375 lb

Power: 3/4 hp, 110 V

Mill / drillhead swivel: 360°

Calibrated dials

Leadscrew: 0.002"

Cross slide: 0.002"

Tailstock: 0.001"

Mill and drill: 0.040"

### Lathe

Swing: over bed 12", over table 7"

Distance between centers: 20"

Spindle bore: 1.03"

Headstock taper: Morse taper #4

Tailstock taper: Morse taper #3

Speeds: six (160 to 1600 rpm)

Thread pitches

Inch-thread: 6 to 30 tpi

Metric: 0.5 to 3 mm

Ground dovetail ways

Travel: cross slide 8", longitudinal 20"

5" three-jaw chuck with inside and outside jaws

### Mill

Millhead taper: Morse taper #3

Spindle nose to table: 6" to 12-3/4"

Spindle to lathe faceplate: 5-1/2"

Table: 5-3/4" wide x 16-1/2" long

T-slot width: 7/16"

Quill diam: 3"

Quill travel: 3-1/2"

Speeds: 12 (125 to 1600 rpm)

Working height: 6-1/4" min, 13" max

Drawbars

Standard: 3/8 to 16 tpi

Metric: 12 mm to 1.75-mm pitch

Accepts end mills up to 1-1/4" diam

### Drill

1/2" precision chuck (larger available)

Chuck arbor taper: Morse taper #3

Drill chuck to table: 2-1/4" to 9"

Spindle to table: 6-1/4" to 13"

Spindle to support column: 6.9"

Quill travel: 3-1/2"

Speeds: 12 (125 to 1600 rpm)

Cutting depth: 3-3/4"

Working height: 2-1/4" to 9"

<sup>1</sup>Specifications are subject to change.

<sup>2</sup>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, and remember that common sense is a must.

- Know your machine. Read this manual thoroughly before attempting to operate your lathe-mill-drill. Don't try to do more than you or your machine can handle. Understand the hazards of operating a machine tool. In particular, remember never to change speeds or setups until the machine is completely stopped, and never to operate it without first rolling up your sleeves or tying them at your wrists.

- Ground the machine. The CB-1220 XL has a three-conductor cord and a three-pronged grounding-type receptacle. Never connect the power supply without properly grounding the machine.

- Remove all adjusting keys and wrenches from the machine before operating. 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 from the machine while it 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 out of the machine.

- Use safety glasses, goggles, or a face shield at all times. Use glasses designed for machinery operation; regular glasses will not do. Have extras for visitors. Know when to wear a face mask and earplugs, as well.

- 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 your work. Don't reach over the machine when it's operating. Keep your hands out of the way.

- Avoid accidental starts. Turn the switch to Off before plugging in the machine.

- Secure your work. Flying metal is dangerous. Loose work can also bind tools.

- Use the recommended accessories. Understand how to use them 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.<sup>3</sup>

---

<sup>3</sup>Modifying your machine may void the warranty and create potential hazards.

## SECTION THREE

# KNOWING YOUR MACHINE

The CB-1220 XL has a 3/4-hp, 110-V motor. The motor powers the lathe and millhead through the main belt drive and the mill / drill belt drive.

A positive-lock clutch in front of the spindle pulley transfers power to either the lathe or the mill. The clutch has three positions. To power the lathe, pull the clutch handle out. To power the mill, push the clutch all the way in. The middle position is neutral. Always shift the clutch with the motor turned off.

The Smithy can use V-belts or round polyurethane belts. The latter produce less dust and are easier to change than standard V-belts.

The lathe runs at six speeds from 160 to 1600 rpm. To change lathe speeds, adjust the belts in the main belt-drive system. This system uses three pulleys: the motor pulley, floating idler pulley, and spindle pulley.

To run the lathe at lower turning speeds, use two belts. Install one between the motor and idler pulleys and the other between the idler and spindle pulleys. To increase speed, move the spindle belt in two more positions on the pulleys.

For higher speed ranges, use one long belt from the motor pulley directly to the spindle pulley. You can adjust this belt to three different positions on the pulleys.

The millhead also has a belt-drive system. It uses only one belt, adjustable to either the top or bottom position on the pulleys. These two positions, in conjunction with the main belt drive, let you select 12 speeds ranging from 125 to 1600 rpm for milling or drilling.

## SECTION FOUR

# CARING FOR YOUR MACHINE

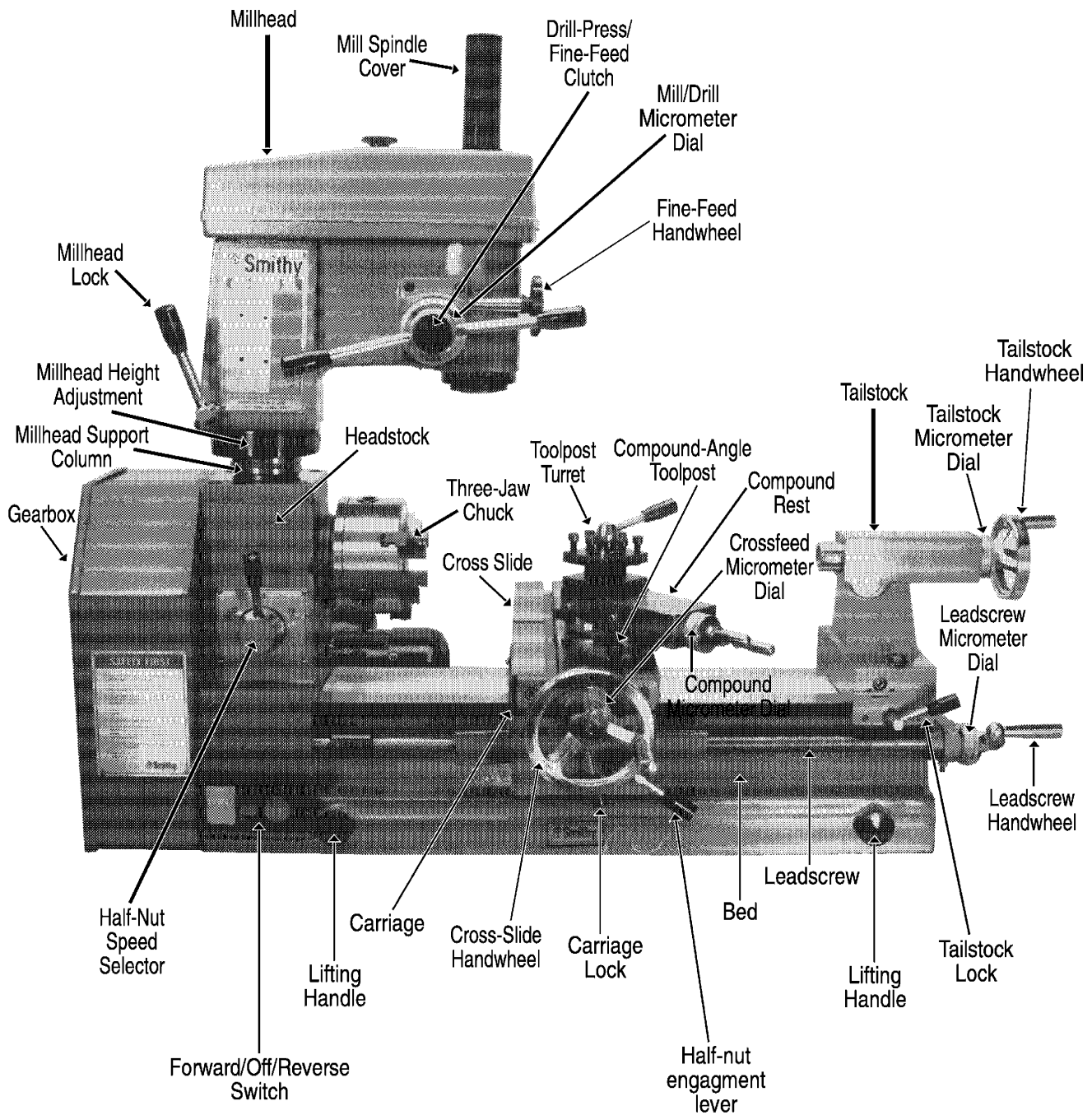
The CB-1220 XL is a delicate, precision tool with ground ways and hand-scraped bearing surfaces under the table and carriage. Any rust spot or battering of the ways, any chips or grit between close-fitting parts, will affect the accuracy of this fine tool. Follow these guidelines whenever you use your Smithy machine:

- When you finish working, wipe machined surfaces with a clean, oily rag. Never leave the machine 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 could 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 6.4*).

## SECTION FIVE BASIC PARTS OF THE CB-1220 XL

To learn the operations of your machine, you have to know the names and functions of its basic parts (Figure 5.1).

5.1 Basic parts of the CB-1220 XL.



- **Bed.** The bed (*Figure 5.1*) 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 5.1*) 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 into place by tightening the carriage lock under the cross-slide handwheel.

- **Change gears.** The change gears (*Figure 5.2*) cut different thread pitches. They also determine the feed rate. Five change gears come installed on the machine; others are packed with it.

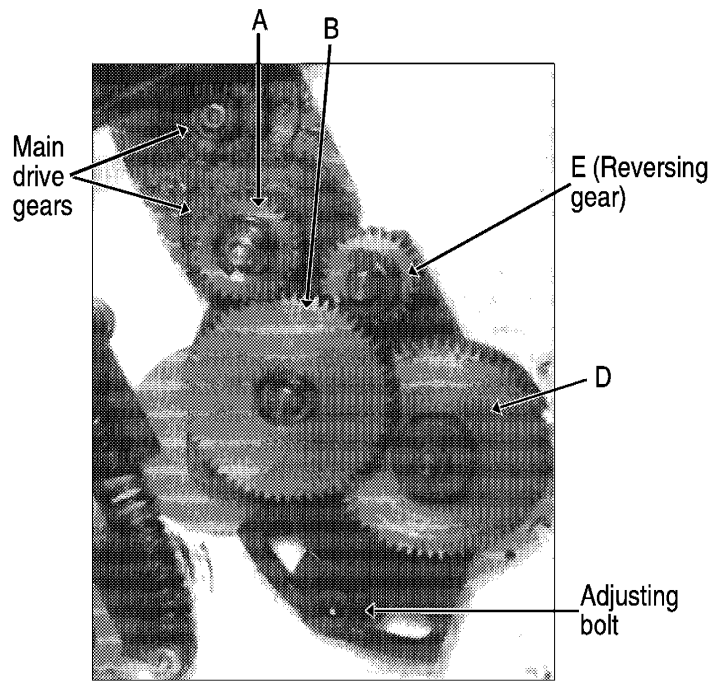
- **Compound rest.** Mounted on the cross slide, the compound rest (*Figure 5.1*) swivels to any angle horizontal to the lathe axis to produce bevels and tapers. Cutting tools fasten to a toolpost on the compound rest. The calibrations on the front of the base are numbered in degrees from 60° right to 60° left.

- **Cross slide.** The T-slotted cross slide (*Figures 5.1 and 5.3*) moves crosswise at 90° to the lathe axis by manual turning of the cross-feed screw hand-wheel. It also serves as the milling table.

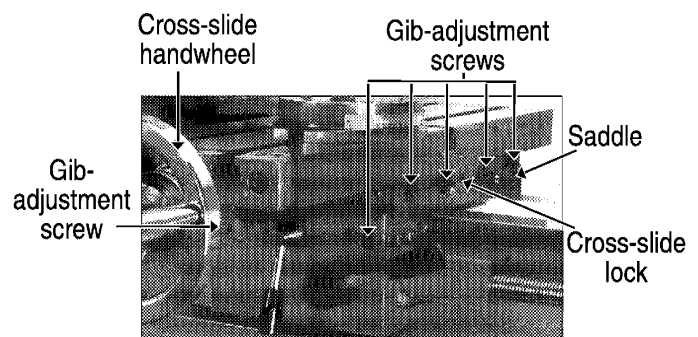
- **Drill-press and fine-feed clutch.** Pushing in the drill-press clutch (*Figure 5.1*) engages the fine feed. To work the clutch, release the spring tension by rotating the drill-press handles clockwise. Pull the clutch out to use it as a drill press or push it in to use the fine feed. Use the fine-feed handwheel to move the quill up and down.

- **Forward / Off / Reverse switch.** This is the main switch used to operate the lathe (*Figure 5.1*). It is simply a forward / reverse switch for the motor. The motor turns counterclockwise for normal lathe operation and clockwise for normal milling and drilling.

- **Gearbox.** The gearbox (*Figures 5.1 and 5.4*) houses the belts that drive the spindle and change gears for the powerfeed. Select the thread pitch (for threading) or the feed rate (for turning) by changing the four



5.2 Your machine arrives with this gear configuration. The C gear, behind B, meshes with D. Note the top two gears are permanent.



5.3 The cross slide moves laterally when you turn the cross-slide handwheel.

change gears on the right side of the gearbox.

- Half-nut engagement lever. This lever, located on the apron (*Figure 5.1*), transmits power to the carriage when rotated 90° to the right.

- Half-nut speed selector. The two-speed selector for powering the leadscrew is on the front of the headstock (*Figure 5.1*). The leadscrew turns twice as fast in the II position as in the I position.

- Headstock. The headstock (*Figure 5.1*), which is secured to the bed, houses the gears that drive the powerfeed and the taper bearings that secure the lathe spindle.

- Lathe belt tensioner. To adjust the lathe belt (*Figure 5.5*), pull the tensioner handle forward to tighten the belt, back to loosen it.

- Lathe/mill/drill clutch. The lathe/mill/drill clutch (*Figure 5.4*) is inside the gearbox. A three-position clutch, it transfers power to either the lathe or the mill, but not to both at the same time. To engage it, rotate it slowly while pushing or pulling on the clutch sleeve. Pulling it to the left runs the lathe, to the right (all the way in) runs the mill / drill. The middle is neutral.

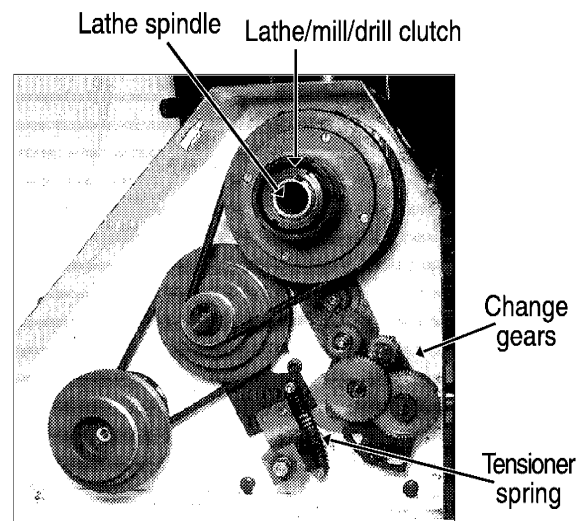
- Lathe spindle. The end of the lathe spindle facing the tailstock (*Figure 5.1*) is the spindle nose. The spindle nose, which has an MT4 taper, rotates the workpiece and drives lathe chucks and other workholding devices. All attachments, such as three and four-jaw chucks, bolt to the spindle flange either directly or via an adapter plate.

- Leadscrew. The leadscrew (*Figure 5.1*), which runs the length of the bed, moves the carriage for lathe turning or thread cutting. It works both manually and under power. You can also use it manually with the mill.

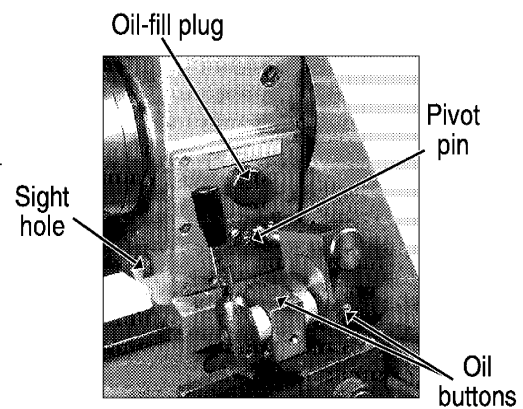
- Locks. Locks on the cross slide (*Figure 5.3*), carriage (*Figure 5.1*), quill (*Figure 5.6*), and tailstock (two), *Figures 5.1* and *5.7*, keep them from moving. During machining, lock all locks except the one on the part you want to move.

- Micrometer dial collars. Just inside the handles of the tailstock (*Figure 5.1*), crossfeed (*Figure 5.1*), drill press (*Figure 5.1*), compound feed (*Figure 5.1*), and leadscrew (*Figure 5.1*) there are collars calibrated in inches. The compound feed, leadscrew, and crossfeed are calibrated in two-thousandths, the tailstock in thousandths, and the drill press in 40-thousandths.

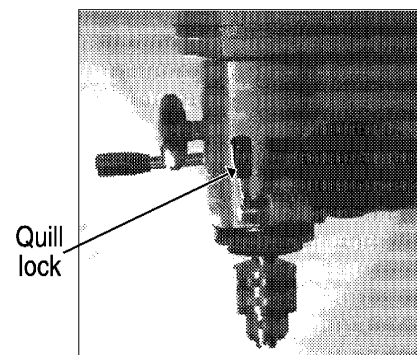
These micrometer dial collars can move independently



5.4 The gearbox houses the belts and change gears.



5.5 To adjust the tension on the lathe belt, move the tensioner handle forward or back.



5.6 The quill moves in and out of the millhead, carrying the spindle.

around the handle shafts. This independent motion is called float. Floating dials on the cross slide, tailstock, and leadscrew let you zero the collars at any point and read the feed travel from that point on the dial for added accuracy.

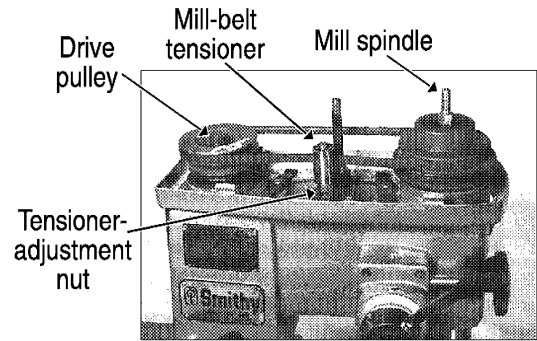
- Mill-belt tensioner. To adjust the mill belt (*Figure 5.7*), swing the roller assembly to the front and place the belt on the back of the roller. Loosen the roller assembly and slide it back and forth in its slot.

- Millhead height adjustment. Unlock the mill-head lock (*Figure 5.1*) and place the height adjustment handle in one of the three holes in the black collar. Turn the collar to raise and lower the millhead.

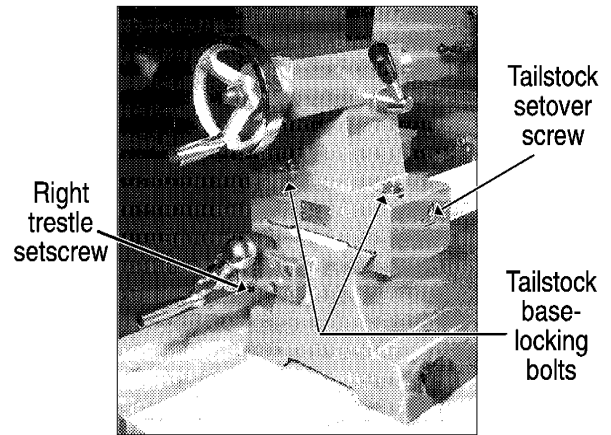
- Mill spindle. The mill spindle (*Figure 5.7*) attaches to the quill, which moves in and out of the head. The quill lock keeps the quill still when you install or remove tools from it and while milling horizontally. Usually, tools fit into collets that attach through the spindle via drawbars.

- Tailstock. The tailstock (*Figure 5.1*), which provides right-end support for the work, moves along the bed and can stop at any point. It has an MT3 taper and holds centers, drills, reamers, taps, and other tools. To move the tailstock spindle, turn the tailstock handwheel.

To offset the tailstock, adjust the two base-locking bolts (*Figure 5.8*). 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.



5.7 With the millhead cover off, you can adjust the mill-belt tension.



5.8 To offset the tailstock, adjust the base-locking bolts.

## SECTION SIX

# UNCRATING AND SETTING UP THE CB-1220 XL

### 6.1 Moving the machine

Moving a machine tool can be dangerous. Improper techniques and methods may injure you and / or damage the machine. To find a professional to move and site your Smithy machine, 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 a claims process.

Trucking company terminals usually have forklifts to assist customers. It's most convenient to transport the machines in trucks without canopies and large vans.

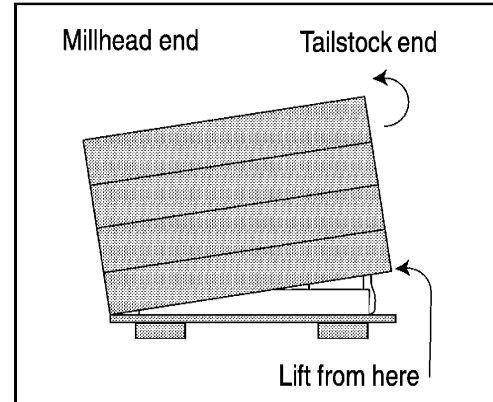
### 6.2 Uncrating and positioning the machine

The machine is assembled, inspected, and ready to go on its stand when you receive it. It's wrapped in a water and greaseproof cover, strongly braced, and crated. A box of accessories is also in the crate.

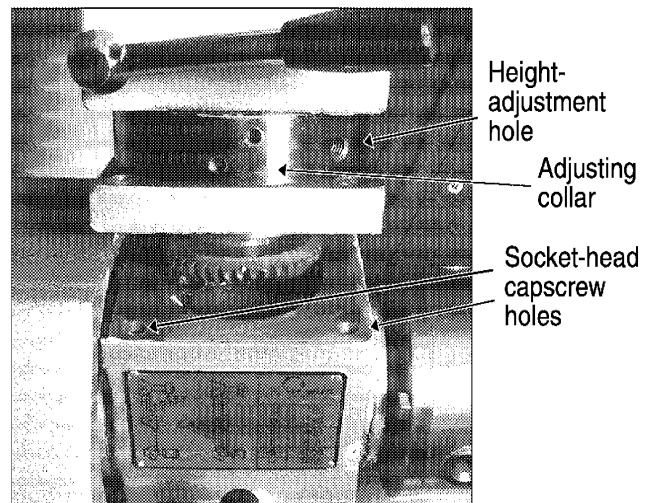
The metal bands around the crate are under tension. Wearing eye protection and gloves, cut the metal bands with tin snips. Be careful—the cut edges are sharp. The bands secure the crate top to the base.

After removing the straps, lift off the crate top. Tip the crate from the tailstock end up and over the machine (*Figure 6.1*). Do not damage the crate; you may need it another time to transport the machine.

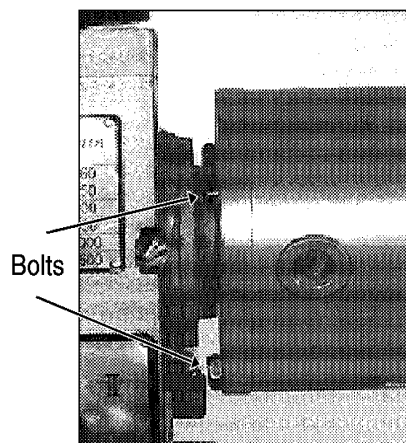
Now open the accessories box. Check the items in it against the accessory checklist. After



6.1 Tip the crate from the tailstock end up and over the machine.



6.2 Remove the millhead and column from the lathe head.

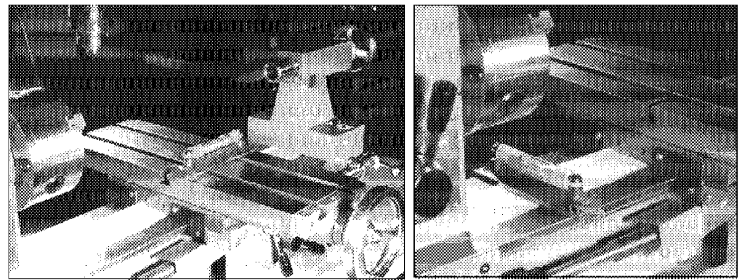


6.3 The chuck attaches to the spindle flange with three bolts. The one bolt located on the other side of the spindle does not show.



accounting for all parts, you're ready to move your CB-1220 XL into its work position.

Four men can move the CB-1220 XL using the four lifting handles (Figure 5.1). You can reduce the weight so two people can move it by following these instructions:

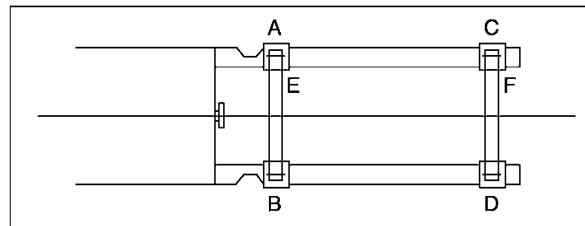


6.4 Check along and across the bed to make sure it is level.

### Millhead

- Remove the four hexagon socket-head capscrews at the base of the millhead support column (Figure 6.2). If a screw runs through the belt box into the flange of the support column, remove it, too.

- Lock the millhead locking handle (Figure 5.1).
- Lift the millhead and column off the lathe head (Figure 6.2). You may have to rock it back and forth while lifting it.



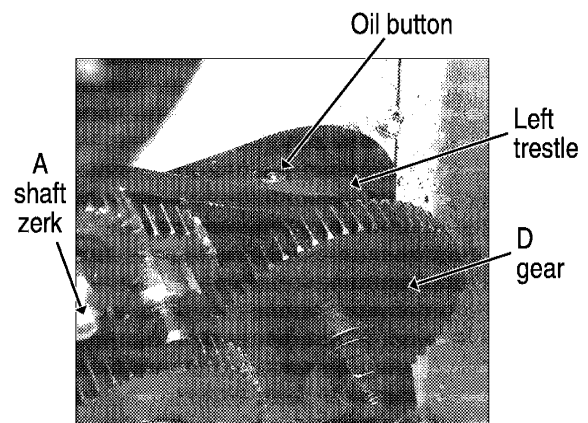
6.5 To check bench and bed level accuracies, successively place level at A, B, C, D (longitudinal positions) and E and F (transverse positions). Bedways alignment in the longitudinal plane should be better than 0.0016/40"; alignment in the transverse plane should be better than 0.0024/40".

### Tailstock

- Loosen the tailstock locks (Figure 5.1) and pull the tailstock off the end of the bed. The gib and locking pin will fall out. Be careful not to lose them.

### Three-jaw chuck

- Remove the three bolts behind the chuck that hold it to the spindle flange (Figure 6.3). The chuck will come off. Place a board between the chuck and ways to protect the ways.



6.6 Oil the button behind the D gear.

Put the machine on a strong, rigid table 40" long x 24" wide x 28–33" high. We recommend you bolt down the machine using the holes in the base of the bed or using the lifting handles the same way they held the machine to the shipping pallet.

Carefully lift the machine by the handles, move it over the stand, and lower it into position. Do not let any part of your body come between the machine and the stand. Bolt the machine to the stand, using one flat washer and one lockwasher per bolt.

Before permanently anchoring the machine, you may want to level the bed (Figure 6.4). The bed is rigid and supports itself, but having a level bed simplifies many setup operations. Use a precision level, both along and across the bed (Figure 6.5). Shim up any low points with sheet metal or other noncompressible material. After tightening the anchor bolts, check the bed again.

## 6.3 Selecting a location

There are several major considerations when selecting a location for your Smithy:

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

- The machine should be on a 30-amp circuit and close to the power outlet. If you must use an extension cord, check with an electrician to make sure the cable can handle the electrical load.

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

- Place the machine on a solid foundation—concrete, if possible. If the floor is wood, make sure it can support the machine and workbench. Brace it if necessary to prevent sagging or settling.

- 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.

#### 6.4 Cleaning and lubricating the machine

Smithy machines are shipped with a protective grease coating. To remove it, spray on WD-40, let it sit for a few minutes, and wipe it off with rags. Use a brush and noncorrosive kerosene or white mineral spirits to clean hard-to-reach places.

Give special attention to the leadscrew. Use a brush or cotton string to clean down into the threads.

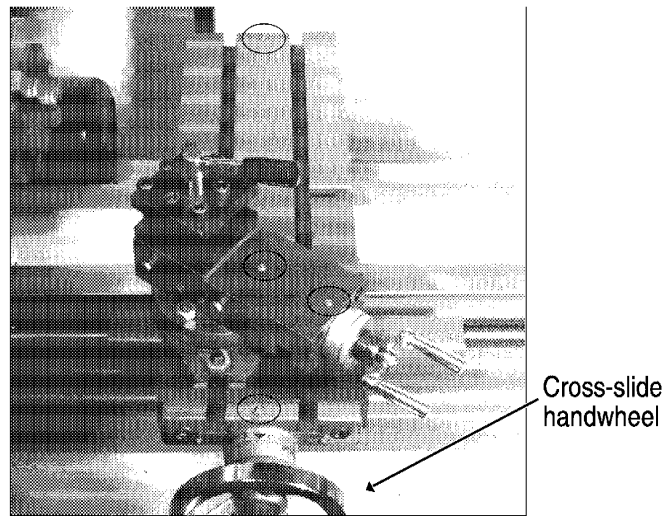
The best way to clean the powerfeed gears is to remove them completely. Wipe the pulleys with a damp rag.

Once it's cleaned, your Smithy is ready for lubricating. Do this carefully and thoroughly before starting the machine. Use a pressure oil can and good-quality SAE No. 20 or 30-weight machine oil on the bearings and headstock.

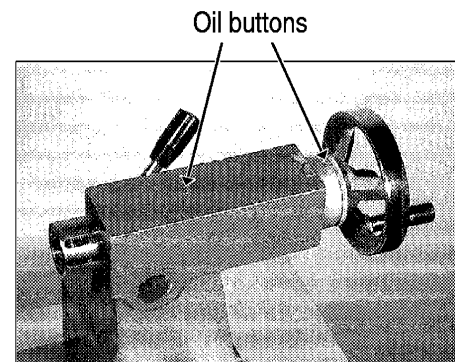
To be thorough and complete, follow this routine:

##### Oiling the headstock

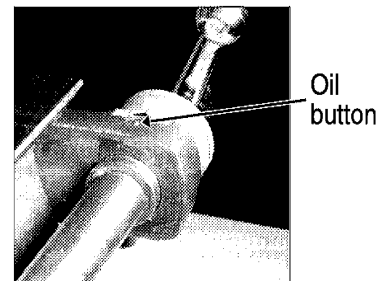
- Open the gearbox door to expose the change gears. Oil the button in the casting behind the D gear (Figure 6.6). Then put a few drops of oil on the teeth of all the gears. Grease the zerk on the A gearshaft.



6.7 Oil the buttons (circled) along the cross-feed table and cross slide.

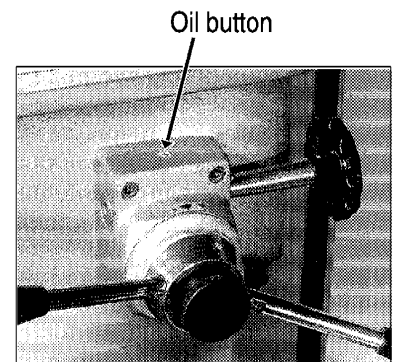


6.8 Oil the two buttons on the top of the tailstock.



6.9 Oil the support for the right end of the leadscrew.

- Check the sight glass under the chuck. If necessary, add oil until it is half full. The oil-fill plug is at the back of the headstock above the motor (Figure 5.5). Be careful not to overfill it. If you have to top it off with oil, pour in only an ounce at a time and wait to see the results in the sight glass. Too much oil will make the motor lug and sling oil out from behind the chuck and inside the belt box.



6.10 Oil the clutch-housing button.

### 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 Way-lube®, an oil specially formulated for ways.

### Oiling the carriage

- Lubricate the oil buttons in the cross-slide table (Figure 6.7). There are two buttons 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 (Figure 6.8).

### Oiling the apron

- Put oil in the button just behind the cross-slide handwheel (Figure 6.7).

- Put oil on the button at the back of the cross slide (Figure 6.7).

### Oiling the leadscrew

- Put oil in the oil buttons in the left trestle (Figure 6.6).

- Put oil in the support for the right end of the leadscrew (Figure 6.9).

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

### Oiling the compound

- Put oil in the two buttons on the top.

**6.11 Setting Lathe Speeds (RPM)**

Low	C	D	E	High	F	G	H												
				<table border="1"> <tr> <td>C</td> <td>160</td> </tr> <tr> <td>D</td> <td>250</td> </tr> <tr> <td>E</td> <td>400</td> </tr> <tr> <td>F</td> <td>630</td> </tr> <tr> <td>G</td> <td>1000</td> </tr> <tr> <td>H</td> <td>1600</td> </tr> </table>				C	160	D	250	E	400	F	630	G	1000	H	1600
C	160																		
D	250																		
E	400																		
F	630																		
G	1000																		
H	1600																		

**6.12 Setting Mill/Drill Speeds (RPM)**

C	D	E	F	G	H
A		B			
AC	125	BC	160		
AD	200	BD	250		
AE	315	BE	400		
AF	500	BF	630		
AG	800	BG	1000		
AH	1250	BH	1600		

## Oiling the mill / drill clutch

- Put oil in the button on top of the clutch housing (Figure 6.10).

To keep your machine in peak condition, lubricate it daily after removing any debris.

## 6.5 Setting up your CB-1220 XL

The CB-1220 XL comes with all major components assembled, but it is not ready to use right out of the crate. Do not start the motor until you correct the positions of the cross-slide and leadscrew handwheels (Figure 5.1). We reversed these handles at our warehouse to protect them during shipment. A drop or two of oil on the shafts will help the handles slide on. Starting the motor with these handles in their shipping positions will damage the machine's gears, bearings, and handles. You must also install the tailstock handwheel, two drill-press handles, millhead lock handle, and millhead height-adjustment handle.

## 6.6 Setting lathe and mill speeds

Changing belts changes lathe speeds. The lower speeds use the two short belts. There is only one position for the motor-pulley-to-idler-pulley belt. It goes on the smallest sheave of the motor pulley (behind the largest sheave, Figure 6.11) and on the largest sheave of the idler pulley. For 160 rpm, set the idler-pulley-to-lathe-spindle-pulley belt on the smallest sheave of the idler pulley to the largest sheave of the spindle pulley (position C). Move it in one sheave for 250 rpm (position D) and one more for 400 rpm (position E).

For the higher speeds, remove the two small belts and use the single long belt from the motor pulley to the spindle pulley. For 630 rpm (position F), run the belt from the outside sheave (closest to the door) on the motor pulley to the largest sheave on the spindle pulley. Move it in one sheave for 1000 rpm (position G). For 1600 rpm (position H), run it from the largest motor-pulley sheave to the smallest spindle-pulley sheave.

Set mill speeds using various combinations of the lathe belts and the belt on top of the millhead. For 125 rpm, place the mill belt in position A and the lathe belts in position C (Figure 6.12). For 160 rpm, place the mill belt in position B and the lathe belts in position C, etc.

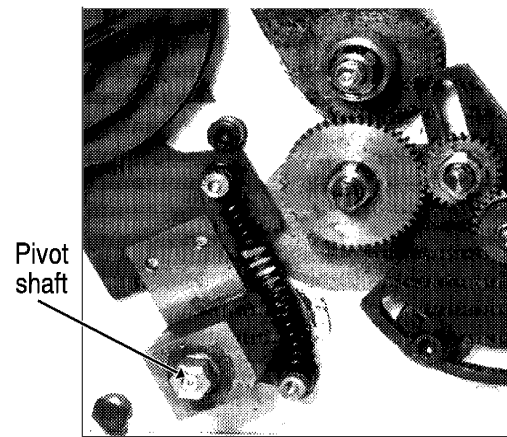
## 6.7 Adjusting belt tension

To get maximum performance from your machine, keep the drive belts snug.

To adjust the tension on the mill belt, swing the roller to the front and place the belt on the back of the roller. Loosen the nut at the bottom of the roller and slide the roller in its shaft to the desired position. Tighten the nut (Figure 5.7).

When you use only the single long belt, the spring at the bottom of the idler-pulley bracket holds the idler pulley so it does not fall onto the motor pulley. To adjust the tension on the spring, loosen the pivot shaft (Figure 6.13) and tighten or loosen the spring as needed. Then retighten the shaft.

To tighten the lathe belts, move the tensioner handle above the motor (Figure 5.5) so it points



6.13 Adjust the spring tension on the idler-pulley bracket.

toward the lathe head. Turn the knurled knob clockwise to tighten the belt and counterclockwise to loosen it. If there is not enough adjustment, remove the pivot pin and turn the knob as needed. Then reattach the pin.

## 6.8 Adjusting the gibs

The CB-1220 XL has straight gibs. Before using the machine, adjust the gibs to compensate for wear and maintain the proper fit between sliding surfaces. Gib adjustment affects cutting tool rigidity and the machine's ability to make accurate cuts.

As the gibs tighten, the effort it takes to turn the handwheels increases. Adjust the gibs according to the work you are doing and personal preference. What's important is to adjust them evenly. The tighter the gib, the more accurate it will be. Removing and polishing the gibs also improves the tolerances.

Before beginning, make sure the ways are clean and well-oiled. You must also understand the locks for the compound, cross slide, carriage, and tailstock act directly on the gibs for their locking power. Back these locks off completely.

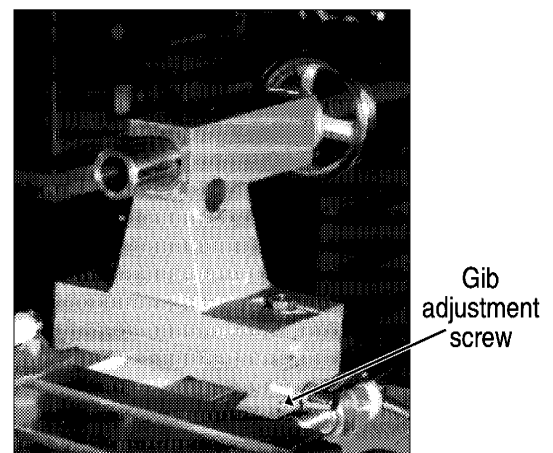
*Cross-slide gib.* Start adjusting the cross-slide gib with the table centered on the carriage. Back off all setscrews and jam nuts. Tighten the two inside setscrews all the way (*Figure 5.3*), locking up cross-slide movement. Then back off each setscrew one-quarter turn and check the movement. Finally, set the tension on the outer screws to match.

The effort it takes to move the table should be the same in both directions. If it is not, the gib is not adjusted evenly. If you feel more handwheel resistance when the cross slide is going away from you, the leading edge of the gib is too tight. Back off the setscrew closest to you a little to relieve the tension. If there is more resistance when the table is coming toward you, the leading edge of the gib is too tight, and you should adjust the screw furthest from you. When everything is set, hold the setscrews carefully with the Allen key and lock the jam nuts.

*Carriage gib.* The carriage has only two gib-adjusting screws, accessed through holes in the front of the apron (*Figure 5.1*). Start the adjustments as you did with the cross slide: tighten the screws all the way, then back them off one-quarter turn and test the setting. Handwheel resistance should be even in both directions.

The big difference between the cross slide and carriage is that the carriage gib travels with the carriage as it moves down the lathe bed. If you have more handwheel resistance when the carriage moves toward the tailstock, the leading edge of the gib is on the right side of the carriage, the same side as the direction you are moving the carriage. If you have more handwheel resistance when the carriage is moving toward the headstock, to the left, release the tension a bit on the left-hand gib setscrew. As you work with the adjustments, you'll feel the difference even gib tension makes on the handwheels.

*Compound gib.* The compound gib has two adjusting screws and jam nuts (*Figure 5.3*). For greater tool rigidity, you can adjust the compound gib a bit tighter than the others.



6.14 You can adjust the tailstock gibs with two setscrews, one on each side of the tailstock.

*Tailstock gib.* The tailstock gib also has two adjusting screws (*Figure 6.14*). This gib can be freer than the others so it is easier to position the tailstock. Again, the lock acts on the gibs. When tightened, it automatically brings the tailstock into alignment.

## 6.9 Reducing backlash

Backlash of 0.008–0.015" as measured on the micrometer dials is normal. If you have more backlash than that, refer to the schematics at the back of this manual, if necessary, and follow these directions:

### Crossfeed

- Tighten the setscrew on the bottom of the screw seat so the bushing inside the screw seat cannot move.
- Tighten the cap nut so the handwheel is secure.
- Turn the handwheel one way, then the other. If a gap opens and closes between the dial and screw seat, you must install one or more shim washers (ask a Smithy technician about our antibacklash shim washer kit, #K99-190).

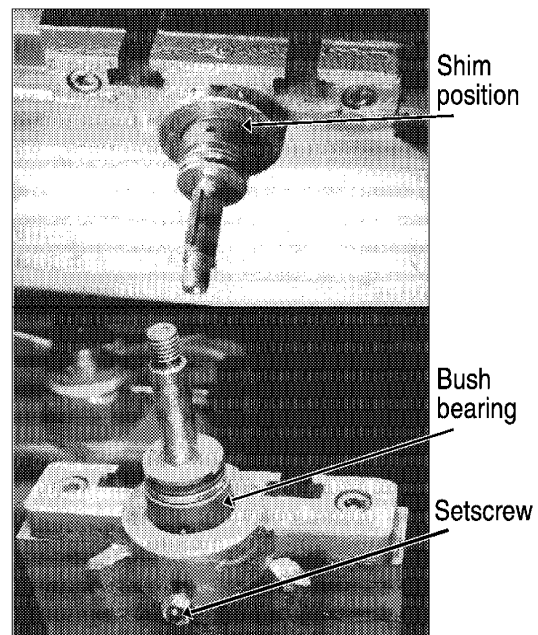
To install shims, remove the handwheel, key, washer, and the outer part of the dial and spring. Remove the inner hub of the dial. Install one or more shim washers between the bearing and dial hub (*Figure 6.15*) and reassemble. Tighten the setscrew on the bottom of the screwseat as well.

- Push and pull on the cross slide. If there is movement, remove the two bolts that attach the rear screw seat to the cross slide, remove the screw seat, and screw the cross slide toward you until the screw comes completely out of the brass nut (*Figure 6.16*).

- Remove the brass nut and put one or two strips of shim stock in the side of the hole. The fit should be tight. Screw the cross slide back onto the carriage. Adjust the screws in the brass nut to remove any play between the threads in the nut and screw. Reassemble the screw seat onto the cross slide.

### Longfeed

- Tighten the setscrew on the bottom of the right trestle so the bushing is tight (*Figure 5.7*).
- If there is still excess backlash, remove the cap nut, hand crank, and key.
- Remove the washer, outer dial, spring, and inner-dial hub key.



6.15 You can reduce handwheel backlash with a shim (top) or by tightening the setscrew on the bottom of the screwseat (bottom). This locks the bush bearing, holds the screw, and reduces any backlash.

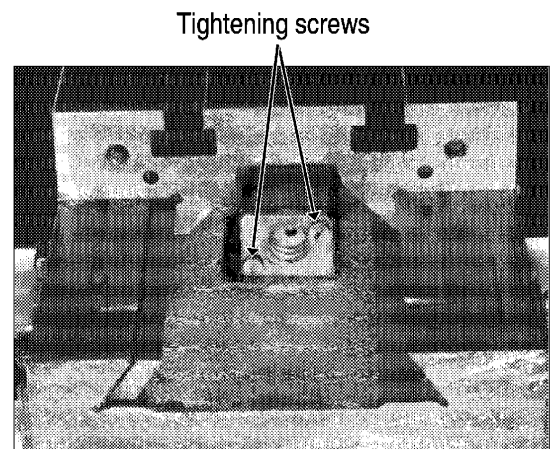
- Install as many shim washers as possible between the bearing and dial. Then reassemble.

## 6.10 Running in

Though all Smithy machines are tested at the factory and again before shipping from the warehouse, it is wise to put your machine through a break-in run before starting to work. After oiling the machine, check the belts to make sure the tension is correct. Do not plug in the machine yet.

Follow these steps:

- Set the lathe to 160 rpm.
- Plug the machine into a grounded 30-amp circuit.
- Start the motor by pushing in on the green button. To reverse the motor, push the red button to stop it, lift the cover over the rocker switch, and push the rocker switch either up or down to reverse the motor's rotation.
- During the run-in, try the controls. Get a feel for your machine.



6.16 Adjust the tightening screws on the two-piece crossfeed nut to reduce the lash between the screw and nut.

## SECTION SEVEN 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.

The modern lathe offers the following:

- The strength to cut hard, tough materials
- The means to apply power

7.1 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

7.1 Table provides exact speeds (rpm). It does not take machine speed limitations into account. Determine the desired rate of speed and find the closest speed available on your machine.



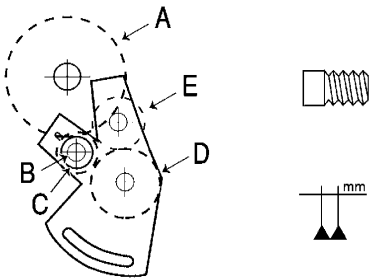
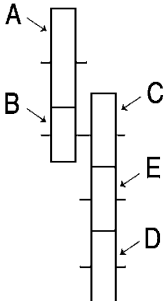
- The means to hold the cutting point tight
- The means to regulate operating speed
- The means to feed the tool into or across, or into *and* across, the work, either manually or by engine power, under precise control
- The means to maintain a predetermined ratio between the rates of rotating works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

### 7.1 Turning speeds

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 7.1*). 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.

Lathes cut threads in various numbers per inch of material threaded, according to the operator's

7.2 Gear Ratios											
											
Threads Per Inch					Metric Pitches						
Speeds		Gear Position				Speeds		Gear Position			
1	2	A	B	C	D	1	2	A	B	C	D
.003	.006	30	60	27	63	0.50	1.00	49	32	42	56
30	15	70	32	40	45	0.60		49	32	45	50
28	14	70	32	40	42	0.70		49	32	42	40
26	13	70	32	40	39	0.75	1.50	63	32	42	48
24	12	70	32	40	36	0.80		49	50	60	32
22	11	70	32	40	33		1.75	49	48	63	32
20	10	70	32	40	30	1.00	2.00	63	32	49	42
18	9	70	27	45	36	1.25	2.50	60	32	49	32
16	8	70	27	45	32	1.50	3.00	63	32	56	32
14	7	60	27	60	32						
12	6	70	27	60	32						

needs. The CB-1220 XL cuts threads to metric or inch standards.

In thread cutting, the carriage carries the thread-cutting tool and moves by rotating the leadscrew (*Figure 5.1*). The basic principle is that the revolving leadscrew pulls the carriage in the desired direction and at the desired speed. The carriage transports the toolrest and the threading tool, which cuts the thread into the metal 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 7.2* shows belt positions for various speeds.

## 7.2 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, or to advance the tool a specified amount each revolution (feed rate expressed as inches per revolution [ipr]).

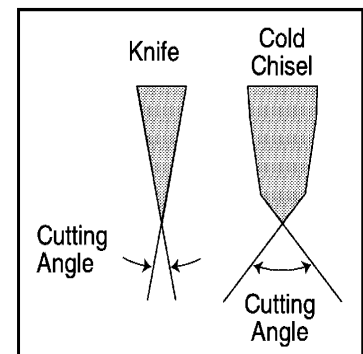
The CB-1220 XL has a pick-gear gearbox (*Figure 5.2*); gears are picked and placed to change the gear ratios. The gearbox mechanism determines the leadscrew's rotation rate in relation to the spindle's for threading, turning, and facing. To change the thread pitch (tpi), replace the gears per *Figure 7.2*.

## SECTION EIGHT METALCUTTING THEORY

A machine tool is no more efficient than its cutting edge. Because lathe operations require continuous regrinding and resharpenering 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 is needed to wedge it through the material. 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 8.1*, 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.



8.1 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.

The primary requirements of the cutting edge of any metalcutting tool are that it 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.

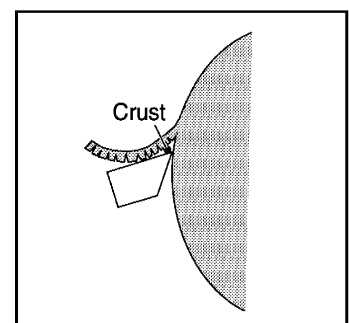
While the workpiece revolves, a strong, rigidly held cutting edge is forced under its surface. As it presses down against the cutting edge, small chips or a continuous ribbon of metal wedges away (*Figure 8.2*). Only in soft, ductile materials is this wedging action continuous.

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.

### 8.1 Tool sharpness

Instead of being the all-important factor in determining tool performance, keenness of the cutting edge is just one of many factors. 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



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

increasing its wedging action. Cutter shape is usually more important than edges, which generally are rough-ground and require honing 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, is 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 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-1220 XL has a four-sided turret toolpost that accommodates up to four high-speed-steel (HSS), carbide-tipped, or indexable carbide turning tools.

## 8.2 Heat

The energy expended at the lathe's cutting point converts largely into heat, and because the energy is great, the heat is intense. Before 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 your machine is running at extremely high speeds on continuous, heavy-duty production work. HSS tools are self-hardening even when red hot. 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 with long shafts, to check the tightness of the lathe centers often 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 workpiece 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. Consider installing a coolant system if you don't have one.

8.3 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)						
<i>Roughing</i>	90	50	45	200	70	100
<i>Finishing</i>	120	65	60	300	80	130
Feed (ipr)						
<i>Roughing</i>	0.010–0.202	0.101–0.020	0.010–0.020	0.015–0.030	0.010–0.020	0.010–0.020
<i>Finishing</i>	0.003–0.005	0.003–0.005	0.003–0.005	0.005–0.010	0.003–0.010	0.003–0.010

Figure 8.3 lists cutting speeds and feeds for HSS cutters so you can set up safe rpm rates. The formula is as follows:

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

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 8.3 is 300. Then

$$\text{rpm} = 300 \text{ sfm} \times 4 / 1$$

$$\text{rpm} = 1200 / 1$$

$$\text{rpm} = 1200 \text{ or next slower speed.}$$

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

$$\text{rpm} = 50 \text{ sfm} \times 4 / 1$$

$$\text{rpm} = 200 / 1$$

$$\text{rpm} = 200 \text{ or next slower speed.}$$

With the four-turret toolpost, you can install all standard-shaped turning and facing tools with 1" or smaller shanks. The centerline is approximately 5/8" above the bottom of the turret. Smithy also offers quick-change tool sets that greatly speed up lathe operations. Contact a Smithy technician for details.

## SECTION NINE

# GRINDING CUTTER BITS FOR LATHE TOOLS

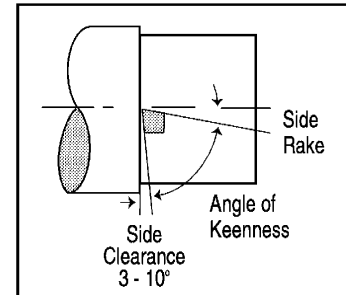
### 9.1 High-speed-steel cutters

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, including squares, flats, and bevels. Many shops buy their cutters as ready-ground or ready-to-grind bits or blades.

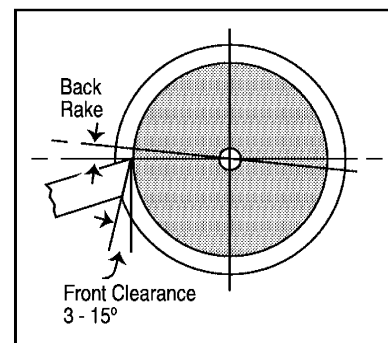
Ready-to-grind bits and blades are of specially selected HSS, cut to length and properly heat-treated. They are fine tools in the rough and generally superior to HSS shapes sold by the pound.

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.



9.1 Keenness angles vary from 60° to 90°.

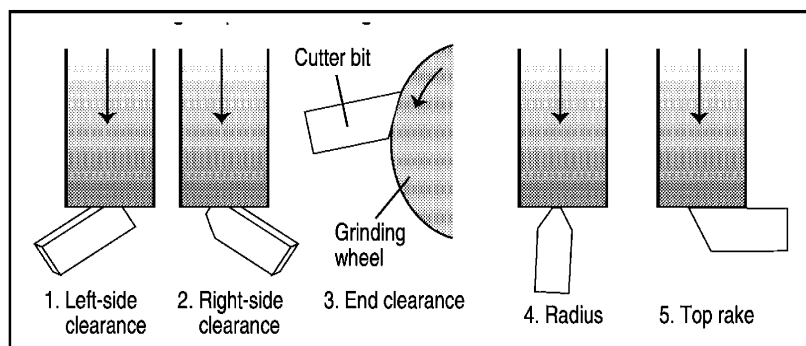


9.2 The edge weakens if front clearance is too great.

Keenness angles can vary from 60° for mild softness to 90° for hard steels and castings (Figure 9.1).

Front clearance must always be sufficient to clear the work. If it is too great, however, the edge weakens and breaks off (Figure 9.2). 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-grit or 68A-grit Carborundum wheel) at 6000 or 6500 rpm. When starting with an unground cutter bit, the procedure (Figure 9.3) is usually to grind the (1) left-side clearance, (2) right-side clearance, (3) end clearance, (4) radius, and (5) top rake, touching in a chipbreaker. If you are honing the cutting edge (for fine finishing or machining soft materials), draw the cutter away from the cutting edge across the



9.3 This is the grinding sequence for an unground cutter bit.

oilstone (*Figure 9.4*).

## 9.2 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 edge. For soft steels, angles as acute as  $61^\circ$  are possible (*Figure 9.5*).

The same general rule applies to cast iron. Chilled or very hard cast iron requires tools with cutting-edge angles as great as  $85^\circ$ . For ordinary cast iron, you obtain greatest efficiency with a more acute cutting edge—approximately  $71^\circ$  (*Figure 9.6*).

## 9.3 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.

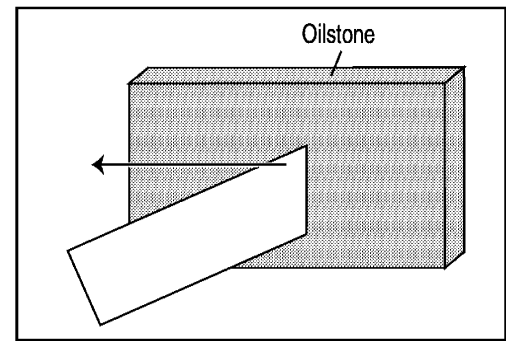
## 9.4 Special chip craters and chipbreakers

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 9.7*).

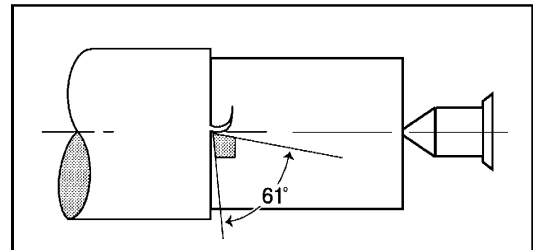
## 9.5 Using a center gauge to check V-thread forms

It may be convenient to grind a standard cutter bit for thread cutting, especially for cutting standard  $60^\circ$  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.

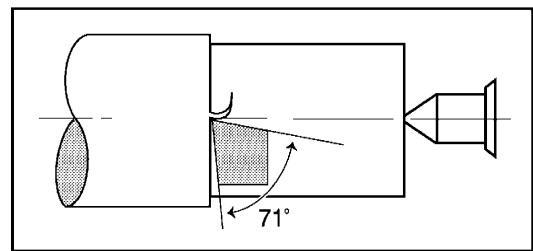
To grind a cutter for an ordinary V-thread, grind first the left side of the tool, then the right side, to  $30^\circ$ . Be careful to grind equally from both sides to center the toolpoint. Then test for true form by inserting the newly ground point in the closest-sized V in a standard center gauge (*Figure 9.8*). Examine the gauge and cutter before a light. When the cutter is ground perfectly, no light streak shows between tool and gauge. Use a grinding chart for other rakes.



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



9.5 With soft steels,  $61^\circ$  angles are possible.

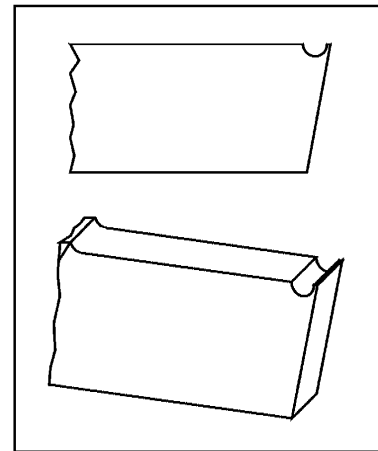


9.6 With cast iron, a  $71^\circ$  angle is most efficient.

### 9.6 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.

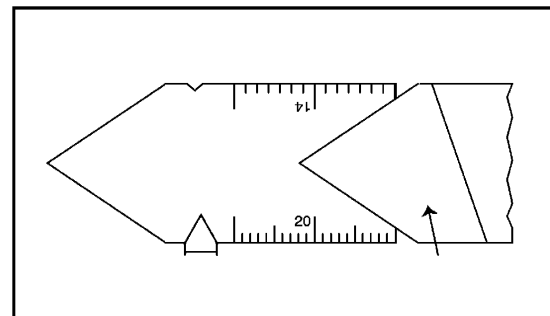
Thread forms must be accurate if threads are to fit snugly and smoothly. Every resharpening of this type of cutter requires regrinding the entire form. It is far better, when doing any amount of threading, to use a threading tool with a special form cutter. Sharpening such cutters requires only flat,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



9.7 A crater starts the chip curling smoothly.

### 9.7 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 industry is for long production runs on ordinary steels. On such work, carbide-tipped tools permit higher running speeds and much longer runs between resharpenings. The cutting edge of carbide tools stands up 10 to 200 times as long as the edge of HSS tools (Figure 9.9).



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

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 must have adequate support in the toolpost to prevent vibration and breakage.

9.9 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

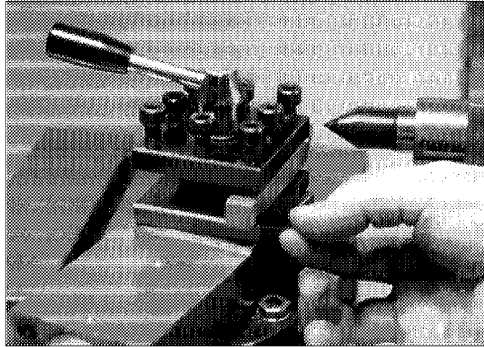
\*C-2 and C-6 are the most commonly used carbides.



## SECTION TEN 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. 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.



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

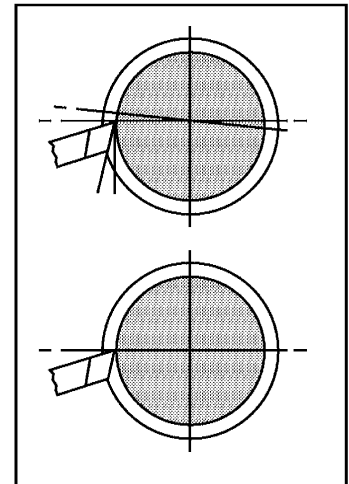
### 10.1 Cutting-tool height

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 10.1*). 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.

### 10.2 Turning tools

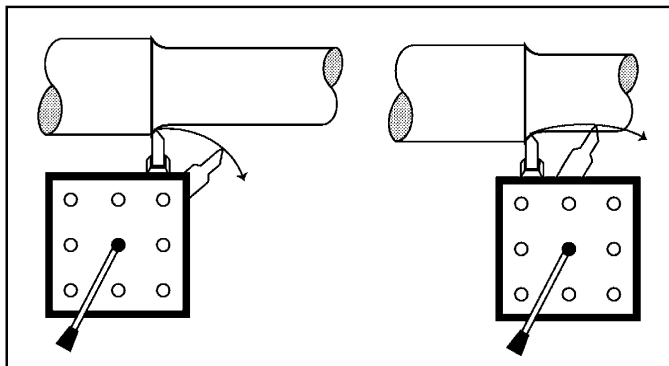
For general turning operations, set the point of the cutter bit slightly above the centerline of the work. In steel, the harder the material, the less above center (*Figure 10.2, top*). Exceptions are soft brass, aluminum, and materials that tend to pull or tear. When machining these materials, set the cutter on dead center (*Figure 10.2, bottom*).

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  $29-1/2^\circ$  left of perpendicular to the line of centers, except for extremely heavy, rough forcing



10.2 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.

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 10.3*).

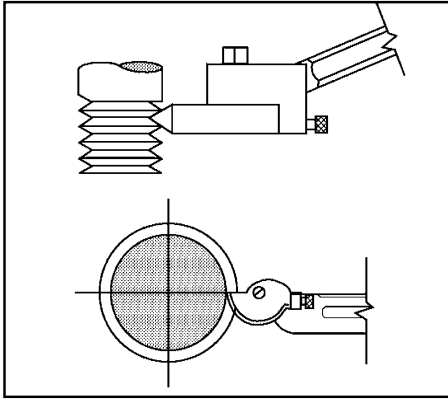


10.3 The tool will swing out of the cut (left) rather than hog into the work (right) if you reach a stalling point. Note the tool is in the right-hand side of the toolpost.

### 10.3 Threading tools

Threading tools should always engage the workpiece on dead center. Any deviation above or below will affect the thread profile (*Figure 10.4*).

### 10.4 Cut-off, thread-cutting, and facing tools



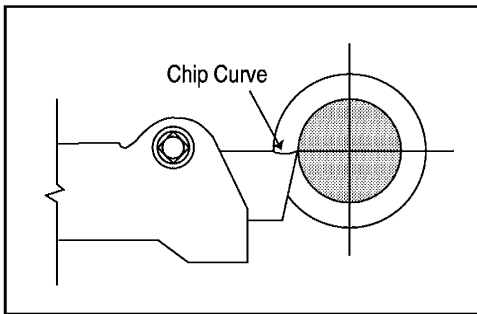
10.4 Threading tools engage the work on dead center.

For cutoff, thread cutting, and facing, feed the cutter to the work on dead center (*Figure 10.5*). For the beginner, the average feed should not exceed 0.002 inches per revolution (ipr).

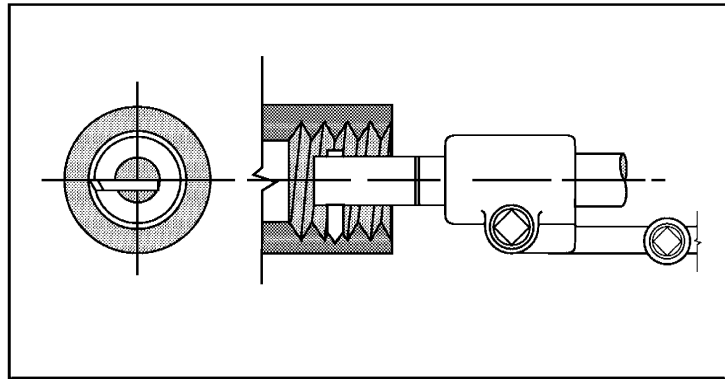
### 10.5 Boring and inside threading tools

For boring and inside threading, the cutter point engages the work on dead center (*Figure 10.6*). For greater cutting efficiency, position the bar while parallel to the line of lathe centers sufficiently below center to give the cutter a 14-1/2° approach angle. For internal threading, grind the top face of the cutter 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.



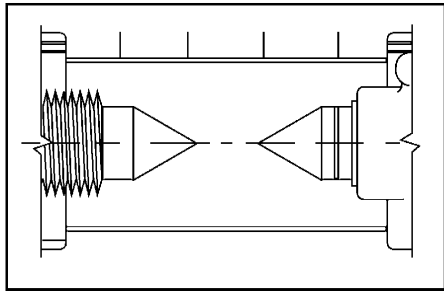
10.5 Feed the cutter on dead center for cutoff, thread cutting, and facing.



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

## SECTION ELEVEN

# SETTING UP WITH CENTERS, COLLETS, AND CHUCKS



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

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 11.1*). You can correct any lateral alignment error by adjusting the tailstock set-over screws (*Figure 5.8*).

the center of the bar or block. Locating these holes is called centering.

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

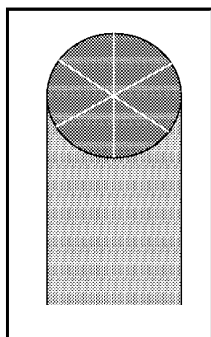
### 11.1 Centering

You can improve centering greatly by first squaring or facing the ends of the workpiece (see Section 13.1). This gives you a true cross section in which to locate the centering holes. First, chuck the stock, letting it 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 and perhaps cause the center drill to run off center.

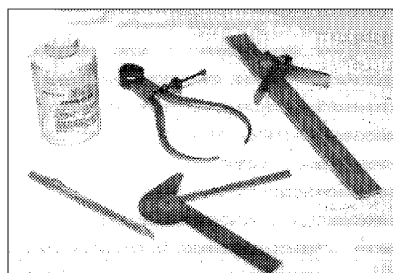
Start your lathe on the slowest speed. Bring the tool into the cutting position against the center of the workpiece. Feed the tool from the center of the stock outward, toward yourself, using the hand crossfeed. One or two light cuts is usually enough to true up an end roughened by the hacksaw. After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

You can center on round stock (*Figure 11.2*) with calipers, dividers, or special centering instruments (*Figure 11.3*). Centering square or rectangular stock is done by scribing lines from opposite corners. The intersection of these lines is the center (*Figure 11.4*).

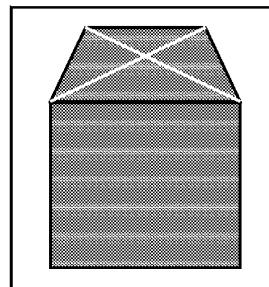
After locating the center of each end, drive a starting depression for the drill into the stock with a center punch. Check centering accuracy by placing the workpiece between the spindle and tailstock centers. Revolve the headstock slowly against the tip of a tool or a piece of rigidly held chalk. The chalk should touch just the high spots (*Figure 11.5*). If the center is off 0.002" or



11.2 Centering on round stock.



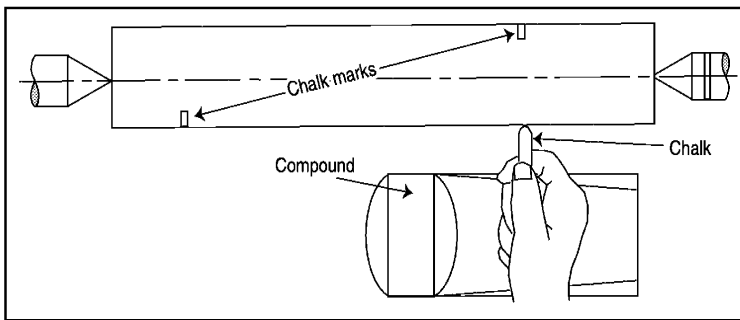
11.3 Use centering instruments include calipers and dividers.



11.4 Centering on square or rectangular stock.

more, correct its position by repunching at an angle.

Next, drill and countersink the centers to conform to the profile of the lathe centers. This is best done with a



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

combination center drill / countersink held in the tailstock arbor chuck. The centers now will take the lathe centers without play or chatter.

If a combination drill is not available, you can drill centers with a small drill and countersink them with a drill of sufficient diameter ground to a 60° point. A 60° taper is standard for lathe centerpoints. Correct center depth is given in *Figure 11.6*. Take care to get an accurate 60° countersink in the center (*Figure 11.7*).

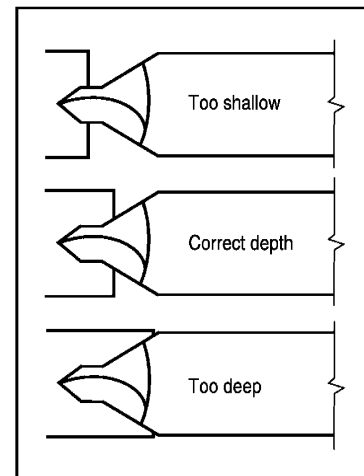
## 11.2 Mounting work between centers

Remove the chuck from the lathe, bolt the faceplate to the spindle flange (*Figure 11.8*), and put in both headstock and tailstock centers. Fasten a lathe dog (*Figure 11.9*) 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. 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.

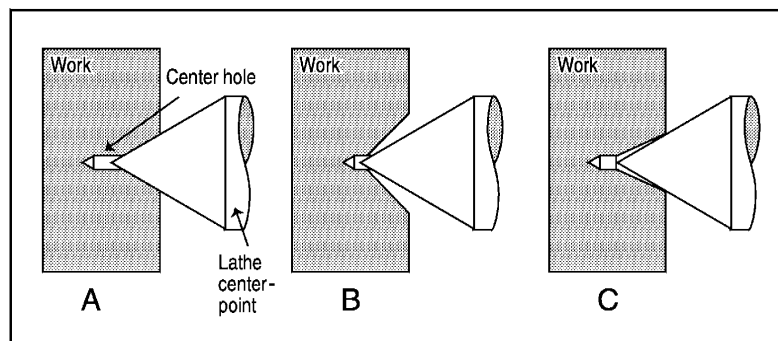
If, after partially machining the workpiece, you find you must machine the stock under the lathe dog, remove the workpiece from the lathe and place the lathe dog on the machined end. Then turn this new tailstock center end of the shaft down to the desired diameter or form.

## 11.3 Using a clamp dog

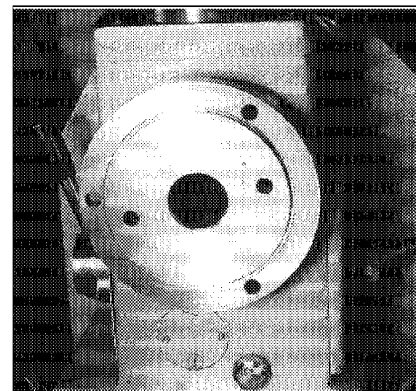
Standard lathe dogs drive round, or near-round, shapes. Rectangular or near-rectangular stock requires a clamp dog. In a properly made clamp dog, the underface of the heads of tightening screws are convex and fit into concave seats, while the holes in the upper bar are elongated. This design allows a firm grip of off-square shapes without bending the screws. Top and bottom bars should



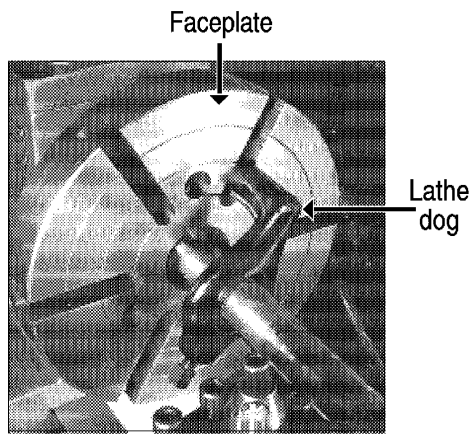
11.6 The correct depth of center is illustrated above. If it's too deep (bottom), only sharp outer edges will contact the center.



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



11.8 Bolt the faceplate to the spindle flange.



11.9 Fasten a lathe dog to one end of the workpiece.

also have V-notches to give a firm grip on triangular or other odd-shaped stock. You can use clamp dogs or special V-jaw dogs also to hold highly polished round bars.

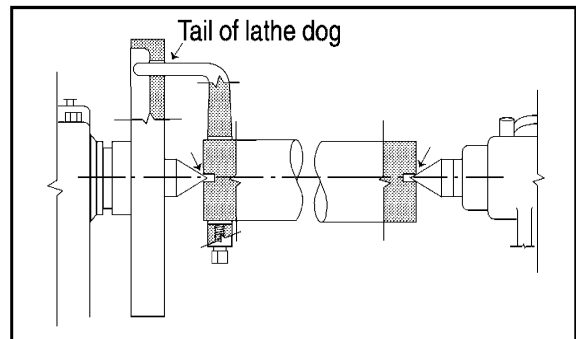
### 11.4 Using faceplates

For work setup, faceplates serve two purposes. First, they drive workpieces held between centers. Second, 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 11.9). 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. Workpieces mount on such faceplates with T-slot or standard bolts, strap clamps, angle plates, or other standard set-up tools.

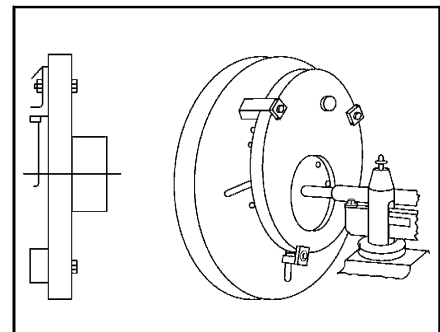
Note: Before starting to machine work set up on centers, check to see the lathe dog tail is free in the faceplate slot (Figure 11.10) so it won't lift stock off its true line of centers. Also, be sure lathe centers fit closely into the center holes to eliminate side play but not so tightly they bind. If you're working on a long workpiece, check it frequently to be sure the center does not bind. Also, balance unbalanced setups with counterweights to overcome any "throw" as the work revolves (Figure 11.11).



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

### 11.5 Setting up work on a mandrel

You can machine cylindrical or bored pipe work or cored castings too long to fit in a chuck by mounting them first on a mandrel (Figure 11.12). Then mount them between centers. 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, drive it back out of, instead of through, the hole.

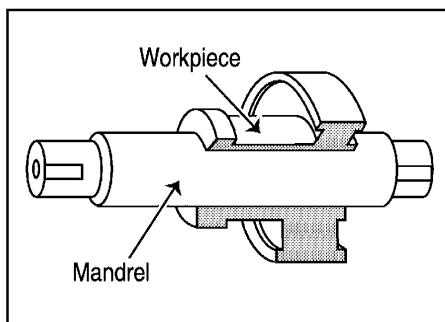


11.11 Counterweights can help with unbalanced setups.

You can purchase hardened steel mandrels, which have a slight (0.003") ground taper and an expanding collar, to facilitate mounting and demounting (Figure 11.13). Mandrels with compressible ends for holding single or ganged pieces are also available. When a workpiece is mounted on a mandrel, machine it as you would a solid shaft. You can drill eccentric centers in mandrel ends to permit eccentric turning.

### 11.6 Steady rests and follow rests

Rests are for setting up (1) work that is relatively long in proportion to its diameter or (2) work whose



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

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.

**Steady rests.** Steady rests mount on the lathe bed (*Figure 11.14*). 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.

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 not be true. Because the tips of the jaws are bronze, they 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 11.15*). Follow rests mount on the carriage of the lathe and move with the tool, backing up the workpiece opposite the point of the tool thrust. They have two adjustable supporting jaws, one holding the work to keep it from climbing up on the tool and the other behind the work to counter the thrust of the tool.

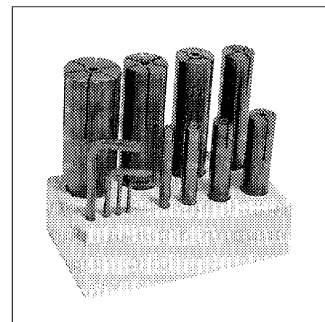
**Note:** Take great care in adjusting the jaws of rests, as they must form a true axial bearing for the work and let it turn freely but without play.

## 11.7 Setting up work in a chuck

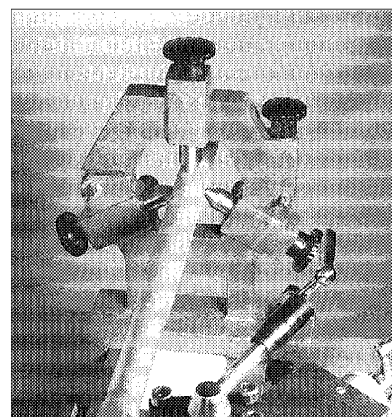
Chucks usually hold work that is too short to hold conveniently between centers or work requiring machining at, into (boring or inside threading), or across its end. While it is possible to set up such work on a faceplate, the convenience of chucks has made them part of every complete lathe.

Lathe chucks come in many types and sizes and hold workpieces of diameters approaching the swing of the lathe.

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 11.16*). 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. The three-jaw chuck has the advantage of being self-centering—all jaws move in or out together (*Figure 11.17*).



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



11.14 Steady rests mount on the lathe bed and provide three bearing surfaces.

## 11.8 Mounting work in a four-jaw independent lathe chuck

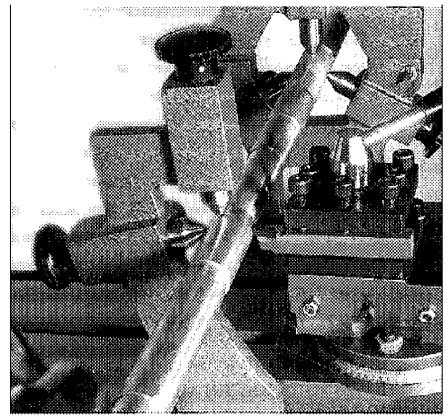
For small-diameter, short work, insert jaws in the chuck with high ends to the center. This gives the maximum gripping and tool clearance (*Figure 11.18*). For large-diameter work, insert the jaws in the chuck slots with the high steps of the jaws to the outside of the chuck (*Figure 11.19*).

To place work in a chuck, follow these steps:

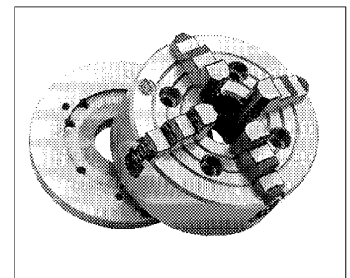
- Adjust the chuck jaws to the approximate opening to receive the work. Roughly center them by matching the nearest concentric ring on the chuck face with the corresponding mark on the jaws.
- Place the work in the chuck and grip it. Turn up the opposing jaws a uniform number of turns with the key. This will 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 11.6*).
- Guided by the chalk marks, readjust the jaws until a chalk line will carry completely around the work. Then tighten all the jaws securely.

For greater accuracy, after roughly centering the stock using chalk, set a dial indicator at the back of, and square to, the stock. Make sure you can see it clearly. Rotate the chuck by hand. Looking at two opposing jaws, determine which side is higher. Align the higher side with the dial indicator, loosen the opposite jaw, and tighten the higher jaw. Do the same with the other two jaws. Repeat the process until you have located the stock within necessary tolerances.

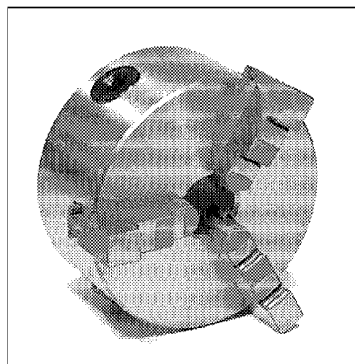
When making several identical pieces, after completing each workpiece release only two adjoining jaws, leaving the others to hold the center. The jaws of the four-jaw independent chuck are reversible. You can insert them with high steps to the inside or outside.



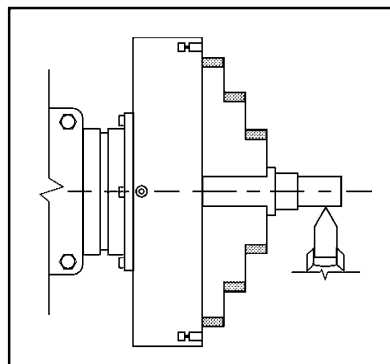
11.15 Follow rests mount on the lathe carriage and move with the tool.



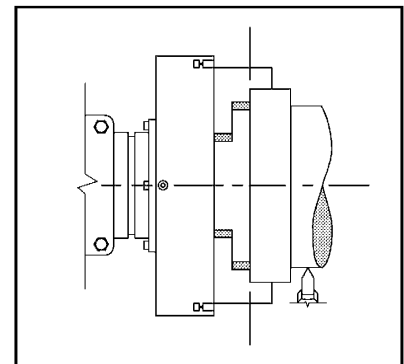
11.16 Four-jaw independent lathe chucks hold round, square, eccentric, or odd-shaped workpieces.



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



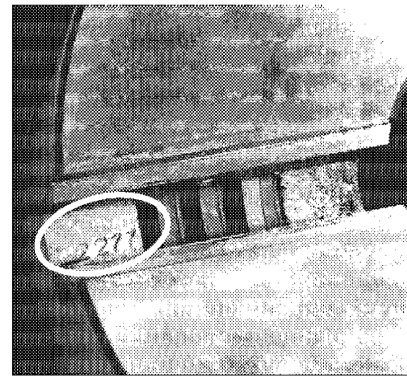
11.18 For short, small-diameter workpieces, insert the jaws with high ends to the center.



11.19 For large-diameter workpieces, insert the jaws with the high steps of the jaws to the 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 machine. 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.



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

### 11.9 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:

- On three-jaw chucks, the key moves all the jaws at once.
- You need not center or check for concentricity because these chucks 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.

Each jaw is stamped with a serial number and jaw number (#1, #2, or #3). The slots in the chuck are not numbered, but there is a serial number stamped at the #1 slot (*Figure 11.22*). With the #1 slot in the 12:00 position, the #2 slot is at 8:00 and the #3 slot at 4:00 (*Figure 11.23*).

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

### 11.10 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 11.24*) 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 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.



- They are housed within the spindle nose for maximum tool clearance, making it possible to machine, thread, or cut off close to the spindle.

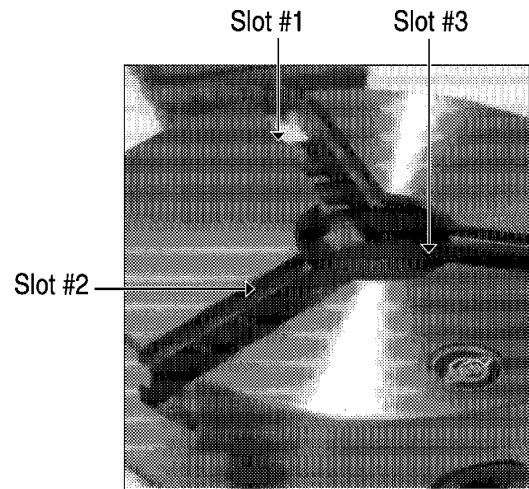
While chucks are universal tools that hold a range of stock sizes and shapes, collets are special tools. There is a collet for every size and shape of workpiece.

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 MT4 taper to the collet's MT3 taper. A drawbar holds the collet in place.

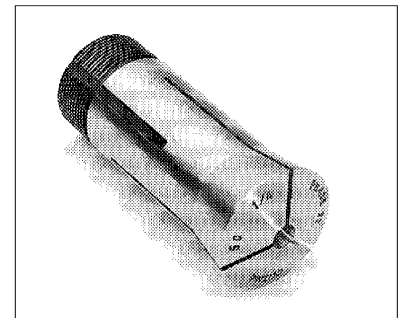
### 11.11 Toolpost grinders

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 tools.



11.21 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.



11.22 Collet attachments are best for small-diameter work.

## SECTION TWELVE LATHE TURNING

### 12.1 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 the chuck. Use a right-hand turning or roundnose cutter.

After selecting a cutter, place it into the left side of the turret (*Figure 12.1*). 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 (*Figure 10.1*).

With the tool properly positioned, tighten the Allen capscrews. Next, run the carriage to the right end of the workpiece with the hand crank. 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.

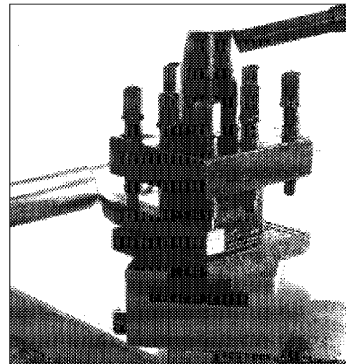
Say, for example, you need to reduce a diameter by a known number of thousandths of an inch. If you zero the collar and watch the movement of the dial, you'll know the depth of the feed from the zeroing point. Note: The dial gives a good approximation, but for exact measurements, use a measuring instrument.

To reduce the diameter, advance the tool only half as many thousandths on the dial. This is because the tool takes off 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 1-1/4 calibrations.

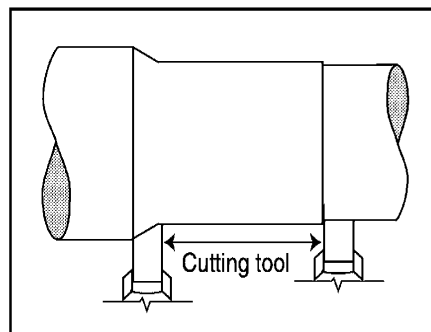
Engage the tool before setting the floating dial. The tool must be moving in the direction you want to go before you set the dial to zero to compensate for the backlash.

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 before setting the collar or when drawing the tool from the cut. Normal backlash is 0.008–0.015".

Engage the longitudinal feed by rotating the half-nut lever 90° to the right. Always cut deeply enough to reach below the scale on oxidized bars or iron castings. Hard, oxidized surfaces dull tools rapidly.



12.1 Place a cutter into the left side of the turret.



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

## 12.2 Finish turning

After you've rough-turned the workpiece to approximate finished size (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 12.2*, the left-hand tool illustrates the first roughing cut and the right-hand tool shows the following finishing cut.

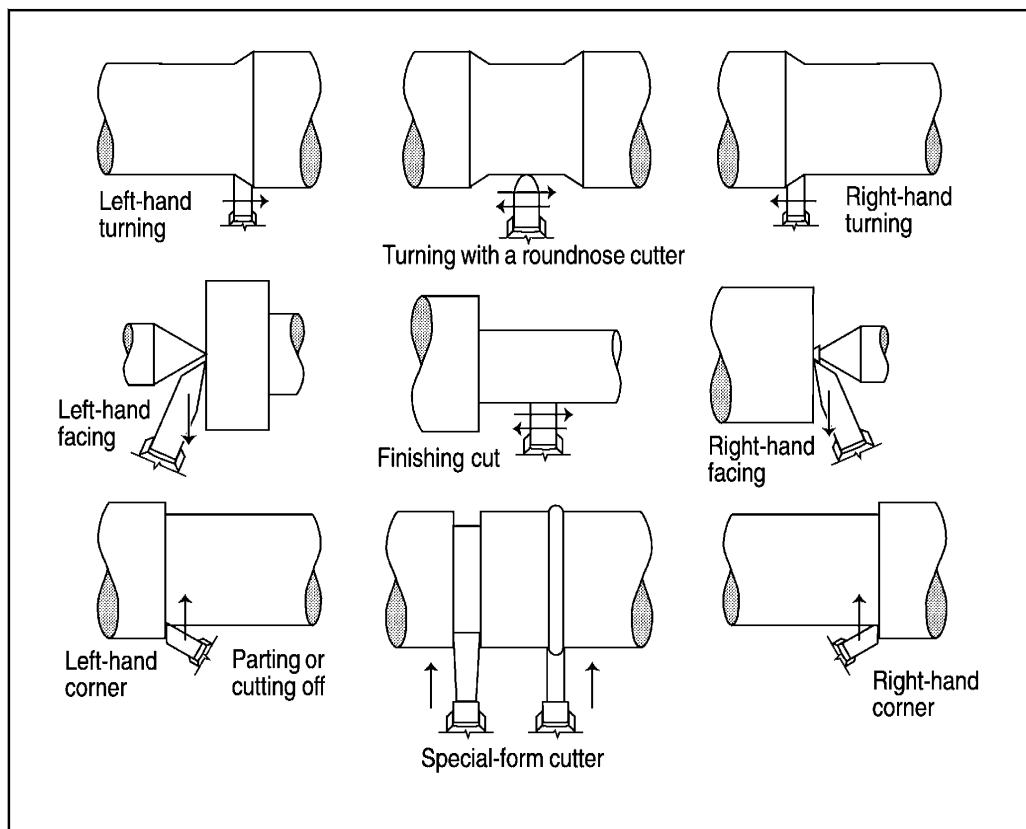
## 12.3 Turning to shapes

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 12.3*).

## 12.4 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.



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

- Using the longitudinal feed, turn a small diameter to finish up to the shoulder.

- With 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.

## 12.5 Finishing and polishing

After machining, you'll want a smooth, polished surface free of machine marks. You'll obtain the best results with a toolpost grinder. If you don't have one, use a file.

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 12.4*).

For an even finer file finish, rub 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 12.5*).

## 12.6 Taper turning

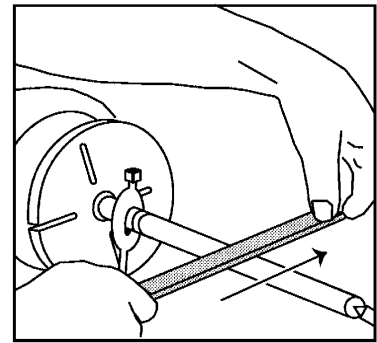
There are two ways to turn a taper: with the compound rest and by setting over the tailstock. In both methods, the cutter must engage the work on dead center if the taper is to be accurate.

**Compound rest.** Tapers cut with the compound rest are usually short, abrupt angles, such as centers, bevel gear blanks, and die parts (*Figure 12.6*). In general, these are not considered taper turning, which applies to machining longer, more gradual tapers.

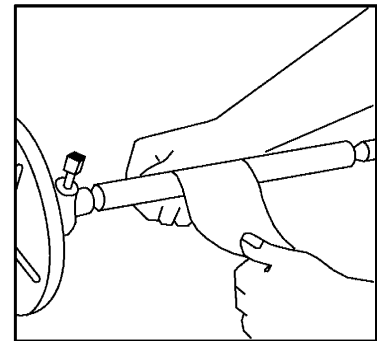
**Setting over the tailstock.** Cutting tapers by setting over the lathe tailstock involves misaligning the lathe centers. The lathe centers move from their position parallel to the tool's transverse travel, giving the desired degree of taper (*Figure 12.7*). The tailstock has a set-over scale calibrated both forward and backward from the straight turning or zeroing point for measuring set-over distances.

To offset the tailstock, adjust the two base-locking bolts (*Figure 5.8*). To offset to the right, loosen the right adjusting bolt and tighten the left. To offset to the left, loosen the left adjusting bolt and tighten the right.

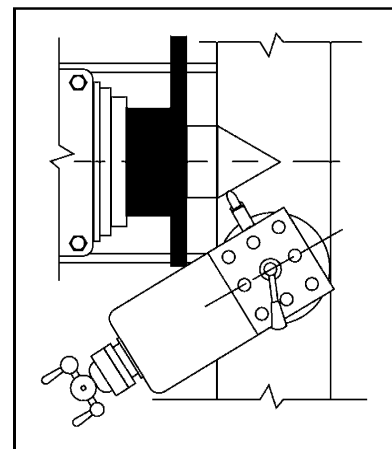
You can turn long, gradual tapers by setting over the tailstock, but take care. Your computations



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



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



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

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:

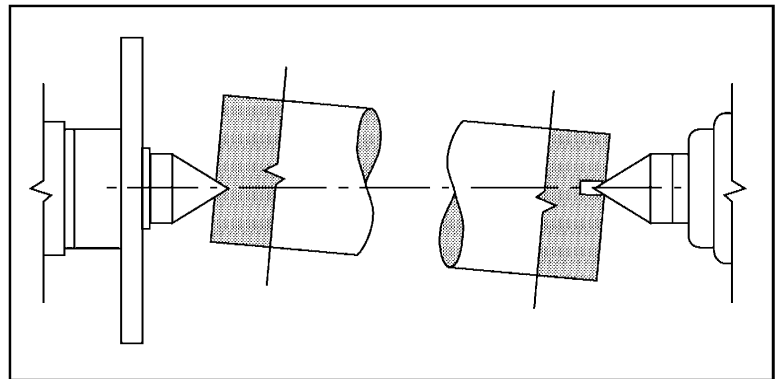
- The differential between the finished diameters of the extreme ends of the taper
- The length of the taper in relation to its extreme diameters, if the entire shaft is to be tapered
- The ratio between the length of the tapered portion to the entire length of the shaft (or work between centers when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

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 12.8*). When a taper extends only part of the length of the shaft, divide the total shaft length 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.

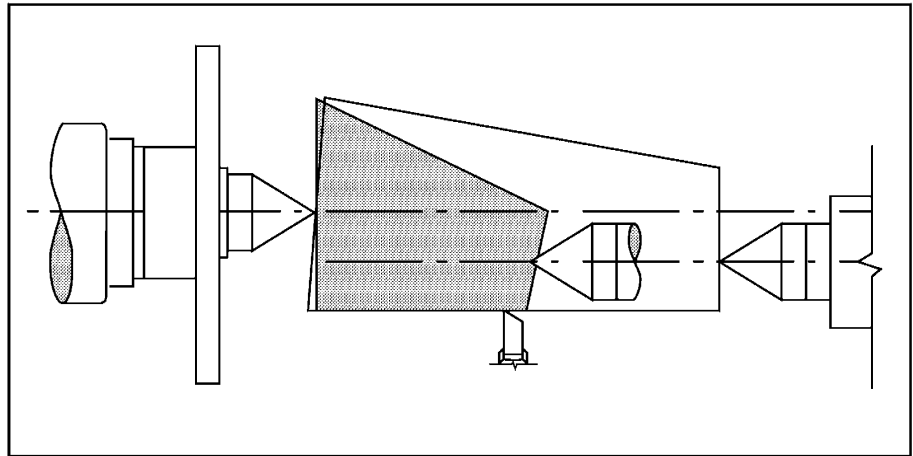
Notes: (A) Because most drawings give the taper in inches per foot of length, it may be easier to convert all dimensions to inches. (B) Be sure to zero the tailstock before resuming straight turning.

### 12.7 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. You can also use a taper attachment to bore a tapered hole.



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



12.8 Tailstock setover should be half the difference between the finished diameters of the ends. That is,  $O = T \times L / 2$ , where  $T$  = taper per inch and  $L$  = length of work in inches.

## SECTION THIRTEEN

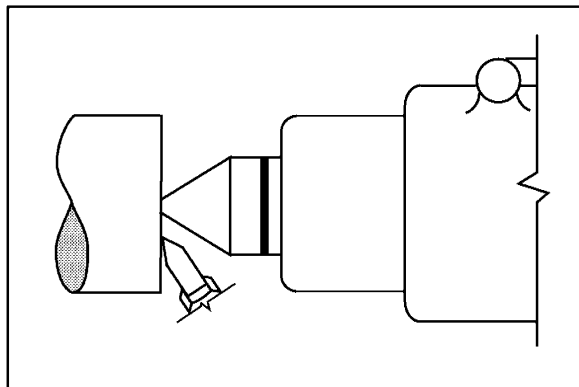
# LATHE FACING AND KNURLING

Before removing your work from the centers, face or square up the ends. On highly 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 13.1).

### 13.1 Facing across the chuck

When facing a stub-end workpiece held in the chuck, the same rules apply. Chuck the stock, letting it 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 and perhaps cause the center drill to run off center.



13.1 With a facing cutter, feed the tool from the center outward.

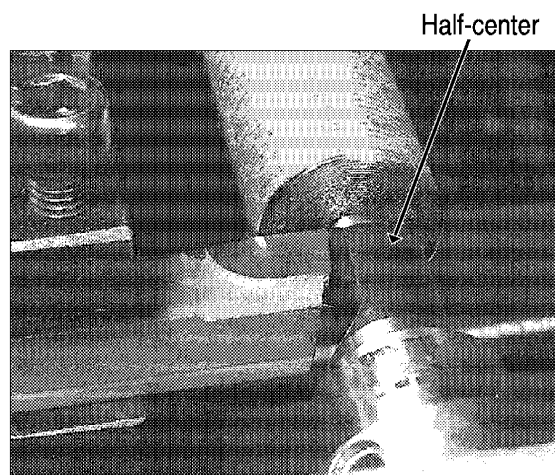
Start your lathe on the slowest speed. Bring the tool into cutting position against the center of the workpiece. Do not start with a heavy feed because the sfm increases rapidly as the cutter moves through increasing peripheries. One or two light cuts is usually enough to true up an end roughened by the hacksaw. After facing one end, reverse the workpiece and face the opposite end.

If you must finish the ends of the shaft, use a half-center (Figure 13.2). This lets you extend the tool across the entire face of the work.

### 13.2 Knurling

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

To make a true, uniform knurl, maintain equal 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. Operate the lathe at the slowest speed (160 rpm).

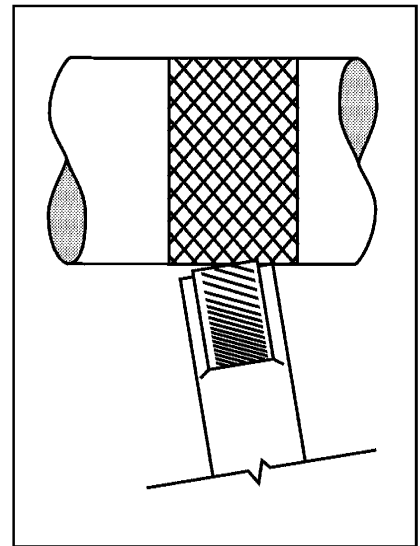


13.2 With a half-center you can extend the tool across the entire face of the work.

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 13.3*).

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  $1/64$ ", 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.



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

## SECTION FOURTEEN CHANGING GEARS

To change the gears on the CB-1220 XL, follow these steps. You will need a 10-mm wrench, 6-mm Allen wrench, screwdriver (to remove C clips), and pliers (to replace C clips).

- Remove all C clips, nuts, and gears, starting with the A gear and ending with the D gear. With the 10-mm wrench, loosen the B & C gearshaft in its bracket by turning the gearshaft counterclockwise (*Figure 14.1*). This lets the shaft 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 Allen wrench to loosen the bolt at the bottom of the bracket assembly for full swing and easy gear replacement (*Figure 14.1*).

- Place the selected D gear on the D shaft, flange side in. Replace the spacer, washer, and nut.

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

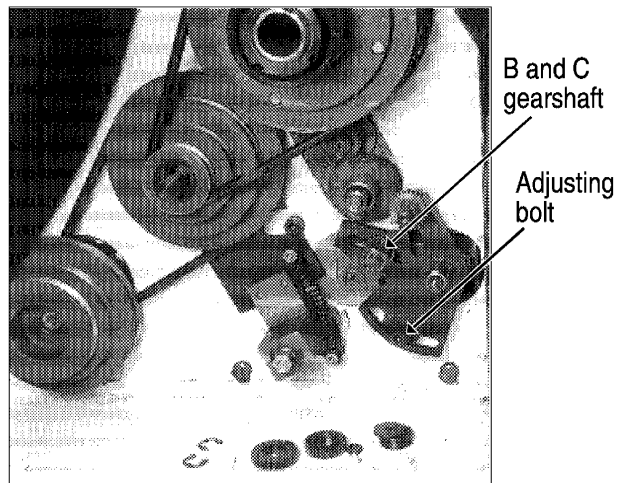
- Place the selected B gear, flange side in, on the B & C gearshaft. Replace the C clip.

- Slide the B & C gearshaft until the C gear meshes properly with the D gear and tighten it with the 10-mm wrench (*Figure 14.2*).

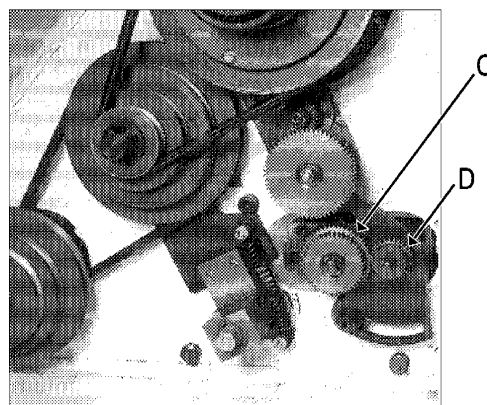
- Place the selected A gear, flange side in, on the A gearshaft and replace the C clip.

- Swing the bracket assembly until the A and B gears mesh. Hold the bracket assembly in place and tighten the bolt. Make sure the gears turn smoothly before engaging the powerfeed. You may need to make some adjustments.

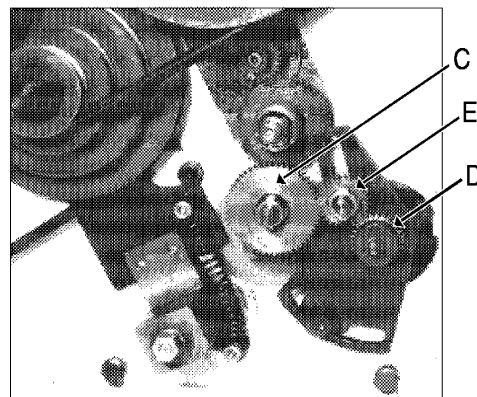
- Engage the E gear between the C and D gears to reverse the leadscrew (*Figure 14.3*).



14.1 Remove all C clips, nuts, and gears.



14.2 Slide the B and C gearshaft until the C gear meshes with the D gear.



14.3 Engage the E gear between the C and D gears to reverse the leadscrew.



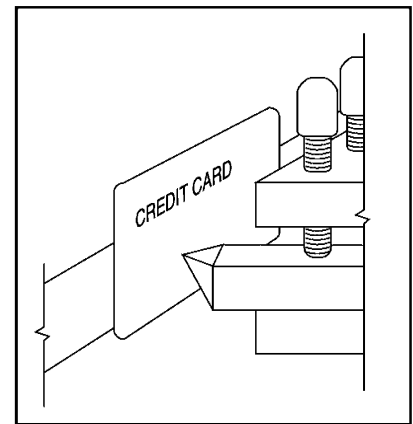
## SECTION FIFTEEN CUTTING SCREW THREADS

### 15.1 Threading terms

Before beginning to cut threads, it's useful to learn the major terms used in thread cutting:

- **Pitch.** Metric 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).

Because thread cutting is so much a part of machine work, anyone interested in building things of metal should master it. Threading requires a bit of patience and skill. Before trying to cut a thread on a workpiece, cut a few practice threads on odd bits of steel, iron, and aluminum.



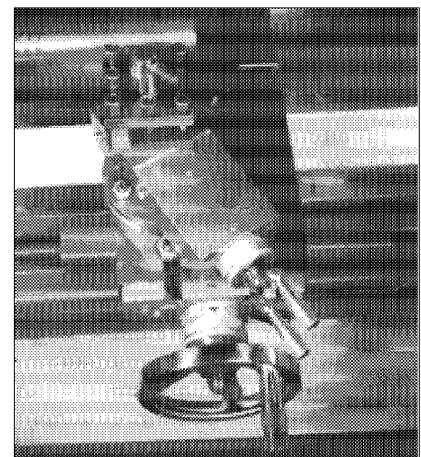
15.1 Check dead center with a credit card.

Built for thread cutting, the CB-1220 XL cuts standard internal and external threads, as well as special threads. You may cut coarse or fine threads in a great range 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.

Beginners will use the Unified National Standard most of the time, which is a V-form thread slightly flat on top and at the root. Screw threads are usually referred to by pitch numbers, such as 18 or 24, meaning 18 or 24 threads per inch (tpi). The CB-1220 XL cuts standard threads in pitches from 6 to 30 tpi and metric threads from 0.50 to 3 mm.

Because the lathe spindle turning 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 threads of various pitches by changing both the change gears at the head of the lathe and the speed selection lever.

Thread charts inside the gearbox door show both inch and metric measures (*Figure 7.2*). The inch chart on the headstock



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

shows the tpi from 6 to 30. The metric chart shows the distance from thread crest to crest from 0.12 to 3 mm.

For right-hand threads, start the threading tool at the right end of the workpiece and feed it toward the headstock. For left-hand threads, reverse the leadscrew's rotation direction and feed the threading tool from left to right.

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.

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 as a guide.

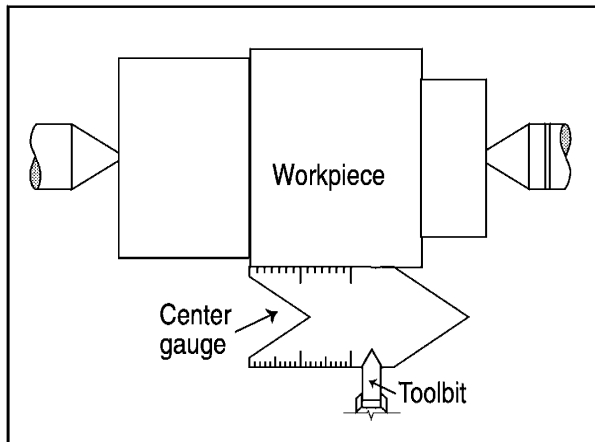
To make sure your cutter is on dead center, place a credit card or shim between the cutter point and workpiece (*Figure 15.1*). When the tool is on dead center, the card or shim will remain vertical. With a credit card, there is no possibility of chipping the cutter as the workpiece and cutter come together.

Set the compound perpendicular to the line of centers and rotate it  $29\text{-}1/2^\circ$  to the right (*Figure 15.2*). Place the thread gauge on the point of the threading tool and feed the tool toward the workpiece (*Figure 15.3*). 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.

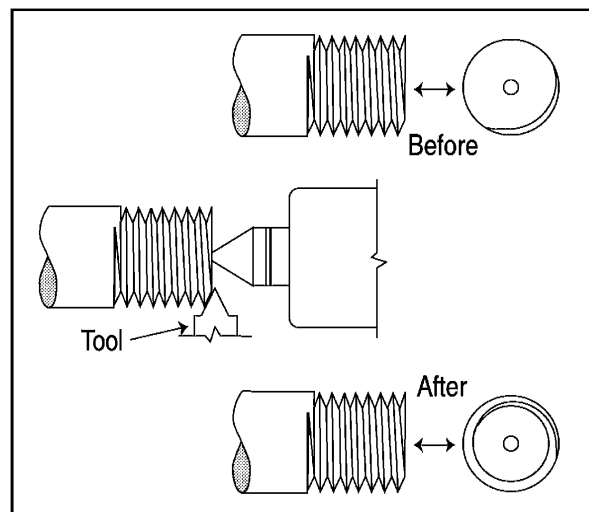
## 15.2 Cutting right-hand threads

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 by carefully turning it one-quarter turn to the right. Do not force it, and do not disengage it until you are completely done.

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.

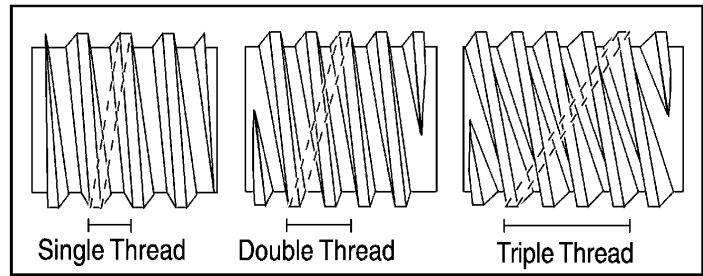


15.3 Using a center gauge, set the threading tool at exactly dead center on the workpiece.



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

It's time to take the real cut now, so apply the appropriate cutting fluid to the workpiece. Feed the cross slide 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. Zero the cross-feed calibration and make the second cut.



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

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 workpiece in the chuck and not remove it for testing.

After returning the workpiece to the setup, continue taking 0.00–0.002" cuts, checking the fit between cuts. When you thread the nut, it should go on easily but without end play. When you have the desired fit, chamfer the end of the thread (that is, take a 45° cut off the end) to protect it from damage, *Figure 15.4*.

### 15.3 Cutting left-hand threads

Cut left-hand threads exactly as you cut right-hand threads, except this time feed the carriage toward the tailstock instead of away from it. Reverse the cutter clearances and grind the cutters back with a clearance angle on the left side. Swing the compound rest to the left rather than to the right.

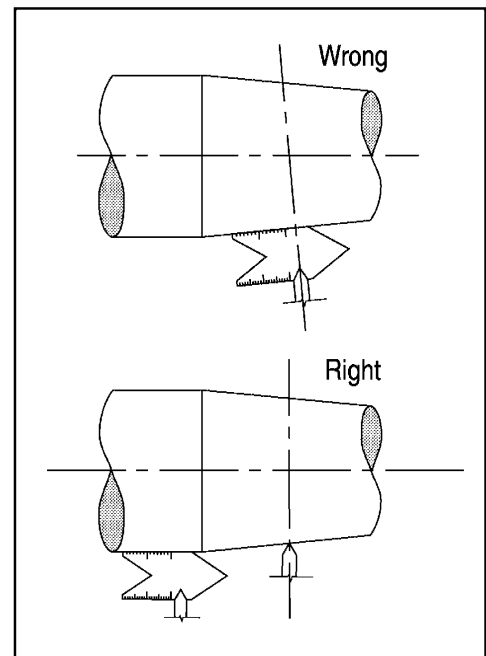
### 15.4 Cutting multiple threads

Cut multiple threads (*Figure 15.5*) 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.

### 15.5 What not to do when cutting threads

Do not disengage the half-nut lever. Do not shift the powerfeed speed lever. 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 finished and ready for testing, and only if absolutely necessary, remove the work from the center, leaving the lathe dog attached. Then test the



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

thread. If it does not fit properly and you have to remove another chip or two, place the workpiece back in the centers exactly where it had been. Then remove the chips and test again. Repeat until you are finished.

### **15.6 Finishing off a threaded end**

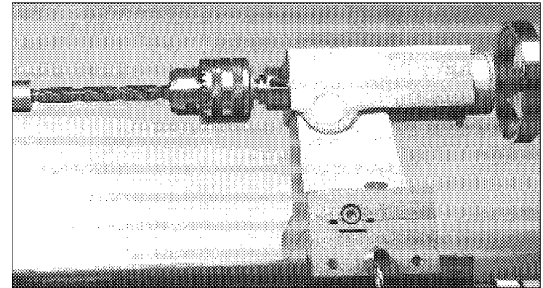
After cutting a thread and before removing the threading tool, chamfer the end. This improves its appearance and removes sharp corners and burrs. It also aids the screw as it engages a nut or threaded hole.

### **15.7 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  $90^\circ$  to the axis of the taper, rather than at  $90^\circ$  to its surface (*Figure 15.6*).

## SECTION SIXTEEN LATHE DRILLING AND BORING

You can lathe drill on the CB-1220 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 gives a straighter hole (*Figure 16.1*). Without changing setup and recentering, the work is ready for any succeeding operations, such as boring and internal threading.

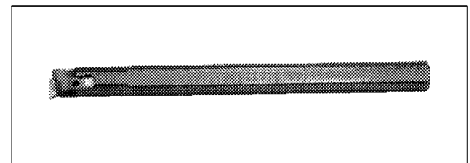


16.1 Holding the drill stationary in the tailstock chuck gives a straighter hole.

In all lathe drilling operations, keep the drill sharp and properly ground. This is essential for obtaining 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 greater the number of 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. Do this by turning the tailstock handwheel. Make a locating center for the drill point, or even a counter-sunk center for large diameters, to keep the drill from creeping.



16.2 A tool with a smooth-ended bar won’t mar the workpiece.

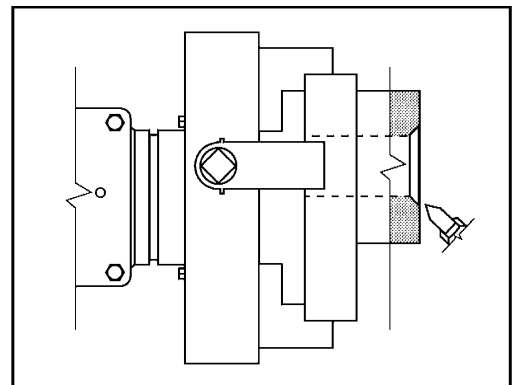
### 16.1 Reaming

When a hole must be accurate to 0.002” or less, drill it slightly undersized (0.010” to 1/64” on small diameters and 1/64” to 1/32” on holes 1” to 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. Use slow speeds and feed the reamer slowly and evenly into the workpiece. Be sure the reamer teeth are free of burrs and chips.

### 16.2 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.



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

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 technician about the Smithy bar, #K99-120).

It is also wise to select tools with smooth-ended bars without a projecting nut or hardened edge that might mar the work (*Figure 16.2*). 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.

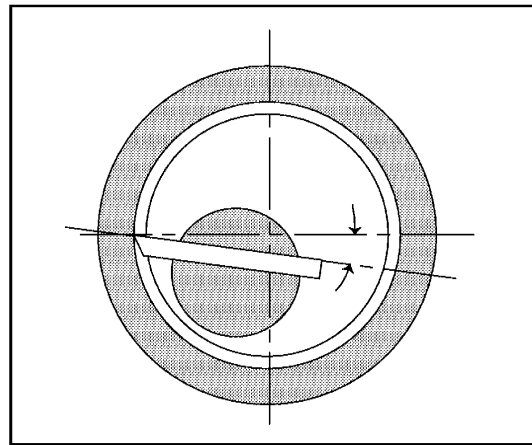
Except with cored castings, pipes, or tubing, begin by drilling a hole large enough to admit the end of the boring bar. Because the holes in cored castings often deflect boring bars from their true axis, you may want to chamfer or turn a starting cut in the opening of the hole to be bored with a turning tool (*Figure 16.3*) before introducing the boring tool.

With the boring toolholder set up (in the toolpost or toolpost T-slot, depending on the type), 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 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  $14^\circ$  from the radius at the cutting point (*Figure 16.4*). This takes into consideration the ground angle (top rake) of the cutter itself.

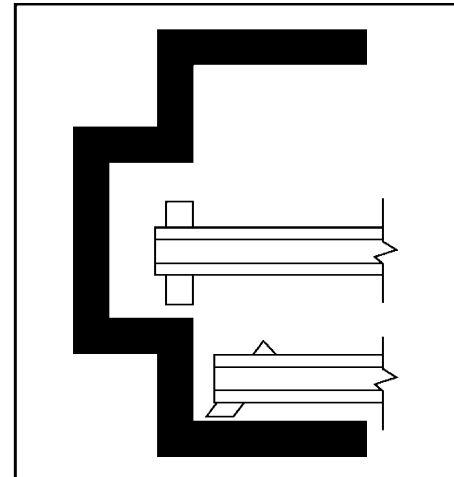
For straight longitudinal cuts, you can hold the cutter closer, and thus more rigidly, if it's at a  $90^\circ$  angle to the bar. For machining bar ends, however, you need a boring bar that holds the cutter at an angle or angles so it extends beyond the end of the bar (*Figure 16.5*). For maximum visibility, position the cutting edge at the near side, parallel to the centerline.

The rules that apply to external turning apply to boring as well, except—as noted earlier—where the rake angles differ. The rake angles are governed by cutter type and bore diameter. Feeds must be lighter to keep the tool from springing. This is especially true when enlarging out-of-round holes, when you take several small cuts rather than one heavy cut.

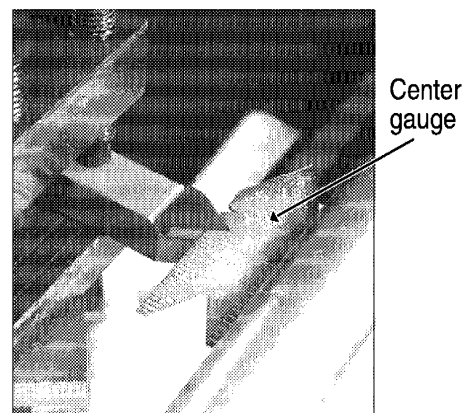
After the last finish cut, it is common to reverse the feed and take one last, fine cut with the tool coming out of the work. This



16.4 The cutting edge engages the workpiece along a line in the mounted plane of the lathe centers.



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



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

last cut, taken without movement of the crossfeed, avoids a slightly undersized hole because you compensate for any spring in the bar.

### 16.3 Cutting internal threads

Internal thread cutting is like external thread cutting, except 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.

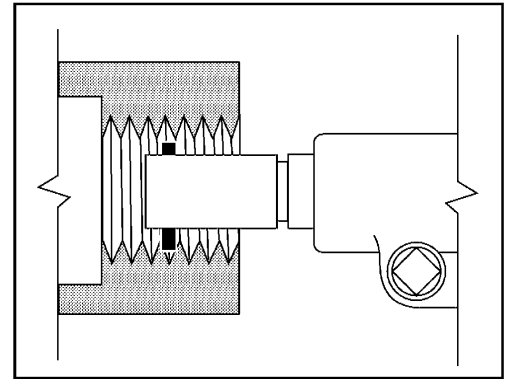
Another difference between boring and inside threading is the cutting angle at which the cutter approaches the workpiece. As with external thread cutting, the internal threading tool must engage the work on dead center and be held so the cutter coincides with the workpiece's center radius.

In squaring the cutter with the work, use a center gauge (Figure 16.6) or thread gauge. Internal cutters require greater end and side clearance, and cutter length is also restricted because internal thread cutters must have enough end clearance that the cutter lifts clear of the thread for removal (Figure 16.7). Before cutting an internal thread, bore the workpiece to the exact inside diameter.

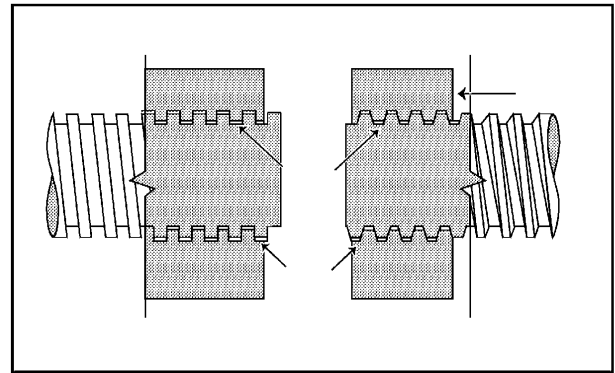
Because the feed of successive cuts is toward, not away from, the operator, the thread-cutting set reverses. Also, you must take lighter cuts because of the cutter's extension from the toolpost. Take an extra finishing cut without changing the setting of the compound rest.

### 16.4 Cutting special-form internal threads

You can cut internal forms in all the 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 screw recommended for different thread types (Figure 16.8). If you don't have recommended clearances, it is safe to cut a nut (internal) thread 0.005" to 0.010" per inch larger in the screw's outside diameter.



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



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

## SECTION SEVENTEEN

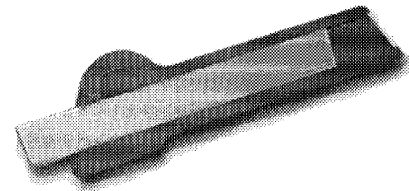
# CUTTING OFF OR PARTING WITH A LATHE

You can cut off in a lathe only when holding one end of the work rigidly, as in a chuck. It is not practical for long workpieces 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. Cutting off also requires a narrow cutting edge with ample (5–10°) side clearance, which should feed into the work slowly to prevent hogging in. Once considered a difficult, costly operation, cutting off became much simpler with development of narrow tools with special cut-off blades (*Figure 17.1*).

The toolpost should hold the cut-off tool as close to the workpiece as possible, with the top of the blade on dead center and exactly perpendicular to the line of centers. Extend the blade only far enough to pass through the workpiece, just beyond its center. The tool should feed to the workpiece on exact center, slowly and evenly with the crossfeed. 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 as possible to the headstock. If you must make a parting cut on a long shaft or on work between centers, don't complete the cut in the lathe. Finish the parting with a hacksaw and return it to the lathe for facing. Slow the spindle speed until you have a good feel for cutting off. Although lubricants and coolants are not essential on small-diameter workpieces, use them amply on deep cut-off work.

Smithy offers a line of cut-off tools for light, medium, and heavy-duty work. Contact a sales technician for details.



17.1 Specially designed tools like this one make cutting off easier.



## SECTION EIGHTEEN

# 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 makes the mill as important a tool as the lathe.

The millhead (*Figure 5.1*) rotates horizontally and moves vertically. A quill that moves in and out of the head carries the spindle.

You can move the table horizontally in two directions by turning the cross-slide and leadscrew handwheels (*Figure 5.1*). The cross-slide handwheel turns the table longitudinally (at right angles to the spindle axis); the long-feed hand crank moves it transversely (parallel to the spindle axis).

To raise and lower the millhead, release the lock lever and put the handle into one of the three holes in the black adjusting collar. To raise the millhead, turn it counterclockwise; to lower it, turn it clockwise.

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

## SECTION NINETEEN 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 19.1*). 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.

### 19.1 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.

### 19.2 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). Tilting and universal 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.

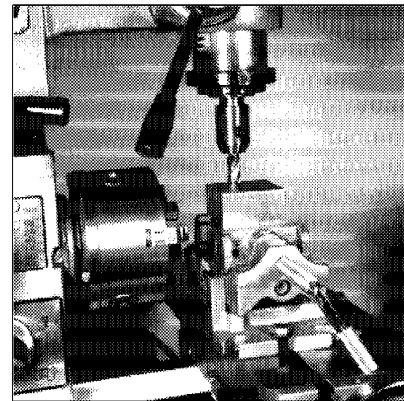
Plain vises have a flanged base with slots that lets them bolt to the table with the jaw faces either parallel to, or at 90° to, the longitudinal table travel. Swivel vises have a swivel base that bolts to the table. They're marked with degree graduations that let you position their jaws at any angle 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.

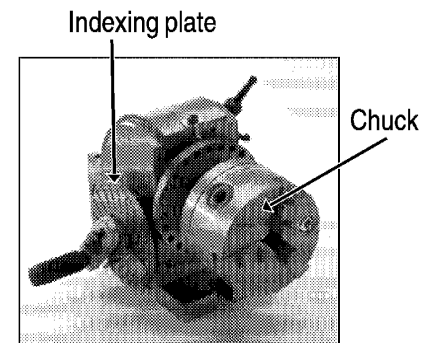
### 19.3 Using a dividing head

Also called indexing heads, dividing heads attach to the table to hold workpieces between centers for machining surfaces, grooves, or gear teeth at precise distances apart.

The main parts of a dividing head are its head and tailstock. The tailstock holds the outer end of the workpiece. The head is more complex. When you turn its handle, a spindle rotates through a precisely machined gearing system. A chuck can attach to the spindle face, which is set at 90° to the handle (*Figure 19.2*). An indexing plate is set in from the handle. By counting how many turns of



19.1 Setting up in a vise is one of the most common ways to secure your workpiece.



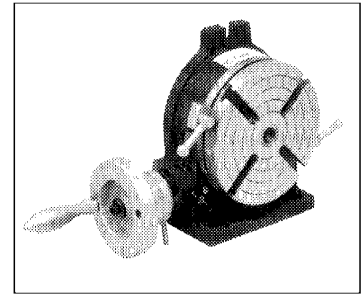
19.2 A chuck attaches to the spindle face of a dividing head.

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.

#### 19.4 Using a rotary table

A rotary table (*Figure 19.3*) 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. Contact a Smithy technician for more information.



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

## SECTION TWENTY

# HOLDING MILLING CUTTERS

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

### 20.1 Arbors

Arbors come in different sizes and lengths, with one end tapered to fit the bore in the end of the machine spindle. The CB-1220 XL arbor, which has an MT3 taper, is driven by the friction between the it and the 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 20.1*).

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.

### 20.2 Collets and holders

Straight-shank end mills fit into spring collets (*Figure 20.2*) or end mill holders (*Figure 20.3*). Their precision-ground shanks go into the mill spindle. When you tighten a spring collet, its hole reduces and the collet grips the end mill shank evenly. 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.

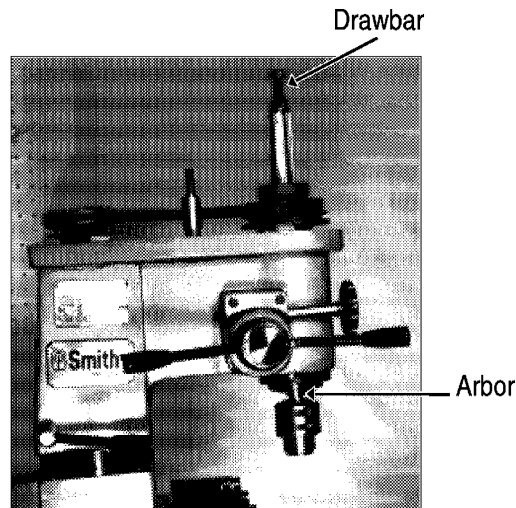
### 20.3 Adapters

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. Shell end mill adapters come in different sizes to accept different-sized shell end mills.

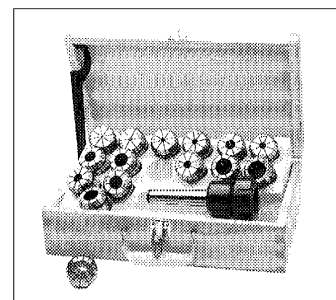
To remove arbors or adapters held with a drawbar, follow these steps:

- 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.

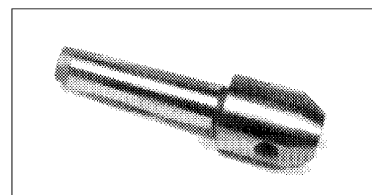
Your machine includes a tapered drift for removing tapers. Follow



20.1 The arbor stays in place via a drawbar.



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



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

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.

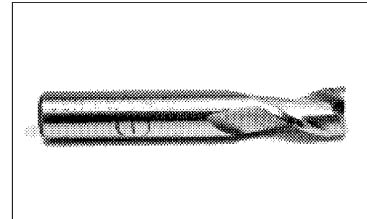
Cutters mounted in the spindle must fit accurately. There are two ways to make sure they do. For small cutters, fit the shank of the arbor that carries the cutter directly into the taper hole at the end of the spindle. A drawbar holds the arbor in place. For large cutters, bolt the cutter directly to the end of the spindle.

## SECTION TWENTY-ONE 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.



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

### 21.1 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.

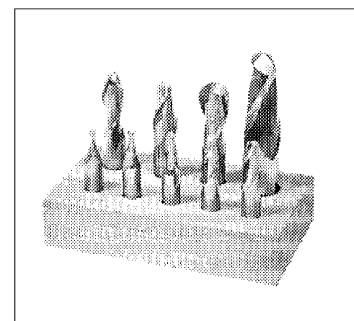
End mills machine horizontal, vertical, angular, or irregular surfaces in making slots, keyways, pockets, shoulders, and flat surfaces.

Two-flute, or center-cutting, end mills (*Figure 21.1*) 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-ended) or both ends (double-ended).

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 noncenter-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, dovetail end mills, T-slot cutters, keyseat cutters, and shell end mills.

Ball end mills (*Figure 21.2*) cut slots or fillets with a radius bottom, round out pockets and bottoms of holes, and perform diesinking and diemaking. Four-fluted ball end mills with center cutting lips are on the market.



21.2 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.

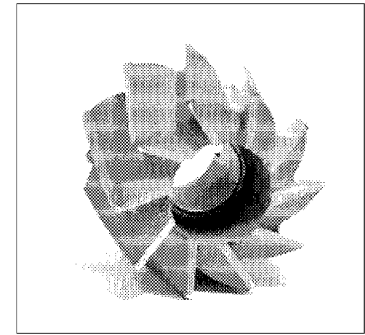
T-slot cutters cut T-slots. After machining a groove for the narrow part of the T-slot with an end or side mill, finish up with the T-slot cutter.

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

Shell end mills (*Figure 21.3*), 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 are as large as 6".

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" 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.



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

## 21.2 Plain cutters

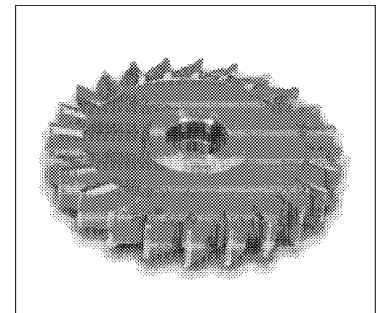
Plain cutters have teeth only on their periphery. Used to mill plain, flat surfaces, they may combine with other cutters to produce various shapes. They are cylindrical and come in many widths and diameters.

- 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° to 45°.
- Helical plain 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.

## 21.3 Side cutters

Similar to plain cutters, side cutters also have teeth on one or both sides (*Figure 21.4*). 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.

- Plain side 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 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 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.

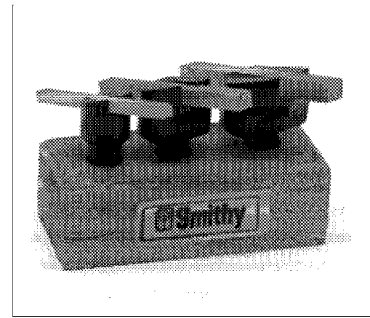


21.4 Side milling cutters are similar to plain milling cutter, but they have teeth on one or both sides.

## 21.4 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" and wider cuts and may do deeper cuts with standard feeds.
- 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.

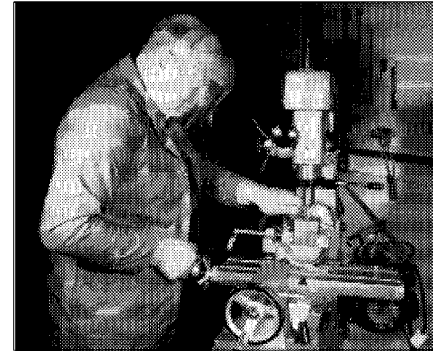


21.5 Flycutters take light face cuts from large surface areas.

## 21.5 Angle cutters

Angle 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°.



21.6 When using compressed air, wear a face mask and protective clothing.

## 21.6 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.

## 21.7 Flycutters

With one or more single-point toolbits or cutters, flycutters (*Figure 21.5*) perform 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 9).

You can also use flycutters for boring. Note: When the tool revolves, the cutting tool becomes almost invisible, so be careful.



## 21.8 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. With cast iron and steel, however, they tend to reduce the life of carbide tools, leaving tiny cracks along the cutting edge. Follow the advice of tool manufacturers to avoid tool failure. 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 a face mask and protective clothing (*Figure 21.6*), and be careful to keep cast-iron dust from getting between the lathe and carriage ways.

## 21.9 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.

You sharpen others 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-TWO

# SPEEDS AND FEEDS FOR MILLING

### 22.1 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 22.1*). 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 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:

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

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:

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

where CS = cutting speed, mpm = meters per minute,  $D (\text{mm})$  = 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 6.12*.

### 22.2 Feeds

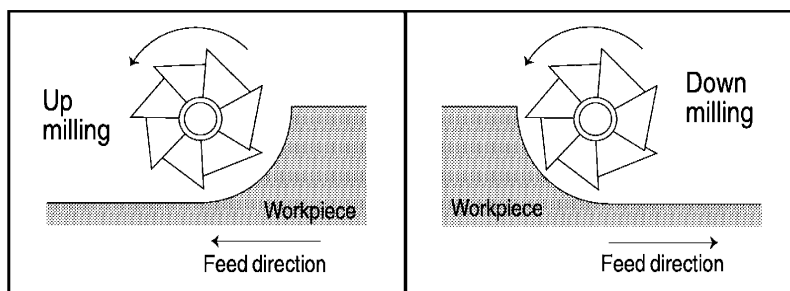
Set the direction of feed before you begin milling. Up milling, or conventional milling, is when the direction of feed is opposite to the direction of cutter rotation (*Figure 22.2*). Down milling, or climb milling, is when the direction of feed is the same as the direction of cutter rotation).

**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 the best way to mill.

**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

22.1 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
	1095	175–225	60–110	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
	H20–H43	200–250	40–85	175–300
	D1–D7	200–250	30–60	100–200
Stainless steel	302	135–185	70–100	225–350
	430F	135–185	100–140	350–450
Gray cast iron	ASTM class 20 Through scale Under scale	110–160	140–200	350–700
			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



22.2 In up milling (left), the workpiece feeds into the cutter in the opposite direction of the cutter's revolutions. In down milling (right), the workpiece feeds into the cutter in the same direction as the cutter is turning.

(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 this formula to calculate table feed rate in inches per minute (ipm):

$$\text{ipm} = \text{ipt} \times N \times \text{rpm}$$

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

## SECTION TWENTY-THREE COMMON MILLING OPERATIONS

### 23.1 Milling flat surfaces

One way to mill a flat surface is by plane milling (*Figure 23.1*). 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 23.2*) face-mill large surfaces.

Bevels and chamfers are cut at an angle to the main workpiece surface. A bevel cut (*Figure 23.3*) goes from side to side, completely removing the perpendicular edge. A chamfer removes only part of the perpendicular edge.

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 fixture held in a vise.

### 23.2 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.

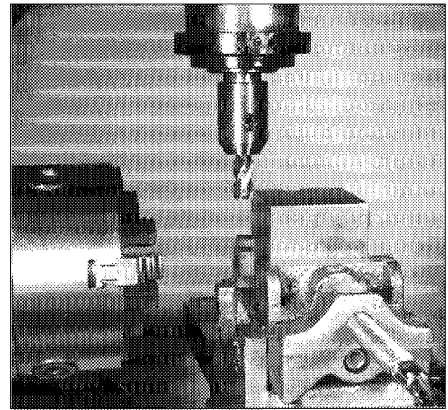
Plunge cutting is efficient for removing material quickly on low horsepower. Plunge the end mill a predetermined width and depth, retract it, then advance and plunge it again repeatedly. The maximum cutting force is in the machine's strongest (axial) direction.

### 23.3 Milling a cavity

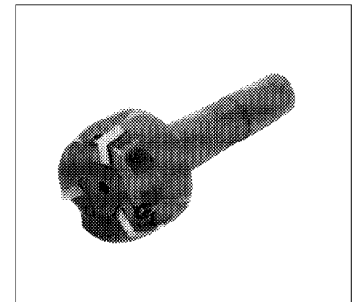
After laying out the outline of the cavity to cut, rough it out to within 0.030" of the finished size before making finish cuts. Use a center-cutting end mill for the starting hole.

### 23.4 Tapping

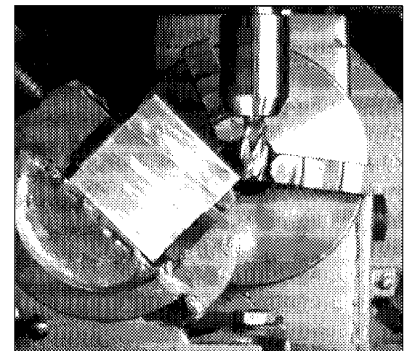
Drill a hole. Then remove the drill bit and put a tap into the chuck. By turning the chuck slowly by hand with slight downward pressure, you can get a perfectly threaded hole.



23.1 One way to mill a flat surface is by plane milling.



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



23.3 A bevel cut goes from side to side, completely removing the perpendicular edge.

## SECTION TWENTY-FOUR COMMON ADJUSTMENTS

### 24.1 Bevel gear adjustment

Occasionally you will have to adjust the bevel gears that drive the millhead. The adjustment is on top of the millhead support column (*Figure 5.1*). Using the sight hole on the back of the lathe head (*Figure 5.5*) will make the job easier. Remove the 20-mm plug on the back of the lathe head to access the sight hole.

There are two ways to reach the adjustment screws on top of the support column. Either unlock the millhead and lift it up off the machine or remove the four bolts under the drive pulley (*Figure 5.7*). Then lift the pulley and bearing seat from the millhead (*Figure 6.2*). There are six screws on top of the support column (*Figure 24.1*). To raise the bevel gear, loosen the three screws with heads and tighten the three without heads. To lower the bevel gear, loosen the three screws without heads and tighten the three with heads. When you are done, make sure all six screws are tight.

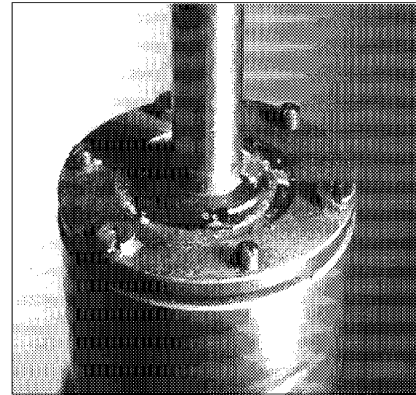
To replace the bevel gear, remove the four socket-head capscrews below the black adjusting collar (*Figure 6.2*) and remove the millhead and support column from the lathe head. Remove the screw at the bottom of the bevel gear (*Figure 24.2*) and pull the gear from the shaft. Then reassemble it with the new gear. Before replacing the millhead, adjust the bevel gear so it won't break as you tighten the millhead onto the lathe head. Then adjust the bevel gear.

### 24.2 Millhead T-key

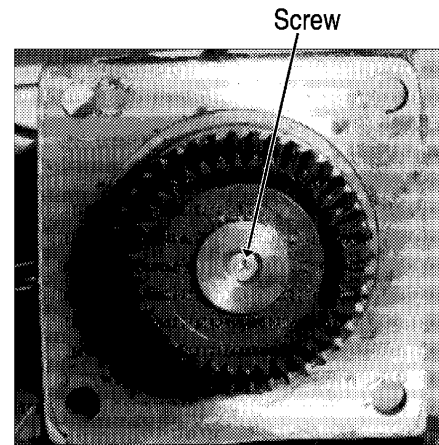
The millhead has a T-key between the drive shaft and drive pulley. If you raise the millhead more than usual, this key may come partly or completely out. If it is out only partially, raise the millhead slightly. Then reach down inside the drive pulley with a screwdriver and push it back in. Align the key with the keyway in the shaft and lower the millhead into place.

If the key has fallen out, it is probably sitting on top of the column. Raise the millhead completely off of the machine and reinstall the key into the keyway on the pulley. Then replace the millhead.

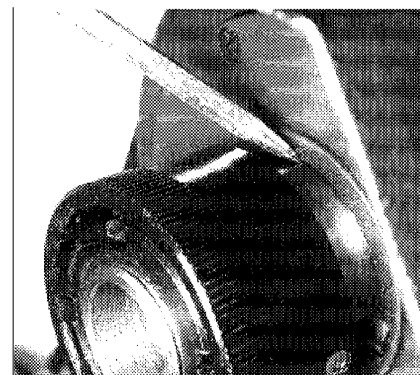
Repositioning the T-key means inserting the shaft into a blind hole. If you put some heavy grease or Cosmoline on the back of the key, it will stay in place when you insert it. If you use your finger to put it into place, put a bit of the grease on your finger.



24.1 Use the six screws on top of the millhead support column to adjust the bevel gear.



24.2 Remove the screw at the bottom of the bevel gear.

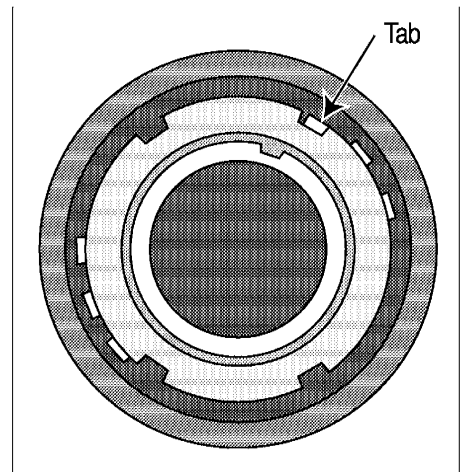


24.3 Scribe a mark on the jaw clutch at the base of the bearing cap.

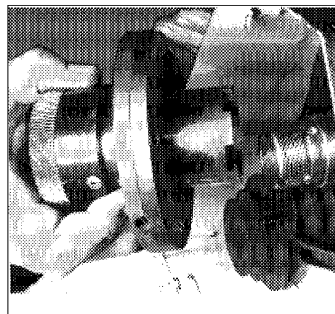
### 24.3 Lathe / Mill clutch

If the lathe clutch drops into neutral while turning, there will be too much gap between the jaws of the clutch. You must first determine how much gap there is. A spring-loaded steel ball in a detent in the jaw clutch holds the clutch in place. Pull on the clutch sleeve until you feel the ball drop into the detent and scribe a mark on the jaw clutch at the base of the bearing cap (*Figure 24.3*). Then pull the clutch sleeve out as far as possible and scribe another mark at the same place. The distance between these two marks is the amount of gap between the jaws.

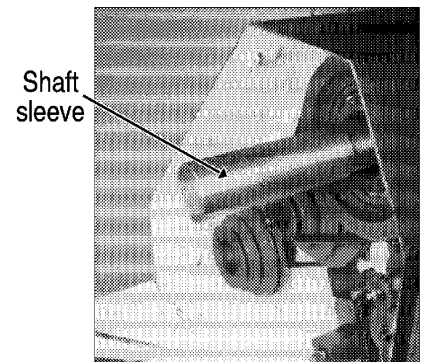
Bend back the tab on the lockwasher (*Figure 24.4*) and remove the spanner nut and lockwasher from the spindle. Remove the four screws from the bearing cap and pull on the clutch sleeve to remove the left jaw clutch and entire clutch assembly from the lathe spindle (*Figure 24.5*). Set it aside. Remove the key from the spindle, taking note of which keyway it is in. Remove the shaft sleeve from the spindle (*Figure 24.6*), shorten the sleeve by the amount of gap, and reassemble the entire unit. The left jaw clutch will now slide on further, eliminating excess gap between the jaws.



24.4 Bend back the tab on the lockwasher and remove the spanner nut and lock washer.



24.5 Pull on the clutch sleeve to remove the left jaw clutch and entire clutch assembly from the lathe spindle.



24.6 Remove the shaft sleeve from the spindle.

## SECTION TWENTY-FIVE TROUBLESHOOTING

### 25.1 Powerfeed and thread cutting

Powerfeed does not move carriage

Cause	Solution
<ul style="list-style-type: none"> <li>• Carriage locked</li> <li>• Speed selector not engaged</li> <li>• Sheared pin</li> <li>• Gears not meshing or teeth missing</li> <li>• Half-nut not fully engaged</li> </ul>	<ul style="list-style-type: none"> <li>Unlock carriage</li> <li>Select speed I or II</li> <li>Replace pin</li> <li>Check gears and mesh</li> <li>Keep half-nut engaged</li> </ul>

Cut is not smooth

Cause	Solution
<ul style="list-style-type: none"> <li>• Tool dull</li> <li>• Tool not on center</li> <li>• Tools not mounted tightly in post</li> <li>• Cross-slide gibs to bed and base loose</li> <li>• Gibs in toolpost loose</li> <li>• Tool turret not tight</li> <li>• Feed rate too fast</li> <li>• Gears loose</li> </ul>	<ul style="list-style-type: none"> <li>Sharpen or replace tool</li> <li>Center tool (shim, if needed)</li> <li>Remount tools</li> <li>Adjust gibs</li> <li>Adjust gibs in toolpost</li> <li>Tighten toolpost</li> <li>Install correct gears</li> <li>Tighten gears and posts</li> </ul>

Thread is not smooth

Cause	Solution
<ul style="list-style-type: none"> <li>• Tool dull</li> <li>• Tool not centered</li> <li>• Tools not mounted tight in post</li> <li>• Cross-slide gibs to bed and base loose</li> <li>• Gibs in compound loose</li> <li>• Tool turret not tight</li> <li>• Gears loose</li> </ul>	<ul style="list-style-type: none"> <li>Sharpen tool</li> <li>Center tool</li> <li>Remount tools</li> <li>Adjust gibs</li> <li>Adjust gibs</li> <li>Tighten toolpost</li> <li>Tighten gears and posts</li> </ul>

Tool is not cutting "on thread"

Cause	Solution
<ul style="list-style-type: none"> <li>• Half-nut not fully engaged</li> </ul>	<ul style="list-style-type: none"> <li>Keep half-nut engaged</li> </ul>

### 25.2 Carriage and milling table

Table won't move



Cause	Solution
<ul style="list-style-type: none"> <li>• Table locks engaged</li> <li>• Gibs too tight</li> <li>• Half-nut not engaged</li> </ul>	<ul style="list-style-type: none"> <li>Loosen locks</li> <li>Loosen gibs</li> <li>Keep half-nut engaged</li> </ul>

Horizontal movement in cross-slide table

Cause	Solution
<ul style="list-style-type: none"> <li>• Carriage gib improperly adjusted</li> <li>• Table gib improperly adjusted</li> </ul>	<ul style="list-style-type: none"> <li>Adjust carriage gib</li> <li>Adjust table gib</li> </ul>

Vertical movement in cross-slide table

Cause	Solution
<ul style="list-style-type: none"> <li>• Carriage gib improperly adjusted</li> <li>• Table gib improperly adjusted</li> </ul>	<ul style="list-style-type: none"> <li>Adjust carriage gib</li> <li>Adjust table gib</li> </ul>

Carriage moves smoothly in only one direction

Cause	Solution
<ul style="list-style-type: none"> <li>• Debris on way or gib</li> <li>• Burr on gib</li> <li>• Gib improperly tensioned</li> <li>• One or more wipers mounted too low</li> </ul>	<ul style="list-style-type: none"> <li>Remove debris</li> <li>Remove burr with fine file</li> <li>Loosen gib and re-tension</li> <li>Reposition wiper(s)</li> </ul>

Cross-slide handwheel turns during cutting operations

Cause	Solution
<ul style="list-style-type: none"> <li>• Cross-slide brass nut worn</li> <li>• Carriage locks not tight</li> <li>• Gibs too loose</li> </ul>	<ul style="list-style-type: none"> <li>Tighten or replace brass nut</li> <li>Tighten carriage locks</li> <li>Readjust gibs</li> </ul>

Too much backlash in the cross slide

Cause	Solution
<ul style="list-style-type: none"> <li>• Loose screw</li> </ul>	<ul style="list-style-type: none"> <li>Tighten screw, review how to eliminate backlash</li> </ul>
<ul style="list-style-type: none"> <li>• Loose brass nut</li> </ul>	<ul style="list-style-type: none"> <li>Put a shim between the stud on the nut and the side of the hole</li> </ul>
<ul style="list-style-type: none"> <li>• Worn brass nut</li> </ul>	<ul style="list-style-type: none"> <li>Replace brass nut or adjust screw at end of nut</li> </ul>
<ul style="list-style-type: none"> <li>• Too much space between bearing and dial</li> </ul>	<ul style="list-style-type: none"> <li>Add shim washers</li> </ul>

## 25.3 Lathe turning

Clutch drops into neutral while turning

Cause

- Too much gap between the jaw clutch and the left jaw clutch

Solution

Shorten the shaft sleeve

Cut is rough

Cause

- Tool dull
- Tool not ground properly
- Tool at wrong angle
- Tools not held tightly
- Wrong cutter for material
- Cutting speed incorrect

Solution

Sharpen or replace tool  
Regrind tool  
Correct tool position  
Tighten toolholder  
Use correct cutter  
Increase or reduce speed

Work has unwanted taper

Cause

- Work improperly aligned
- Debris in spindle, setup, or tools
- Offset tailstock incorrectly positioned
- Spindle out of alignment

Solution

Realign centers on work  
Clean and reset setup, work, or tool  
Correct position of tailstock  
Tighten taper bearings to return to alignment, replace spindle bearings

Machine vibrates

Cause

- Work mounted wrong
- Speed too high
- Too much pressure at tailstock

Solution

Remount work  
Reduce speed  
Reduce pressure and increase lubrication

Work stops turning but machine continues to run

Cause

- Work not mounted securely
- Tools forced into work
- Belts slipping

Solution

Remount work  
Reduce force on tools  
Tension belts, use belt dressing, or replace belts

Diameter of work is not consistent

Cause

Solution

- |  |   |
|--|---|
| <ul style="list-style-type: none"> <li>• Too much flex in workpiece</li> <li>• Too much flex in compound rest, cross slide, or carriage</li> </ul> | <p>Use a follow rest<br/>Tighten gibs, clean ways</p> |
|--|---|

Too much backlash in the compound

- | Cause  | Solution                                    |
|--|---|
| <ul style="list-style-type: none"> <li>• Loose spanner nuts</li> <li>• Worn nut</li> </ul> | <p>Tighten spanner nuts<br/>Replace nut</p> |

Machine slings oil from behind the chuck or in belt box

- | Cause   | Solution  |
|---|---|
| <ul style="list-style-type: none"> <li>• Oil reservoir overfilled</li> <li>• Worn oil seal</li> </ul> | <p>Check oil level<br/>Replace felt in seal</p> |

## 25.4 Milling

Tool chatters

- | Cause   | Solution  |
|---|---|
| <ul style="list-style-type: none"> <li>• Gibs too loose on cross slide, compound, or carriage</li> <li>• Unused feeds not locked</li> <li>• Millhead not locked</li> <li>• Quill too loose</li> <li>• Tool not on center</li> <li>• Improper tool shape, tool dull</li> </ul> | <p>Readjust gibs<br/><br/>Lock all axes but the one moving<br/>Lock millhead<br/>Tighten quill lock<br/>Center tool<br/>Reshape, sharpen, or replace tool</p> |

Depth of cut is not consistent

- | Cause   | Solution   |
|---|--|
| <ul style="list-style-type: none"> <li>• Quill moving</li> <li>• Setup wrong</li> </ul> | <p>Lock quill<br/>Make sure setup is parallel to table</p> |

## 25.5 Drilling

Hole is off center or bit wanders

- | Cause  | Solution  |
|--|---|
| <ul style="list-style-type: none"> <li>• Bit dull</li> <li>• Bit not mounted correctly in chuck</li> <li>• Bit bent</li> <li>• Chuck loose in spindle</li> <li>• Drawbar not secured</li> <li>• Debris on spindle</li> </ul> | <p>Use sharp bits<br/>Remount tool<br/>Replace bit<br/>Remount chuck on arbor<br/>Tighten drawbar<br/>Clean debris and arbor and remount tool</p> |

- Bearings loose or worn
- Cutting too fast
- Incorrect bit

Tighten or replace bearings  
Reduce speed  
Use correct bit

Entrance hole is out of round

Cause

- Bit dull
- Incorrect drill bit

Solution

Use sharp bits  
Use correct bit

Bit turns erratically or stops

Cause

- Bit fed into work too fast
- Belts slipping

Solution

Reduce feed rate  
Reduce feed rate, re-tension belts

Chuck is difficult to tighten or loosen

Cause

- Chuck sticking
- Debris in chuck

Solution

Apply lubricant  
Clean chuck

Chuck wobbles

Cause

- Chuck loose on arbor
- Drawbar not tight

Solution

Clean arbor and remount  
Clean spindle and replace drawbar

## 25.6 Drive system

Turn on machine and nothing happens

Cause

- Machine unplugged
- Loose electrical connections

Solution

Plug in machine  
Tighten wiring connections

Motor and pulleys turn, but not lathe or mill

Cause

- Lathe mill clutch is in neutral

Solution

Pull the clutch sleeve all the way out (lathe) or push it all the way in (mill)

Pulleys turn and clutch is engaged but the mill does not turn

Cause

Solution

- T-key missing
- Bevel gear stripped or broken
- Bevel gear out of adjustment

Replace key  
Replace bevel gear  
Readjust bevel gear

