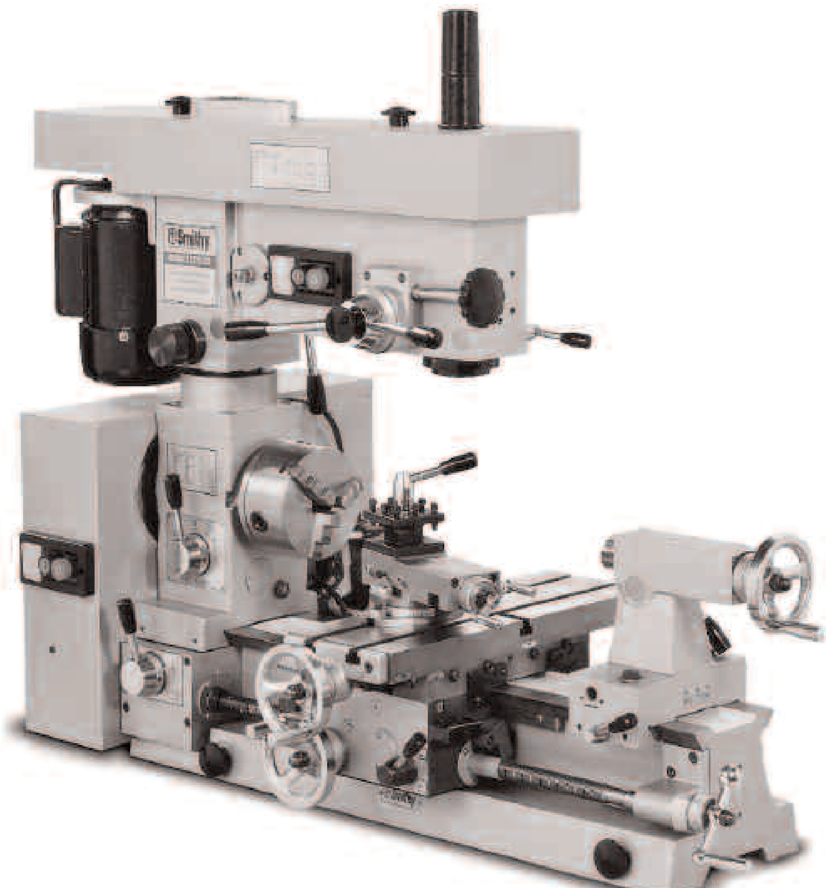
Smithy®

MIDAS 1220 LTD
COMBINATION LATHE/MILL/DRILL



OPERATOR'S MANUAL

Updated July, 2008

170 Aprill Dr., Ann Arbor, MI, USA 48103
Toll Free 1-800-476-4849
www.smithy.com

170 Aprill Dr., Ann Arbor, Michigan, USA 48103
Toll Free Hotline: 1-800-476-4849

Fax: 1-800-431-8892
International: 734-913-6700
International Fax: 734-913-6663

All images shown are from Midas 1220 LTD machine.

Printed and bound in the United States of America.

Table of Contents

Inventory Check List

Chapter 1: Introduction

Chapter 2: Safety

Chapter 3: Caring For Your Machine

Chapter 4: Basic Parts of the MI-1220 LTD

Chapter 5: Uncrating and Setting Up the MI-1220 LTD

Chapter 11: Lathe Turning

Chapter 12: Lathe Facing and Knurling

Chapter 13: Cutting Off or Parting with a Lathe

Chapter 14: Lathe Drilling and Boring

Chapter 15: Changing Gears on Your MI-1220 LTD

Chapter 16: Cutting Threads on Your MI-1220 LTD

Chapter 17: Milling

Chapter 18: Workholding

Chapter 19: Troubleshooting

Chapter 20: Removing the Quill and Quill Feed Assembly

Chapter 21: MI-1220 LTD Full Specifications

Chapter 22: Parts Diagram and Parts List

Inventory Check List

It is a good idea to take inventory of the parts of your machine soon after it is unpacked. By doing so, you can quickly determine if any parts are missing. In addition, should you find it necessary to return the machine to Smithy for any reason, the inventory will ensure that all the parts you received have been returned. It is also good to take a look at the inventory before you operate the machine so that you can be familiar with the names of all the parts of your Smithy machine.

Items Mounted To Your Machine

The items listed below are shipped mounted on the Smithy Midas 1220 LTD. Kindly check if the following items are present. Use the box before the item as your reference.

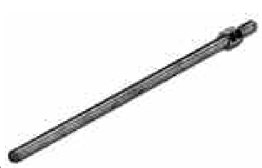
Items Mounted To Your Machine

The items listed below are shipped mounted on the Smithy Midas 1220 LTD. Kindly check if the following items are present. Use the box before the item as your reference.

❑ **5" 3 Jaw Chuck**
Part # C30532
Quantity 1

❑ **Compound Angle Toolpost**
Part # 45-110
Quantity 1

❑ **3/8" Drawbar**
Part # K99-162
Quantity 1



Items Packed in the Larger Smithy Box

The following items are packed in the larger of the two Smithy boxes.

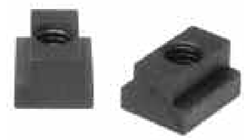
❑ **Air Mask**
Part # 15-020
Quantity 1

❑ **Protective Goggles**
Part # 15-015
Quantity 1

❑ **Ear Plugs**
Part # 15-025
Quantity 1 Set

❑ **Oil Can**
Part # 80-100
Quantity 1

❑ **7/16” T-Slot Nuts**
Part # 35-105
Quantity 2



❑ **Vise, 0-90º Adjustable Angle**
Part # 32-110
Quantity 1

❑ **End Mill, 4 FL HSS 1/4” w/3/8” Shank**
Part # 50-402
Quantity 1

❑ **End Mill, 4 FL HSS 3/8” w/3/8” Shank**
Part # 50-406
Quantity 1

❑ **End Mill, 4 FL HSS 1/2” w/3/8” Shank**
Part # 50-410
Quantity 1

❑ **End Mill Adapter**
Part # 65-010
Quantity 1

❑ **Lathe Bit Set**
Part # 43-000
Quantity 1 Set
(AR-8,AL-8,BR-8,
BL-8 & E8)

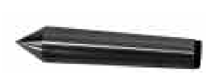
❑ **Lathe Chuck Key**
Part # C30532
Quantity 1

❑ **Drill Chuck, JT33, 0.8-13 mm with key (for tailstock)**
Part #72-001
Quantity 1

❑ **Drill Chuck Arbor, MT3/JT33**
Part # C30523
Quantity 1

❑ **Arbor Plug (M12 x 1.5 x 20 mm setscrew)**
Part # S12898
Quantity 1

❑ **MT3 Dead Center**
Part #41-003
Quantity 1



❑ **MT4 Dead Center**
Part # 41-004
Quantity 1

❑ **8/10 mm Open End Wrench**
Part # C30539
Quantity 1

❑ **17/19 mm Open End Wrench**
Part # C30535
Quantity 1

❑ **Allen Wrench, 4 mm**
Part # C30540
Quantity 1

❑ **Allen Wrench, 5 mm**
Part # C30542
Quantity 1

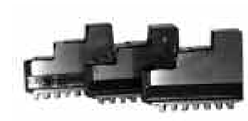
❑ **Allen Wrench, 6 mm**
Part # C30537
Quantity 1

❑ **Allen Wrench, 8 mm**
Part # C30536
Quantity 1

❑ **Belts**
Part # No Assigned Number
Quantity

Items Packed in Smaller Smithy Box

❑ **Outside Jaws for Lathe Chuck**
Part #
Quantity 1



❑ **Gear,27 Teeth**
Part # C30143
Quantity 2

❑ **Gear,30 Teeth**
Part # C30144
Quantity 1

❑ **Gear,32 Teeth**
Part # C30145
Quantity 1

❑ **Gear,33 Teeth**
Part # C30146
Quantity 1

❑ **Gear,36 Teeth**
Part # C30147
Quantity 1

❑ **Gear,66 Teeth**
Part # C30148
Quantity 1

❑ **Gear,40 Teeth**
Part # C30149
Quantity 1

❑ **Gear,42 Teeth**
Part # C30150
Quantity 1

❑ **Gear,45 Teeth**
Part # C30156
Quantity 1

❑ **Gear,48 Teeth**
Part # C30151
Quantity 1

❑ **Gear,49 Teeth**
Part # C30152
Quantity 1

❑ **Gear,50 Teeth**
Part # C30153
Quantity 1

❑ **Gear,56 Teeth**
Part # C30157
Quantity 1

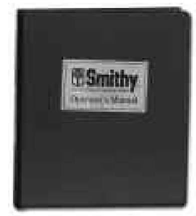
❑ **Gear,60 Teeth**
Part # C30159
Quantity 1

❑ **Gear,63 Teeth**
Part # C30160
Quantity 1

❑ **Gear,70 Teeth**
Part # C30202
Quantity 2

Items Packed in Plastic Bag

❑ **DVD, Machine Tool Basics**
Part # 12-003
Quantity 1



❑ **Manual Cover**
Part # 83-942
Quantity 1

❑ **Operator’s Manual**
Part # 83-950
Quantity 1

Missing Items?

If you find that an item is missing or defective from your Quick Start Tool Pack

Call Us TOLL FREE 1-800-476-4849
or send an e-mail to info@smithy.com

within 30 days of receiving your machine so that we may assist you immediately.
Our sales and service technicians are available 8am to 5pm ET, Mondays to Fridays.

Chapter 1

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 need to operate the Smithy Midas 1220 LTD. It will teach you about the machine's parts and how to care for them. We'll explain how to grind cutters, set up lathe tools, hold work pieces, 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 manual, please call Smithy. Our trained technicians will help you with any machining problems you may have. Dial our toll free number 1-800-476-4849 Monday through Friday, 8:00 am to 5:00pm Eastern Time. You can also find Smithy on the Internet at www.smithy.com. Check for service updated and service bulletins.

We are always interested in your suggestions to improve our products and services. Feel free to contact us by phone or email us at **info@smithy.com**. 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 **Smithy Co., PO Box 1517, Ann Arbor, Michigan 48106-1517**.

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

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

Model No.:______________________________

Serial No.:______________________________
(at the back of the lathe bed)

Purchase Date:______________________________

Delivery Date:______________________________

Sales Technician:______________________________

Chapter 2

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.

1. Know your machine. Read this manual thoroughly before attempting to operate your machine. 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 set-ups until the machine is completely stopped, and never operate it without first rolling your sleeves or tying them at your wrists.

2. Ground the machine. The MI-1220 LTD has three-conductor cords and three-prong grounding-type receptacles. Never connect the power supply without properly grounding the machine.

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

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

5. 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.

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

7. 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 facemask and earplugs, as well.

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

9. 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.

10. Avoid accidental starts. Turn the switch to OFF before plugging in the machine.

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

12. Use the recommended accessories. Understand how to use them before trying them out.

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

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

Note: Your safety depends largely on your practices.

Chapter 3

Caring For Your Machine

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

1. When you finish working, wipe machined surfaces with a clean, oily rag. Never leave the machine without this thin film of protective oil all over parts that might rust, especially ground finished parts.

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

3. Before inserting collars, centers, adapters, or drawbar attachments in either the spindle or tailstock spindle, wipe them 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 mark surfaces and interfere with the assembled part's alignment.

4. Lubricate the machine before each use as seen on Section 5.4.

5. Use good 10W 30 weight non-detergent oil on your machine.

6. Cover your machine to protect it from dust and moisture.

Note: An old machinist trick is to leave camphor in the toolbox and on the machine to prevent rust. Newer compounds that also protect machines that will unused for some time are BoeShield, developed by the Boeing Company and CRC Lubricants. There are also specialty oils that may be purchased.

Chapter 4

Basic Parts of the MI-1220 LTD

Learn the operation of your machine, you have to know the names and functions of its basic units.

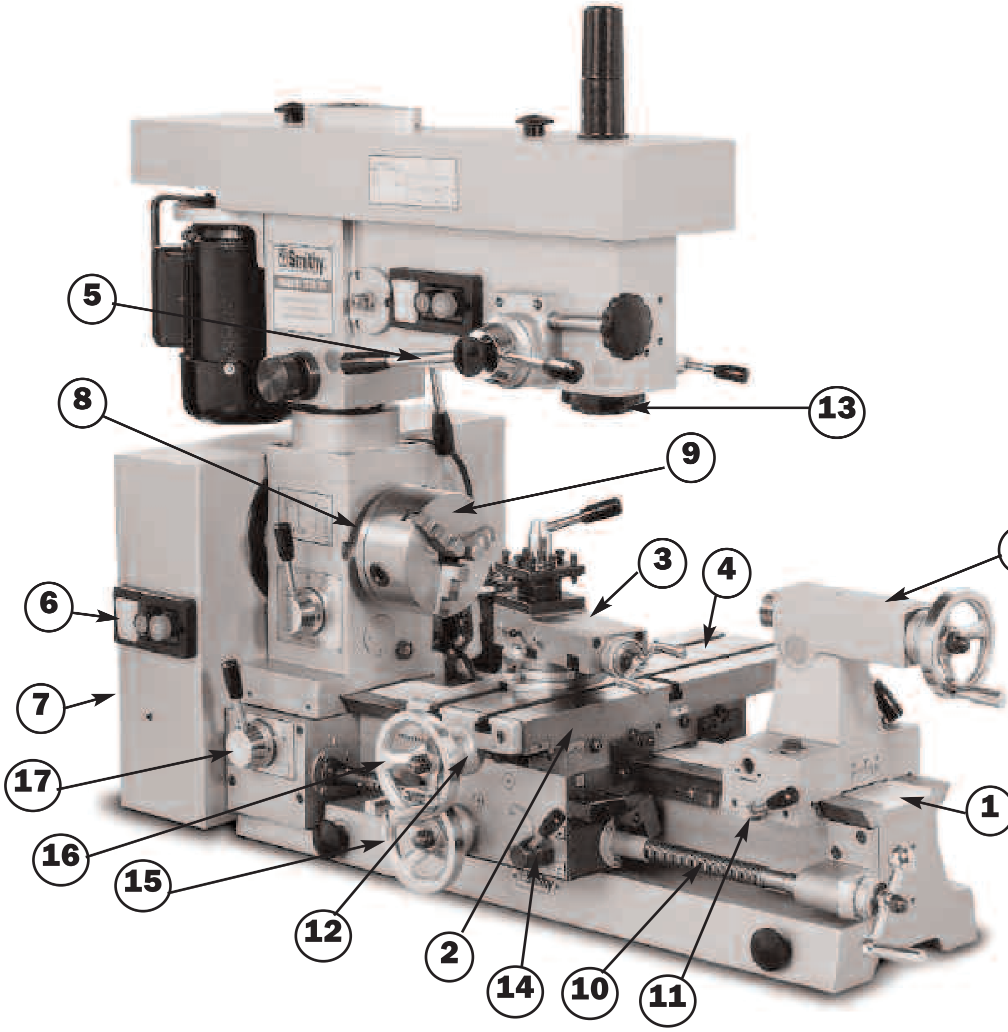


Figure 4.1 Midas 1220 LTD

1. ***Bed.*** The bed 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 tool point and work piece 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 other.

2. ***Carriage.*** The carriage 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 rigidity and move it along the bed for different operations. It locks into place by tightening the carriage lock with the setscrew on the backside of the carriage.

3. *Compound Rest.* Mounted on the cross slide, the compound rest 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 calibration on the front of the base are numbered in degrees from 60 right to 60 left.

4. *Cross Slide.* The T-slotted cross slide moves crosswise 90 degrees to the lathe axis by manual turning of the cross feed screw hand wheel. It also serves as the milling table.

5. *Drill Press and Fine Feed Clutch.* Pushing in the drill press clutch (engages the fine feed). To work the clutch, release the spring tension by rotating the drill press handles clockwise. Pull the clutch out to sue it as a drill press or push it in to use the fine feed. Use the fine fee hand wheel to move the quill up and down.

6. *Forward/OFF/Reverse Switch.* This is the main switch used to operate the lathe. 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 operation. The MI-1220 LTD has two switches, one located on the millhead and one on the right side of the gearbox.

7. *Gearbox.* The gearbox houses the belts that derive the spindle and change gears for the powerfeed. Select the thread pitch (for threading) or the feed rate (for turning) by changing the four change gears on the right side of the gearbox.

8. *Headstock.* The headstock, which is secured to the bed, houses the gears the drive the powerfeed and the taper that secure the lathe spindle.

9. *Lathe Spindle.* The end of the lathe spindle facing the tailstock is the spindle nose. The spindle nose, which has an MT4 taper, rotates the work piece and drives the lathe chicks and other workholding devices. All attachments (like three-jaw chucks, four-jaw chucks, faceplates, etc.) bolt to the spindle flange either directly or via an adapter plate.

10. *Leadscrew.* The leadscrew, 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.

11. *Locks.* Locks on the cross slide, carriage, quill, and tailstock (two) keep them from moving. During machining, lock all axes except the one you want to move.

12. *Micrometer Control and Calibration.* Just inside the handles of the tailstock crossfeed, drill calibrated in millimeters. The compound feed and crossfeed are calibrated in two thousandths, the tailstock in thousandths, the leadscrew in two thousandths, and the drill press in forty thousandths.

Note: These micrometer dial collars can move independently around the handle shafts. This independent motion is called float. The MI-1220 LTD has floating dials on the cross slide, tailstock, longitudinal and mill feeds. They let you zero the collars at any point and read the feed travel from that point on the dial for increased accuracy.

13. *Mill Spindle.* The mill spindle 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 fir into collets that attach through the spindle via drawbars.

14. ***Half-nut Lever.*** This lever transmits power to the carriage for threading.

15. ***Power Longitudinal Feed.*** Push the lever down to engage the power of the long feed for general cutting.

16. ***Power Cross Feed.*** Push the lever down to engage the cross feed and pull it up to disengage.

17. ***Powerfeed Speed Selector.*** The two-speed selector for powering the leadscrew is on the front of the headstock. The leadscrew turns twice as fast in the II position as in the I position.

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

Note: To offset the tailstock, loosed the four base locking bolts. To offset to the left, loosed the left adjusting bolt and tighten the right and do the same on the other side when you want to offset to the right. See figure. 4.2.

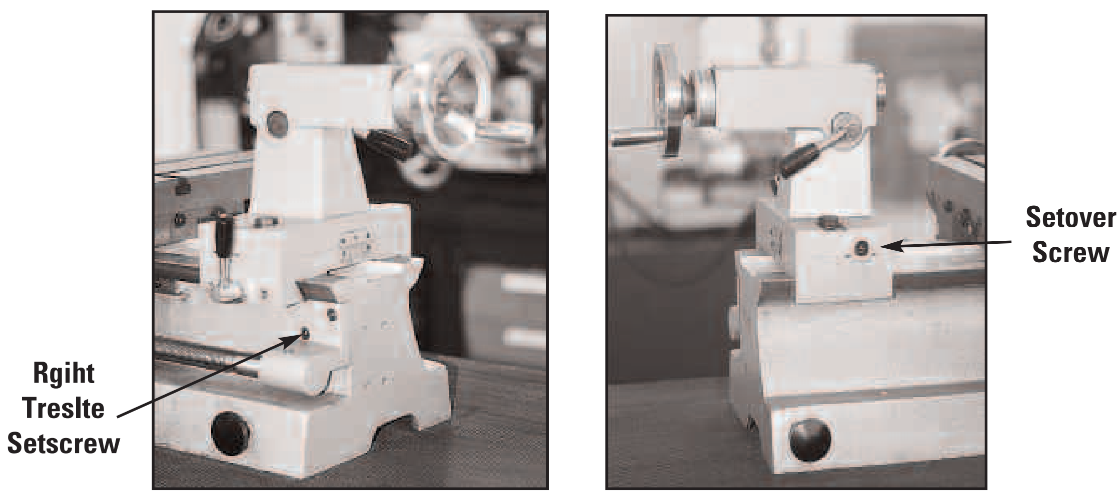


Figure 4.2 Tailstock base locking bolts.

Chapter 5

Uncrating and Setting Up the MI-1220 LTD

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 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.

Uncrating and Positioning the Machine

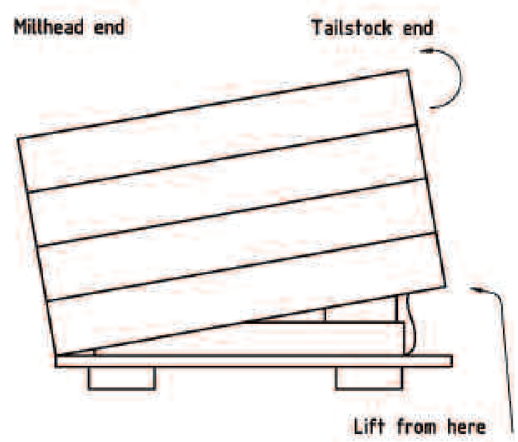


Figure 5.1 Tip the crate from the tailstock end up and over the machine.

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

The metal bands that encircle the crate are under tension. Wearing eye protection and gloves, cut the metal bands with tin snips.

Caution

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 5.1). Do not damage the crate. You may need it another time to transport the machine.

Once your crate cover is removed it is time to put your machine on its bench. The machine is just less than 500 pounds so make sure you have some extra hands to help. There are four lifting pints that pull out from the bed of the lathe. You can use chains or a tow rope to wraparound these pins and the aid of a lifting device such as an engine hoist to list the machine on to a bench rated to support the machine's weight.

Without a mechanical device to aid in your lifting you can lighten the machine by removing a few or all of the following:

Millhead

1. Remove the four hexagon socket-head screws at the base of the millhead support column. If a screw runs through the belt box into the flange of the support column, remove it too.

2. Lock the millhead-locking handle.

3. Lift the millhead and column off the lathe head. You may have to rock it back and forth while lifting it. Make sure that the mill head is locked to the column before removing the millhead.

Tailstock

1. Loosen the tailstock lock and pull the tailstock off the end of the bed. The gib and locking pin will fall out. Be careful no to lose them.

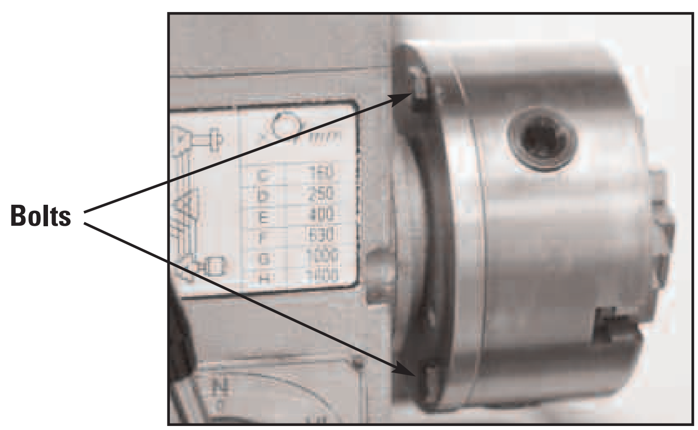


Figure 5.2 The chuck attaches to the spindle flange with three bolts. The one bolt located on the other side of the spindle does not show.

Three-Jaw Chuck

1. Remove the three bolts behind the chuck that hold it to the spindle flange (Figure 5.2). The chuck will come off. Don't let it fall onto the ways. Placing a board between the chuck and ways will protect the ways.
Place the machine on a strong, rigid table 40" long, 24" wide and 28" to 33" high. We recommend you to bolt down the MI-1220 LTD machine using the holes in the base of the bed or using the lifting handles as they held the machine to the shipping pallet.

Selecting a Location

There are several major considerations for selecting a location for your Smithy.

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

The machine should be on a 20-amp circuit, positioned as close as possible to the power supply. Try not to use an extension cord. If you must use one, check with an electrician about the proper size.

Provide ample working light over the operator's shoulder.

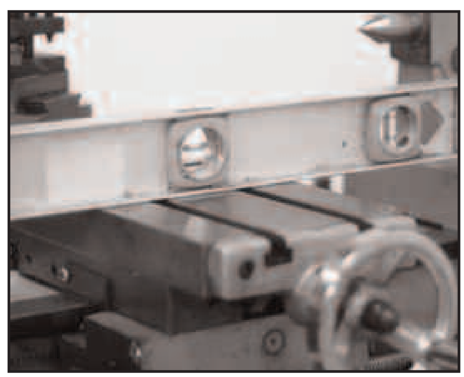
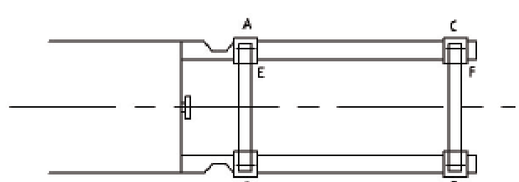


Figure 5.3 Check along and across the bed to make sure it is level.

Place the machine on a solid foundation, concrete if possible. If you must put it on a wood floor, make sure it is adequate. Brace it if necessary to prevent sagging or settling.

Make allowances at the back of the machine tool as at its 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.

Notice **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 place should be better than 0.0016/40"; alignment in the traverse plane should be better than 0.0024/40".**



Cleaning and Lubricating the MI-1220 LTD

Smithy machines are shipped with protective grease coating called cosmoline. Use WD-40 or non-corrosive kerosene to remove the cosmoline.

Once you have your MI-1220 LTD set up and positioned correctly, you are ready for lubricating. You must do this carefully and thoroughly before starting the machine. Use a pressure oil can and a supply of good quality SAE No.10 weight oil.

To be thorough and complete, follow this routine:

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, special oil formulated for the ways.

Figure 5.4 Oiling the ways

Oiling the Millhead Quill

Using your mill handles or your fine feed crank to lower the millhead down. Apply a thin layer of oil to the quill and work it down and up until it runs smoothly.

Figure 5.5 Oiling the Millhead Quill

Oiling the Headstock

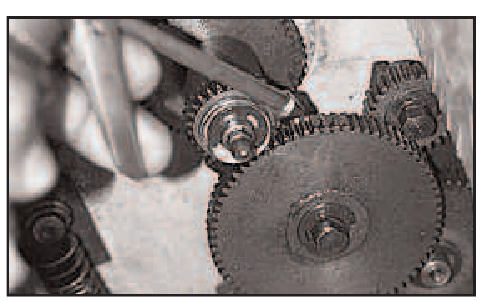


Figure 5.6 Oil the button behind the D gear.

Open the gearbox door to expose the pick-off gears. Oil the button in the casting behind the D gear. Then put a few drops of oil on the teeth of all the gears. Grease the zerk on the A gear shaft.

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. Be careful not to overfill it. The gearbox requires only 8 to 10 ounces of oil.

Oiling the Carriage

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

Oil the button in the center of the cross slide.

Put a few drops of oil on the compound slides.

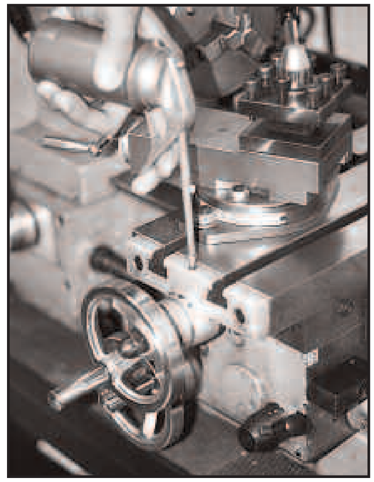


Figure 5.7 Oiling the table

Oiling the Compound Angle Toolpost

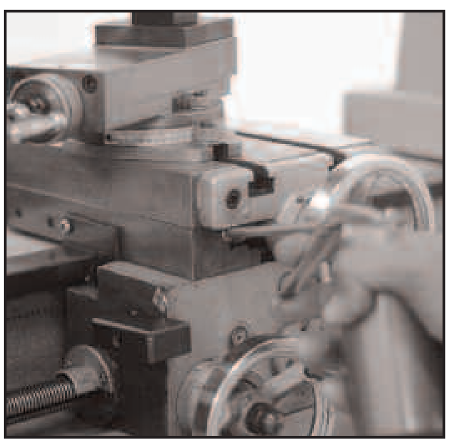


Figure 5.8 Oil the buttons along the cross feed table.

Oil two buttons on top of the compound angle toolpost.

Oiling the Apron

Put oil in the button just behind the cross slide hand wheel.

Put oil on the button at the back of the cross slide.

Oiling the Leadscrew

Put oil in the oil buttons on the left trestle.

Put oil in the support for the right end of the leadscrew.

Oiling the Tailstock

Figure 5.9 Oil the two buttons on the top of the tailstock.

Oil the buttons on top of the tailstock.

Oiling the Mill/Drill Clutch

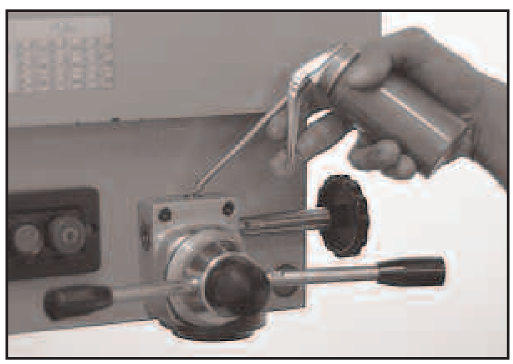


Figure 5.10 Oil the clutch housing button.

Put oil in the button on top of the clutch housing.

Notice

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

Adjusting Belt Tension

The MI-1220 LTD has two belt tensioners installed by the factory. One for the millhead and the other one for the pulley box.

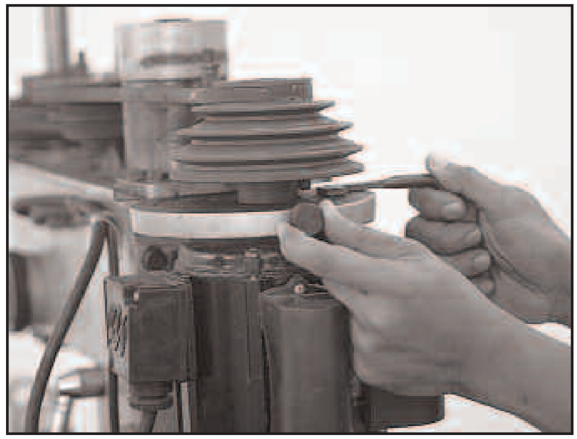


Figure 5.11 Mill belt

Mill

Locate the "L shaped" lever and a thumbscrew at the top of the mill motor. Loosen the thumbscrew and then rotate the lever to increase or decrease the belt tension. Re-tighten the thumbscrew when the desired tension is achieved.

Figure 5.12 Lathe belt tensioner.

Lathe

Locate the belt tensioner handle on the motor mount. To tighten the lathe belts, move the tensioner handle upward so that the handle points toward the lathe head. Turn the knurled knob clockwise to tighten the belt and counterclockwise to loosen it.

Adjusting the Gibs

The MI-1220 LTD machines have straight gibs. Before using the machine, adjust the gibs evenly. First tighten the screws all the way. This will lock the movement. Then loosen each screw one quarter turn and check it. Tighten the gib, the more accurate it will be. Removing and polishing the gibs also improves the tolerances.

With the gibs properly adjusted, review the following instructions on how to reduce the backlash, or lost motion in the screw, which also depends on the type of job you're doing and/or individual preference.

Reducing Backlash

Backlash of 0.008-0.015" as measured on the dial is normal. If you have more backlash than that in your crossfeed table, refer to the schematics at the back of this manual, if necessary and follow these directions:

1. Tighten the cap nut in the center of the cross feed hand wheel securely.

2. Tighten the set screw inside the T-slot so the brass nut cannot move.

3. Tighten the screw in the base of the brass nut. This will remove play between the threads in the cross feed screw and nut. Do not over tighten it or there will be excess wear on the nut.

If there is still excess backlash, place one or more shim washers between the large shoulder of the cross feed screw and the bush bearing. Ask a Smithy technician about our antibacklash shim washer kit, Item number K99-190.

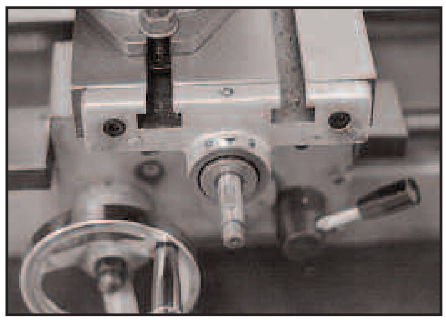


Figure 5.13 To reduce backlash, tighten the setscrew so the bush bearing will be secured.

To install shims, turn the hand wheel clockwise to move the cross table away from the screw seat. Loosen the setscrew. Then pull out on the hand wheel until the bush bearing is free of the seat. Remove the cap nut, hand wheel, dial, keys, and bush bearing. Install one or more shim washers and reassemble.

Running in the MI-1220 LTD

Though all Smithy machines are run at the factory and again before shipping, it is wise to put your machine through a break-in run before putting it to work. After oiling the machine, check the belts to make sure the tensioners are correct. Do not plug your machine yet. ***Follow these steps:***

Millhead Run-in

1. Make sure that the power switch for the lathe motor and the mill motor are both in the off position.

2. Close the door of the gearbox before starting your machine.

3. Plug the machine into a grounded 20-amp circuit.

4. Start the mill motor by pushing in the green start button. After a few minutes, push in the red stop button and allow the motor to stop. Flip the yellow switch cover and switch it to the opposite position and repeat the above procedure.

5. Start the lathe by pushing the green button on the lathe control panel.

6. Engage the half nut by pushing down the half nut handle, pull up to disengage. Do the same with the cross feed and the longitudinal feeds.

7. Push the lathe stop button and allow the motor to stop. Move the direction selector to the left and flip the yellow cover and switch the red toggle switch to the opposite position and repeat the above procedure.

During the run-in, try all of the controls. Get a feel for your machine before you start to work.

Caution

This machine is equipped with power crossfeed and longitudinal feed.

Caution must be taken to not run the power feeds past their limits of travel. As part of normal operation procedures, run each axis through the entire length of the proposed machining operation before engaging any of the power feeds to assure there is sufficient travel to accomplish the desired task. Failure to do so could result in running one of the power feeds to the end of its mechanical limits. This is what is known as a "CRASH". A crash can cause damage to the work piece and severe damage to the machine. Remember that becoming familiar with your machine is the best safety insurance you can have.

Lathe Run-in

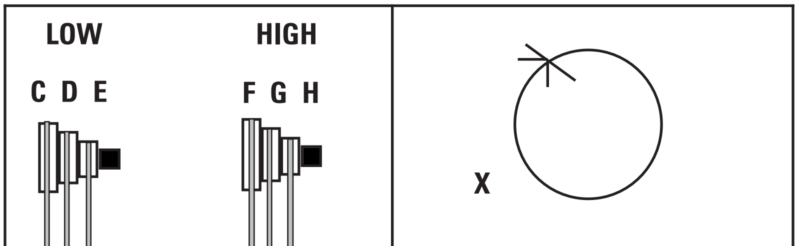
1. Start the lathe by pushing the green button on the lathe control panel.

2. Engage the half nut by pushing down on the half nut handle, pull up to disengage. Do the same with the cross feed and the longitudinal feeds.

3. Push the lathe stop button and allow the motor to stop. Move the direction selector to the left and flip the yellow reversing switch to the opposite position and repeat the above procedure.

During the run-in, try all of the controls. Get the feel for your machine before you start to work.

Setting Lathe and Mill Speeds for the MI-1220 LTD

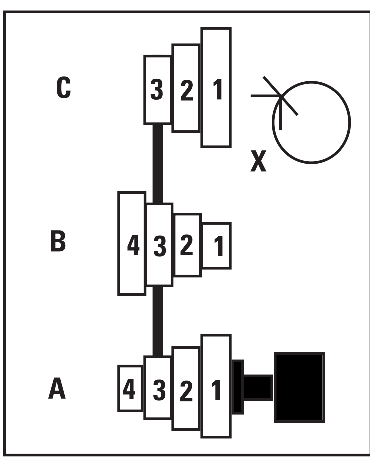


C	160
D	250
E	400
F	630
G	1000
H	1600

Figure 5.14 Setting Lathe Speeds (RPM)

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 5.14) and on the largest sheave of the idler pulley. For 160 RPM, se 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 once sheave for 250 RPM (position D) and one more for 400RPM (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. Move it 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.



A4 X B1 / B4 C1	315	A3 X B1 / B3 C1	630	A2 X B3 / B2 C3	1250
A4 X B2 / B4 C2	400	A2 X B1 / B2 C1	800	A1 X B2 / B1 C2	1600
A4 X B3 / B4 C3	500	A3 X B2 / B3 C2	1000	A1 X B3 / B1 C3	2000

Figure 5.15 Setting Mill/Drill Speeds (RPM)

Set mill speeds using various combinations of the mill belts. For 315 RPM, place belt A/B in position 4 and belt B/C in position 1. For 500 RPM, leave belt A/B belt in position 4 and move the B/C belt to position 3.

Chapter 6

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 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.

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 6.1). A point on a 3" diameter shaft will pass the cutting tool three times as fast as a point on a 1 " diameter shaft rotating at the same speed. This is because the point travels a tripled circumference.

For work in any given material, the larger the diameter, the slower the speed in spindle revolutions needed to get the desired feet-per-minute (fpm) cutting speed.

Lathes cut threads in various numbers per inch of material threaded, according to the operator's needs. The MI-1220 LTD cuts metric threads and inch threads standards.

In thread cutting, the carriage carries the thread-cutting tool and moves by rotating the leadscrew . The basic principle is that the revolving leadscrew pulls the carriage in the desired direction at the desired speed. The carriage transports the toolrest and the threading tool, which cuts the screw thread into the metal 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.

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

Table 6.1 Cutting Speeds for Various Diameters

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.

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 MI-1220 LTD has pick-gear gearboxes; gears are picked and placed to change the gear ratios. The gearbox mechanism determines the leadscrews rotation rate in relation to the spindles for threading, turning, and facing. To change the feed rate, replace the gears per Figure 6.2.

Figure 6.2 (missing)

Chapter 7

Metal Theory

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 increasing its wedging action. Cutter shape is usually more important than edges, which generally are rough-ground and usually must be honed for fine finishing cuts or work in soft, ductile materials like brass or aluminum.

Lack of clearance, which lets a tool drag on the work below the cutting edge, 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 tool holder you use. TheCB-1220 XL LTD have a four-sided turret toolpost that accommodates four high-speed-steel (HSS), carbide-tipped, or indexable carbide turning tools.

Heat

The energy expended at the lathe's cutting point converts largely into heat, and because the energy expended is great, the heat is intense. Before 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 a small lathe 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 when working on long shafts, to check the tightness of the lathe centers frequently and make sure workpiece expansion does not cause centers to bind.

	Low-Carbon Steel	High-Carbon Steel Annealead	Alloy Steel Normalized	Aluminum Alloys	Cast Iron	Bronze
Speed (sfm) **Roughing** **Finishing**	**90** **120**	**50** **65**	**45** **60**	**200** **300**	**70** **80**	**100** **130**
Feed (ipr) **Roughing** **Finishing**	**0.010-0.202** **0.003-0.005**	**0.101-0.020** **0.003-0.005**	**0.010-0.020** **0.003-0.005**	**0.015-0.030** **0.005-0.010**	**0.010-0.020** **0.003-0.010**	**0.010-0.020** **0.003-0.010**

Table 7.1 Cutting Speeds and Feeds for High-Speed-Steel Tools

In everyday lathe operations like thread cutting and knurling, always use 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.

Table 7.1 lists cutting speeds and feeds for HSS cutters so you can set up safe rpm rates.

The formula is as follows:

rpm=CSx4 / D"

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

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

If you were to make a finish cut on a piece of aluminum 1" in diameter, for example, you would see the desired sfm per Figure 7.3 is 300. Then:

rpm = 300 sfm x 4 / 1

rpm = 1200 / 1

rpm = 1200 or next slower speed.

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

rpm = 50 sfm x 4 / 1
rpm = 200 / 1

rpm = 200 or next slower speed.

The four-turret toolpost lets you mount up to four different tools at the same time. You can install all standard-shaped turning and facing tools with 1" 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.

Chapter 8

Grinding Cutter Bits for Lathe Tools

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.

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

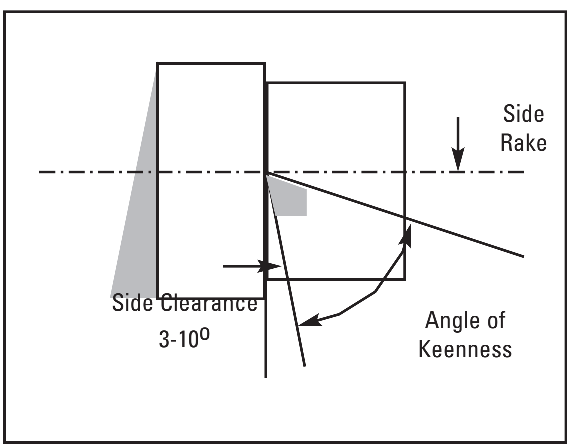


Figure 8.1 Keenness angles vary from 60 to 80 degrees.

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

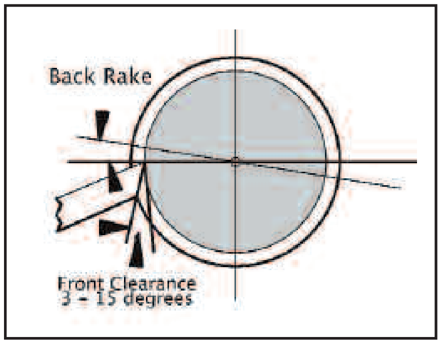


Figure 8.2 The edge weakens if front clearance is too great.

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 ungrounded cutter bit, the procedure (Figure 8.3) is usually to:

1. grind the left-side clearance

2. grind the right-side clearance

3. grind the end form or radius

4. grind the end clearance

5. grind the 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 oilstone as shown in Figure 8.4.

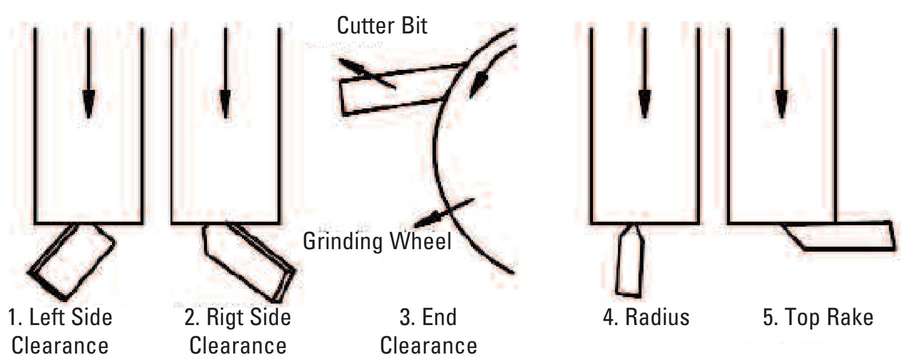


Figure 8.3 Grinding sequence for an unground cutter bit.

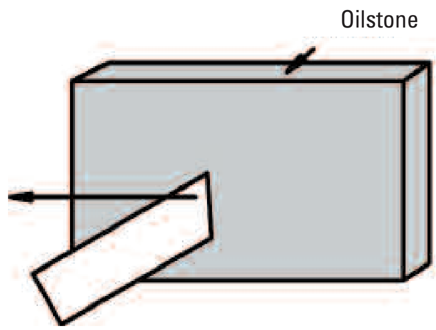


Figure 8.4 When honing, draw the cutter away from the cutting edge across the oilstone.

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° are possible.

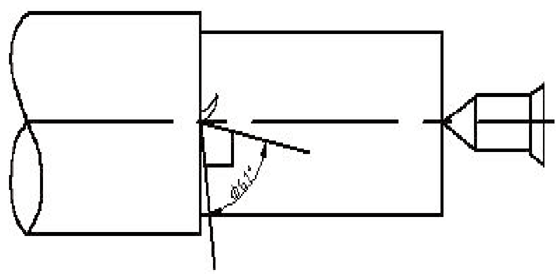


Figure 8.5 With soft steels, 61 degree angles are possible.

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

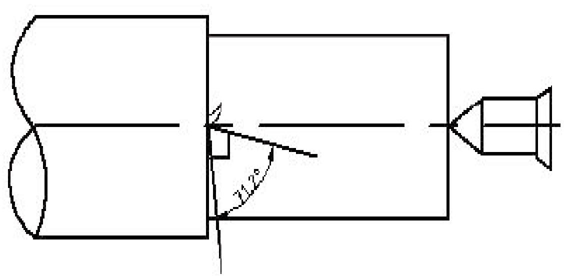


Figure 8.6 With cast iron, a 71 degree angle is most efficient.

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.

Special Chip Craters and Chipbreakers

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

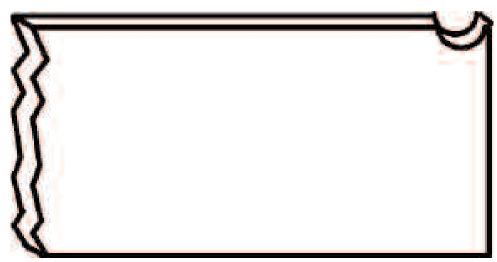


Figure 8.7 A crater starts the chip curling smoothly.

Using a Center Gauge to Check V-Thread Forms

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

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

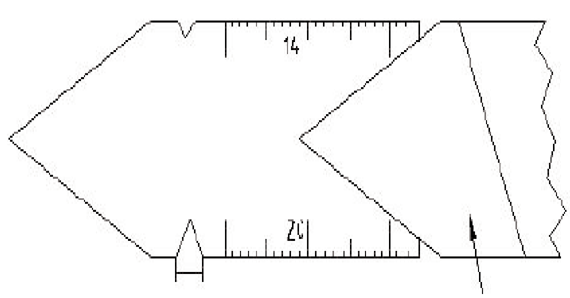


Figure 8.8 Insert the point into the nearest seized V in the center gauge.

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.

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 8.9).

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

Application	Use	Grade
Cast Iron	Roughing cuts	C-1
Non-ferrous, non-metallic, high-temperature alloys	General purpose	C-2*
200 and 300 Series stainless steels	Light finishing Precision boring Roughing cuts General Purpose	C-3 C-4 C-5 C-6*
Alloy steels	Finishing cuts	C-7
400 Series stainless steel, high velocity	Precision boring	C-8

Table 8.1 Carbide Types and Cutting Tool Applications

Chapter 9

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.

Cutting Tool Height

After inserting the cutting tool into the toolpost, adjust the height of the cutting edge in relation to the lathe center. Insert a center in the tailstock. Then run the tool and center together.

The cutting edge on the tool should meet the point on the center. It may be necessary to use shims, which can be of various thicknesses and materials (Figure 9.1). Many seasoned cutting-tool height 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.

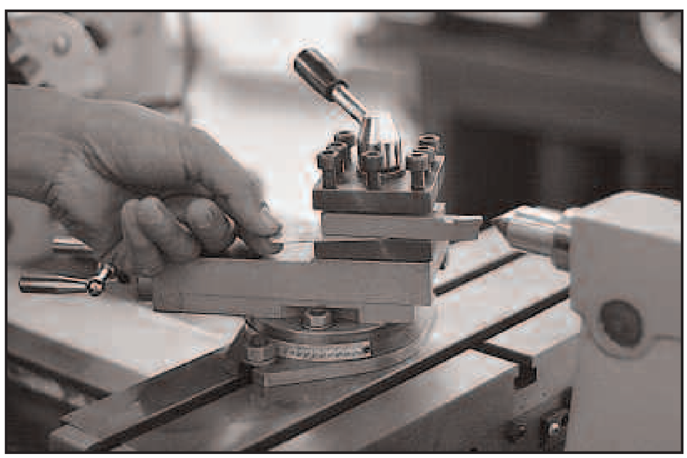


Figure 9.1 Placing shims under the tool can correct cutting too height.

Turning Tools

For general turning operations, set the point of the cutter bit slightly above the centerline of the work. In steel, the harder the material, the less above center (Figure 9.2, left).

Exceptions are soft brass, aluminum, and materials that tend to pull or tear. When machining these materials, set the cutter on dead center (Figure 9.2, right).

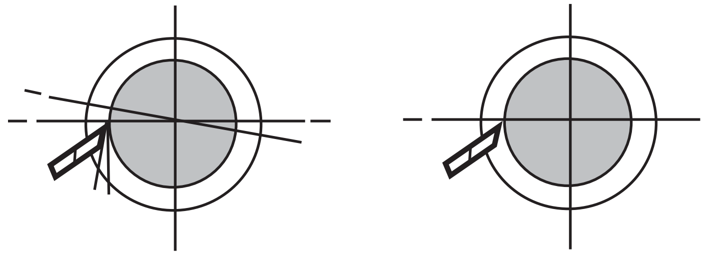


Figure 9.2 The harder the steel (left),the less above center you set the cutter point. For soft brass and aluminum (right), set the cutter on dead center.

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

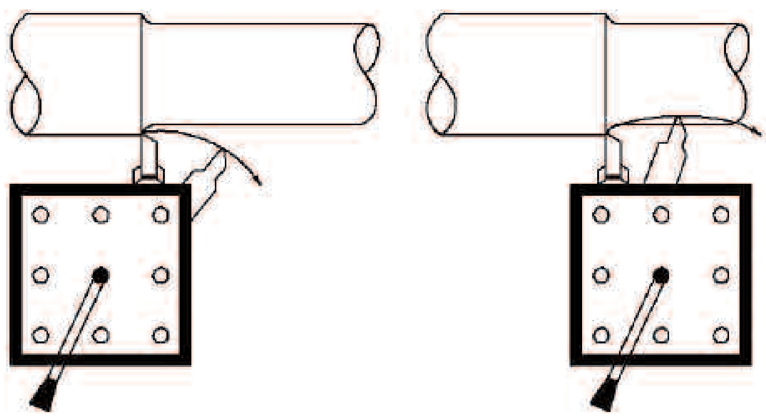


Figure 9.3 The tool will swing out of the cut (left) rather than hog into the work (right) if you reach a stalling point. Note the tool is in the right-hand side of the toolpost.

Threading Tools

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

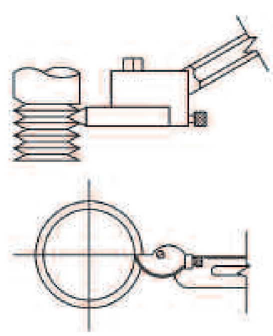


Figure 9.4 Threading tools engage the work on dead center.

Cutoff, Thread Cutting and Facing Tools

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

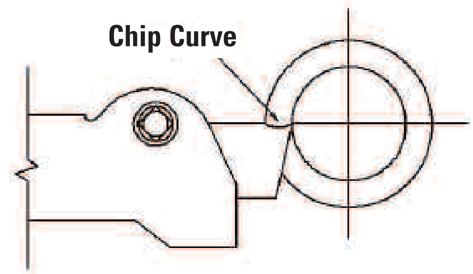


Figure 9.5 Feed the cutter on dead center for cutoff, thread cutting and facing.

Boring and Inside Threading Tools

For boring and inside threading, the cutter point engages the work on dead center (Figure 9.6). For greater cutting efficiency, position the bar while parallel to the line of lathe centers sufficiently below center to give the cutter a 14-1/2 degree 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.

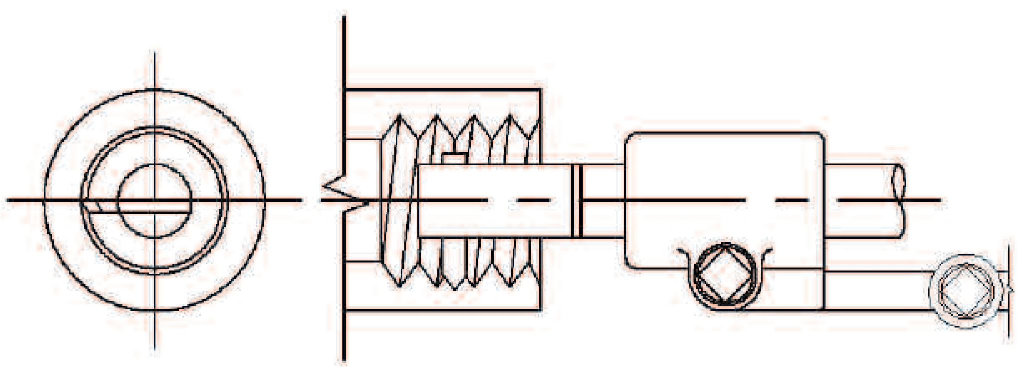


Figure 9.6 For boring and inside threading, the cutter point is at dead center.

Chapter 10

Setting Up with Centers, Collets, and Chucks

Before setting work up on centers, make sure the spindle and tailstock centers align accurately. Do this by inserting a center into the nose spindle and inserting the tailstock center into the tailstock ram. Then move the tailstock toward the headstock until the centers touch (Figure 10.1). You can correct any lateral alignment error by adjusting the tailstock set over screws (Figure 4.8).

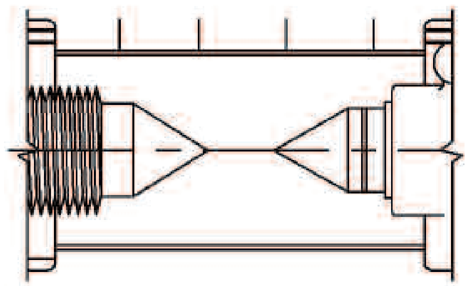


Figure 10.1 When aligning spindle and tailstock centers, move the tailstock toward the headstock until the centers touch.

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

Centering

You can improve centering greatly by first squaring or facing the ends of the workpiece (Section 12.1). This gives you a true cross section in which to locate the centering holes.

First, chuck the stock in the appropriate chuck. Let the stock protrude about an inch. Place a right-hand side tool (or a straight turning tool with a facing cutter) in the toolpost. Carefully adjust the cutting edge so it is exactly on center, then tighten it into the toolpost. If you don't do this, a small tit or projection will remain in the center of the stock 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 10.2) with calipers, dividers, or special centering instruments (Figure 10.3). Centering square or rectangular stock is done by scribing lines from opposite corners. The intersection of these lines is the center (Figure 10.4).

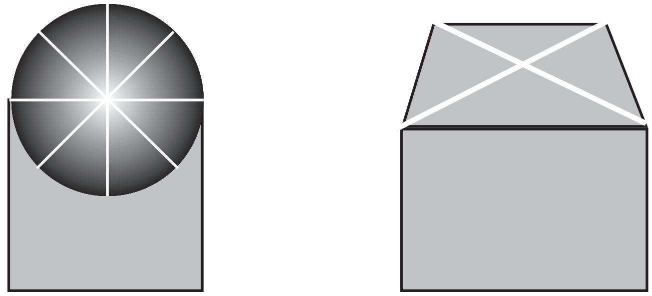


Figure 10.2 Centering on round stock and
Figure 10.4 Centering on square or rectangular stock.

Figure 10.3 Use centering instruments include calipers and dividers.

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 10.5). If the center is off 0.002" or more, correct the position of the center by repunching at an angle.

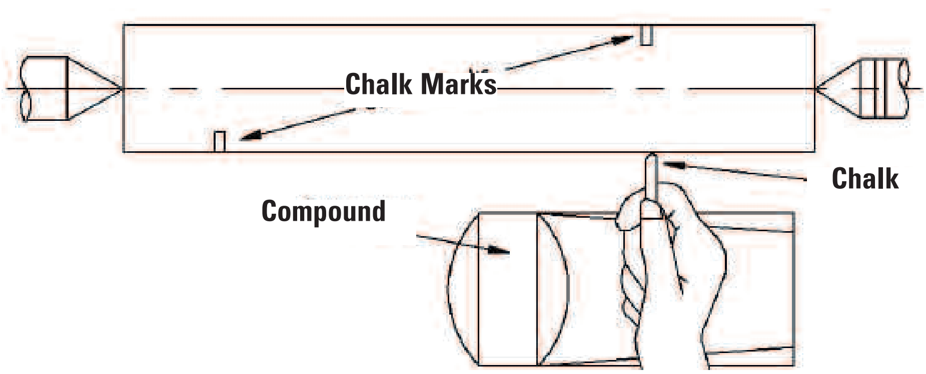


Figure 10.5 When you revolve the headstock against a piece of chalk, the chalk should just touch the high spots.

Next, drill and countersink the centers to conform to the profile of the lathe centers. This is best done with a 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 center points. Correct center depth is given in Figure 10.6. Take care to get an accurate 60° countersink in the center (Figure 10.7).

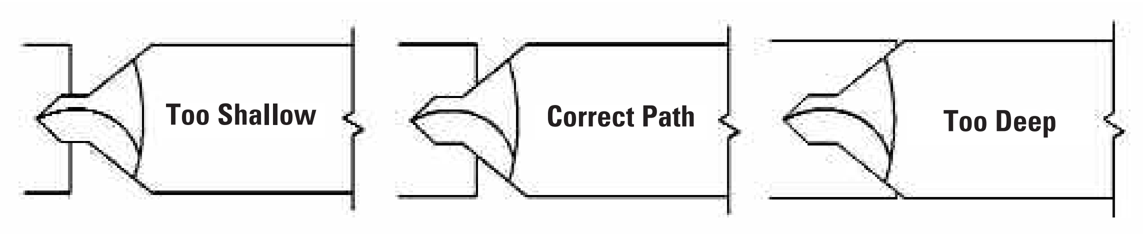


Figure 10.6 The correct depth of center is illustrated above. If it's too deep (bottom), only sharp outer edges will contact the center.

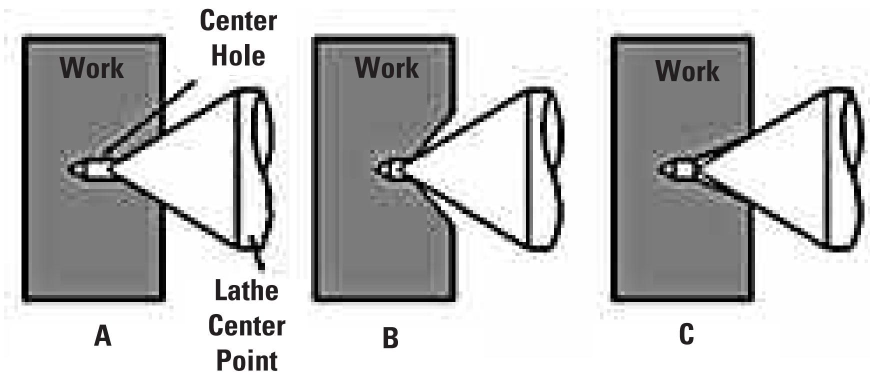


Figure 10.7 Counterbore centers with a drill to a 60° point so they fit the lathe centers (A). Too obtuse (B) or too acute (C) a counterbore will give insufficient bearing, and destroy the lathe centers.

Mounting Work Between Centers

Remove the chuck from the lathe, bolt the faceplate to the spindle if l angle (Figure 10.8), and put in both headstock and tailstock centers. Fasten a lathe dog (Figure 10.9) to one end of the work. For ease of operation, use a live or rotating center in the tailstock end so you won't need lubrication.

Before centers 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.

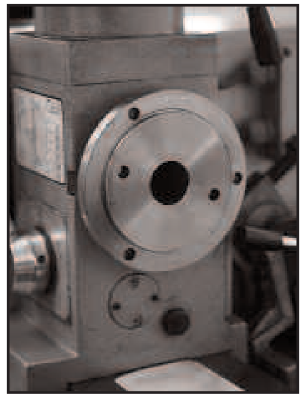


Figure 10.8 Bolt the faceplate to the spindle flange

Using a Clamp Dog

Standard lathe dogs drive round, or near-round, shapes. Rectangular or near-rectangular stock requires clamp dogs. In a properly made clamp dog, the under face 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 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.

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 (Figure10.9). Faceplates for holding workpieces (irregularly shaped casting, machine, or die parts, for example) are usually larger and have varied designs. 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 setup tools.

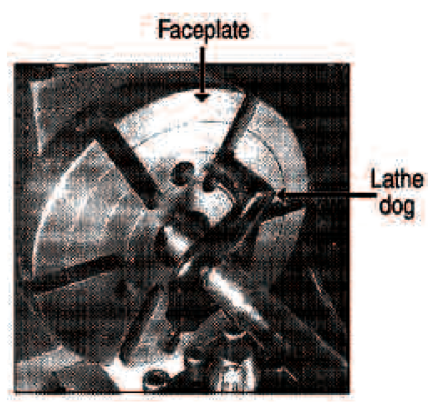


Figure 10.9 Fasten a lathe dog to one end of the work piece.

Note: Before starting to machine work set up on centers, check to see the lathe dog tail is free in the faceplate slot so it won't lift stock off its true line of centers, as in Figure 10.10. 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 10.11).

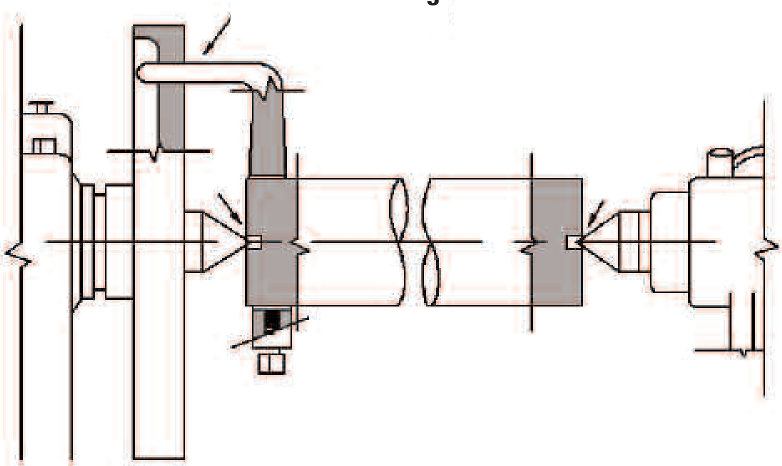


Figure 10.10 Make sure the lathe dog tail is free in the faceplate slot so it won't lift off the true line of centers.

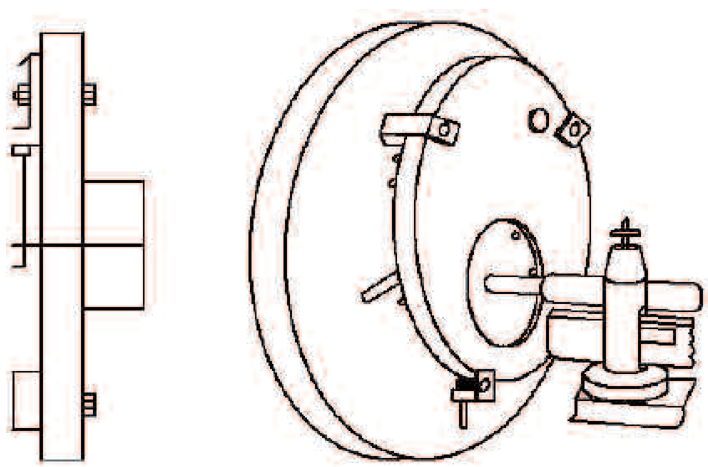


Figure 10.11 Counterweights can help with unbalanced setups.

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 10.12). Then mount them between centers. The solid mandrels, which are driven into the hole of the work-piece, 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. You can purchase hardened steel mandrels, which have a slight (0.003") ground taper and an expanding collar, to facilitate mounting and demounting (Figure 10.13). 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.

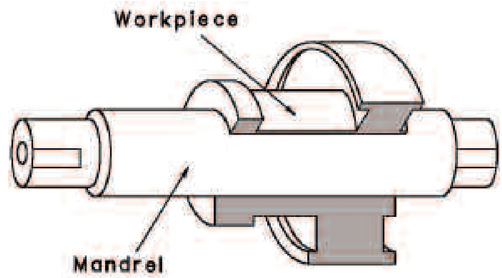


Figure 10.12 Mount workpieces too long for a chuck on a mandrel.

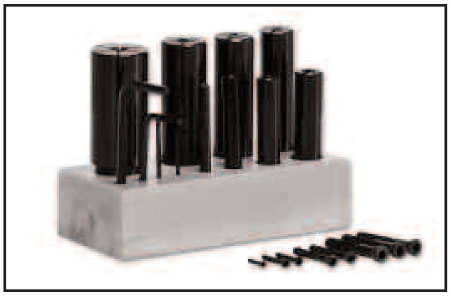


Figure 10.13 Hardened steel mandrels have a slight ground taper and expanding collar.

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 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 (Figure10.14). 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 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 will be untrue. The tips of the jaws are bronze and require lubrication.

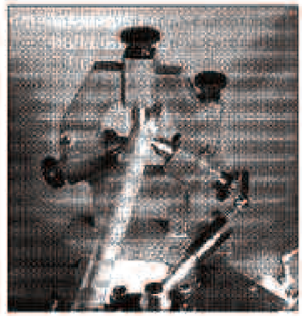


Figure 10.14 Steady rests mount on the lathe bed and provide three bearing surfaces

Follow Rests

Long or slender shafts that are apt to spring out of have a slight ground taper and alignment by the thrust of the cutting tool often require a follow rest expanding collar. 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.

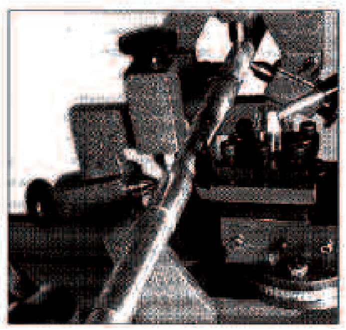
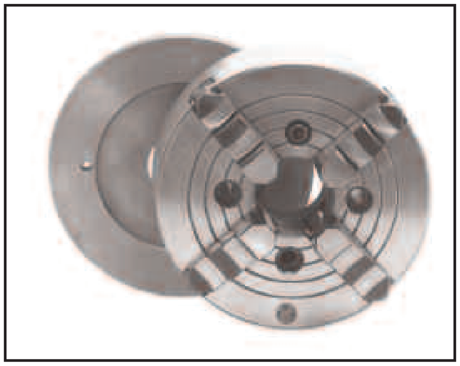


Figure 10.15 Follow rests mount on the lathe carriage and move with the tool.

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 10.16). 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 10.17).



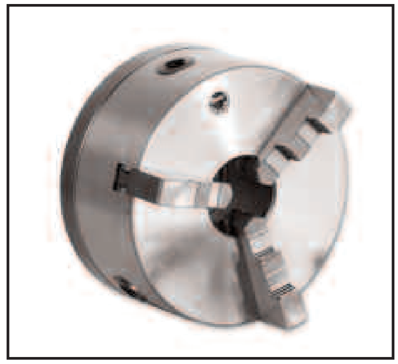


Figure 10.16 Four-jaw independent lathe chucks hold round, square, eccentric, or odd shaped workpieces. and Figure 10.17 Three-jaw universal geared scroll chucks hold round or near-round workpices.

Mounting Work in a Four-jaw Independent Lathe Chuck

For small-diameter, short work, insert jaws in the chuck with high ends to the center. This gives the maximum gripping and tool clearance (Figure 10.18). 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 10.19).

To place work in a chuck, follow these steps:

1. 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.

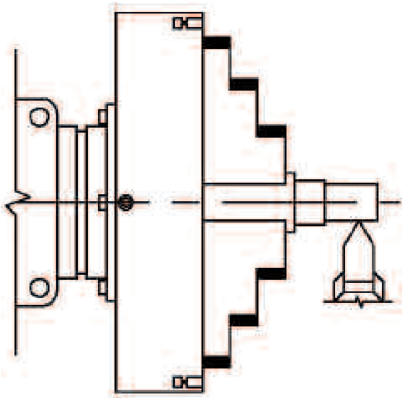
2. Place the work in the chuck and grip it. Turn up the opposing jaws a uniform number of turns with the key provided. This will hold the work in position. Then bring in the other pair of opposing jaws the same way.

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

4. Guided by the chalk marks, readjust the jaws until a chalk line lathe chucks hold round, will carry completely around the work. Then tighten all the jaws securely. square, eccentric, or odd-shaped workpieces.

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.



Figuere 10.18 For short, small-diameter workpieces, insert the jaws with high ends to the center.

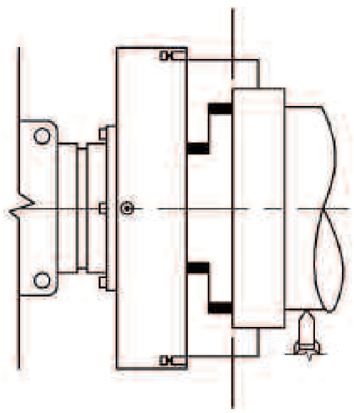


Figure 10.19 For large-diameter workpieces insert the jaws with high steps of the jaws to the outside.

Caution

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

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

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 10.20). With the #1 slot in the 12:00 position, the #2 slot is at 8:00 and the #3 slot at 4:00 (Figure 10.21).

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.

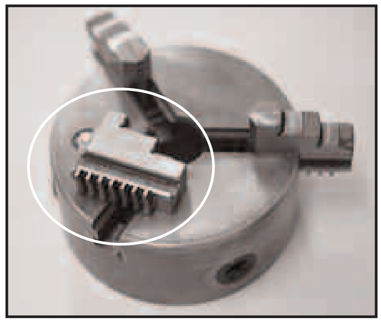


Figure 10.20 A serial number is stamped in the #1 slot of the three-jaw chuck.

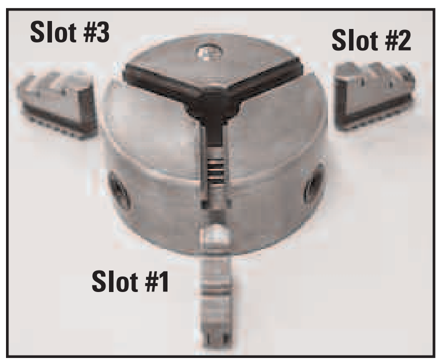


Figure 10.21 With the slot for the #1 jaw in the 12:00 position, the slot for the #2 jaw is at 8:00 and the slot for the #3 jaw is at 4:00.

Collets and Collet Attachments

To hold small-diameter work, whether bar stock fed through the hole in the spindle or small pieces of semi finished parts, collet attachments are preferable to standard chucks (Figure 10.22) for several reasons:

• They have much faster release and grip actions.

• They center the work automatically and accurately

• They grip even small pieces and pieces with a short hold firmly.

Figure 10.22 Collet attachments are best for small-diameter work.

• 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 spindles MT5 taper to the collets MT3 taper. A drawbar holds the collet in place.

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.

Chapter 11

Lathe Turning

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. With the MI-1220 LTD use a transverse powerfeed for heavy cuts-from right to left toward the headstock so the thrust is against the head-stock or the chuck. Use a right-hand turning or roundnose cutter.

Caution

Remember caution must be taken to not run the powerfeed past their limits of travel. As part of the normal operation, procedures, run each axis through the entire length of the proposed machining operation before engaging the powerfeed to assure there is sufficient travel to accomplish for the desired task. Failure to so could result in running the power feed to the end of its mechanical limit. This is what is known as a "CRASH". A crash can cause damage to the work piece and severe damage to the machine.

After selecting a cutter, place it into the left side of the turret . 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 9.1).

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.

Engage the longitudinal feed by moving the powerfeed engagement lever done. Always cut deeply enough to reach below the scale on oxidized bars or iron castings. Hard, oxidized surfaces dull tools rapidly.

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.

Note: The diameter will reduce twice the thickness of the cut.

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

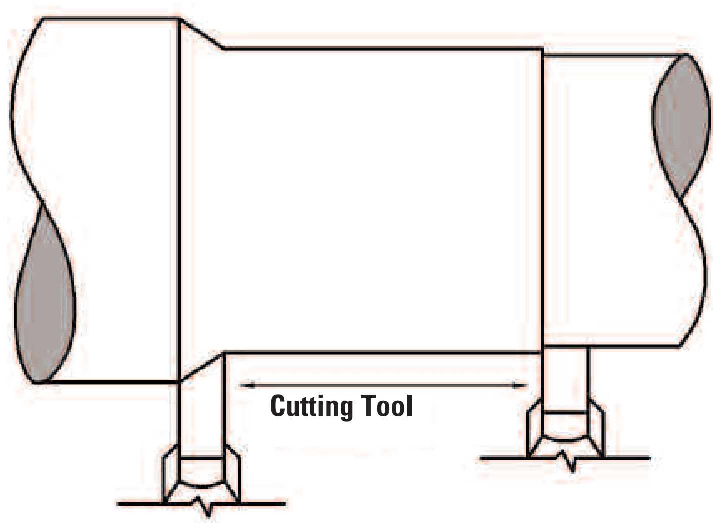


Figure 11.1 Roughing (left) and finishing (right) cuts.

Turning to Shapes

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

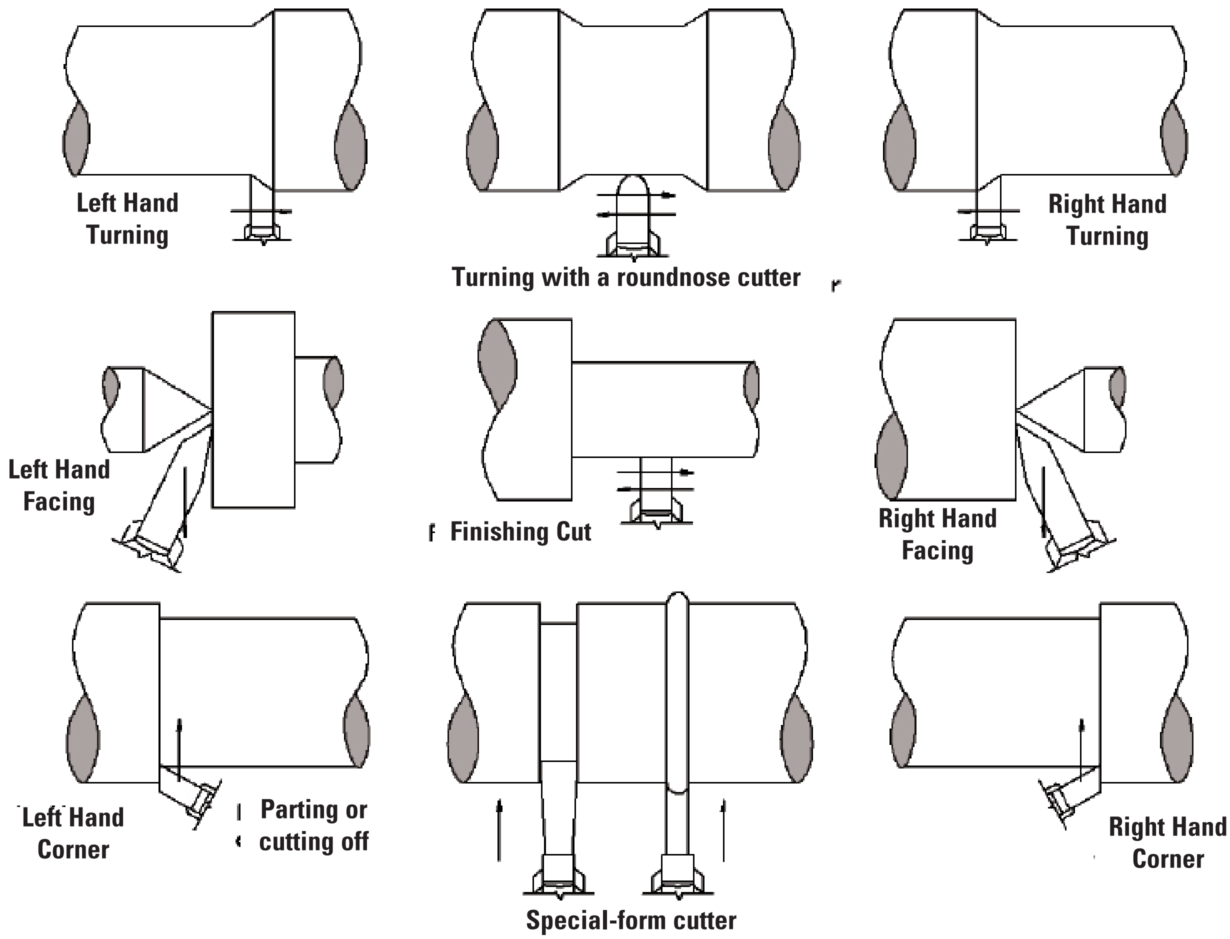


Figure 11.2 You can do other turning cuts with different cutter bits and cutting tools.

Machining Square Corners

To machine an accurate corner, follow these steps:

1. Set the compound rest perpendicular to the line of the centers and insert a right or left-hand corner tool.

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

3. 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.

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 work-piece; instead, lift it clear for each return stroke. Use a clean, dry file and keep the workpiece clean, as well. Wipe the workpiece dry and clean if you've used coolant or cutting oil. Never hold the file stationary while the workpiece is revolving.

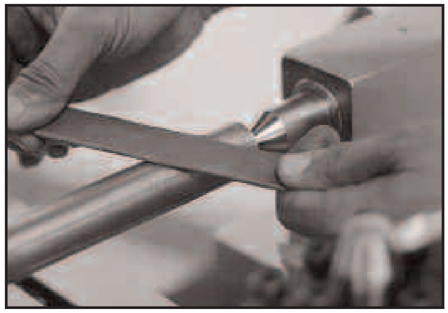


Figure 11.3 With a file, take full strokes at an oblique angle; never hold the file still.

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.

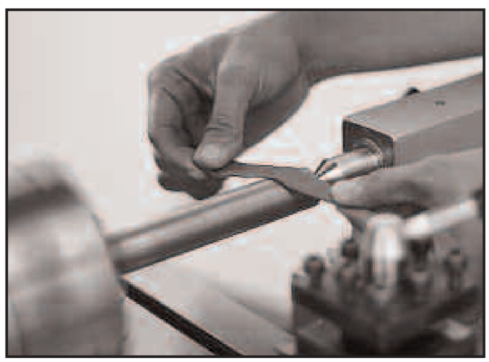


Figure 11.4 You can polish a workpiece with an abrasive cloth and oil.

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 11.5). 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 11.6). 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, loosen the two base-locking bolts (Figure 4.8). 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.

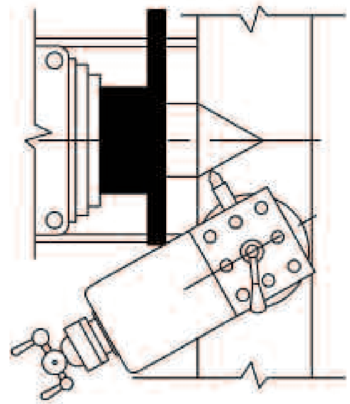


Figure 11.5 Tapers cut with the compound rest are usually short, abrupt angles.

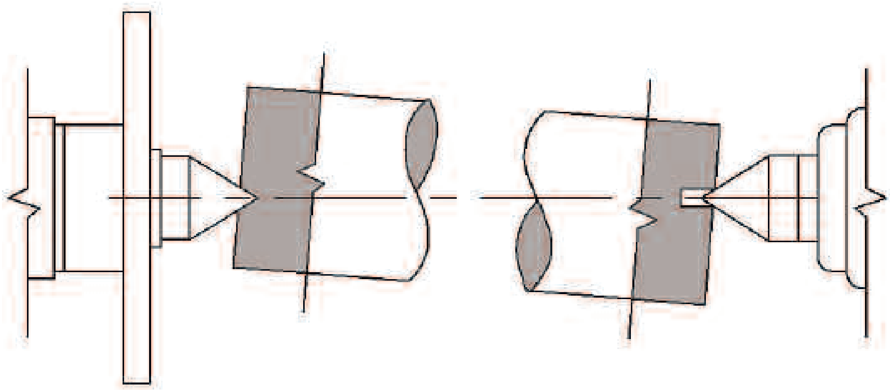


Figure 11.6 In setting over the tailstock, the lathe centers move from their parallel position with the tool's traverse travel.

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

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

• The differential between the parallel positions with the tool's traverse travel

• 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 11.7). 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.

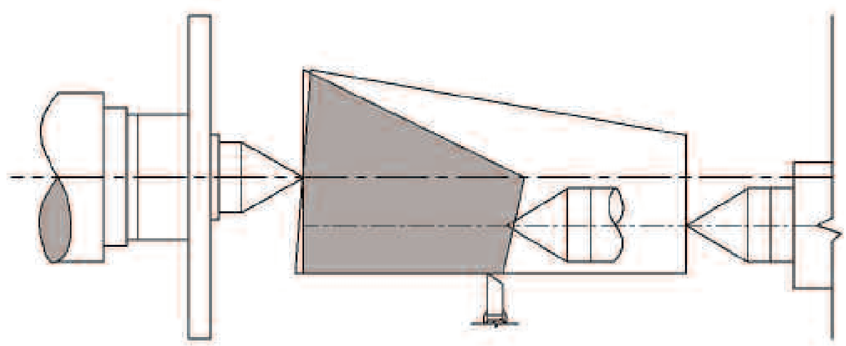


Figure 11.7 Tailstock setover should be half the difference between the finished diameters of the ends, or 0=T" x L"/2, where T= taper per inch and L= length of work in inches.

Note: (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.

Chapter 12

Lathe Facing and Knurling

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

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

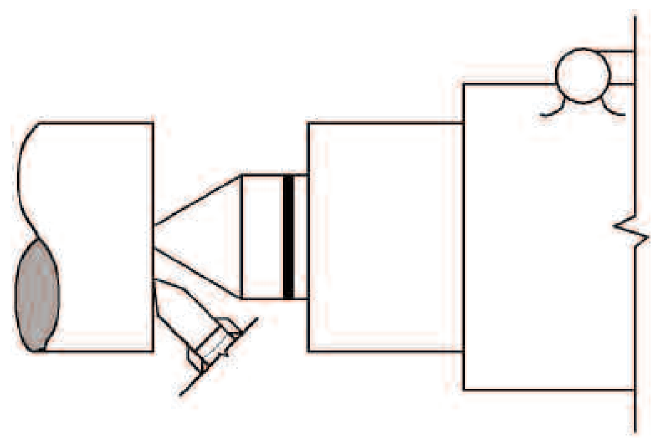


Figure 12.1 With a facing cutter, feed the tool from the center outward.

Facing Across the Chuck

When facing a stub-end workpiece held in the headstock 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.

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 12.2). This lets you extend the tool across the entire face of the work.

To use the powerfeed for facing, place the speed selector into the desired position before the lathe is turned on. Once the cutter has been positioned as per the above paragraph, move the crossfeed lever down. Pull the lever up at the end of the cut to stop the cutter travel.

Caution

Remember caution must be taken to not run the powerfeed past their limits of travel. As part of the normal operation, procedures, run each axis through the entire length of the proposed machining operation before engaging the powerfeed to assure there is sufficient travel to accomplish for the desired task. Failure to so could result in running the power feed to the end of its mechanical limit. This is what is known as a "CRASH". A crash can cause damage to the work piece and severe damage to the machine.

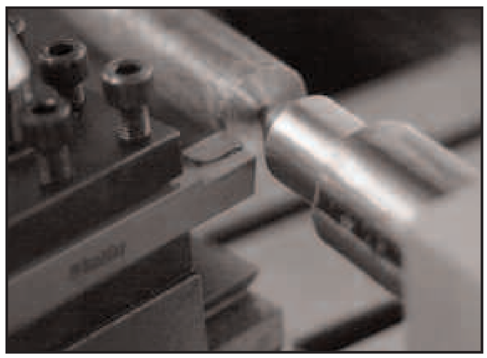


Figure 12.2 With a half-center you can extend the tool across the entire face of the work.

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 uniform pressure on both knurls. Select a self-centering knurling tool that equalizes pressure on the knurls automatically and is strong enough to withstand end and side thrusts. Operate the lathe at the slowest speed (160 rpm).

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

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

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

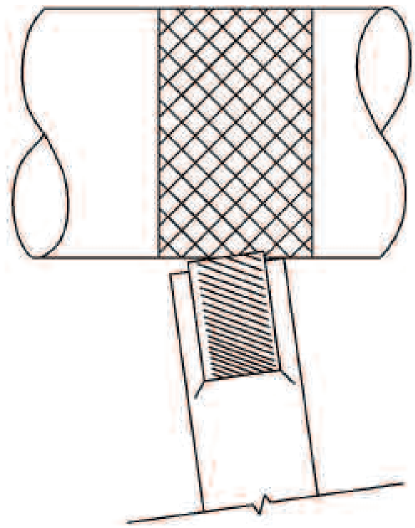


Figure 12.3 Feed the knurling tool at a slight angle off from perpendicular to the line of the work piece.

Chapter 13

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 cutoff blades (Figure 13.1).

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 work-piece, just beyond its center. The tool should feed to the workpiece on exact center, slowly and evenly with the cross-feed. 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.

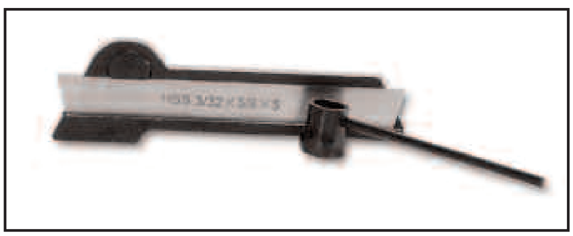


Figure 13.1 Specially designed tools like this one make cutting off easier.

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.

Chapter 14

Lathe Drilling and Boring

You can lathe drill on the MI-1220 LTD 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 14.1).

Without changing setup and re-centering, the work is ready for any succeeding operations, such as boring and internal threading. In all lathe drilling operations, keep the drill sharp and properly ground. This is essential for obtaining a straight, accurate hole.

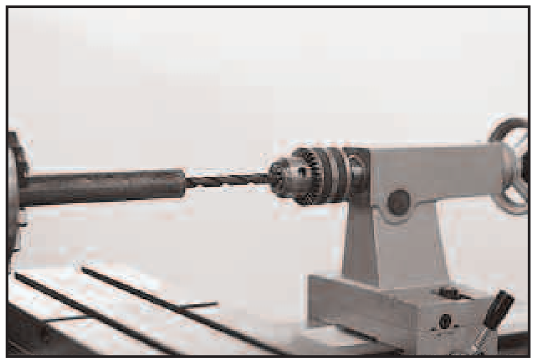


Figure 14.1 Holding the drill stationary in the tailstock chuck gives a straighter hole.

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 countersunk center for large diameters, to keep the drill from creeping.

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 solid reamers 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.

Boring

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

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

It is also wise to select tools with smooth-ended bars without a projecting nut or hardened edge that might mar the work (Figure 14.2). 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.

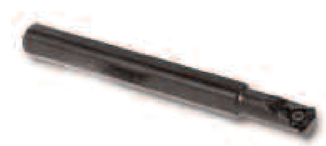


Figure14.2 A tool with a smooth-ended bar won't mark the workpiece.

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 out a starting cut in the opening of the hole to be bored with a turning tool before introducing the boring tool.

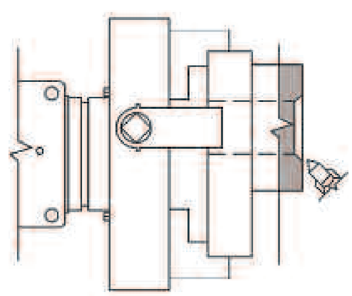


Figure 14.3 Chamfer a starting cut in the opening of the hole.

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 holes 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° from the radius at the cutting point (Figure 14.4). This takes into consideration the ground angle (top rake) of the cutter itself.

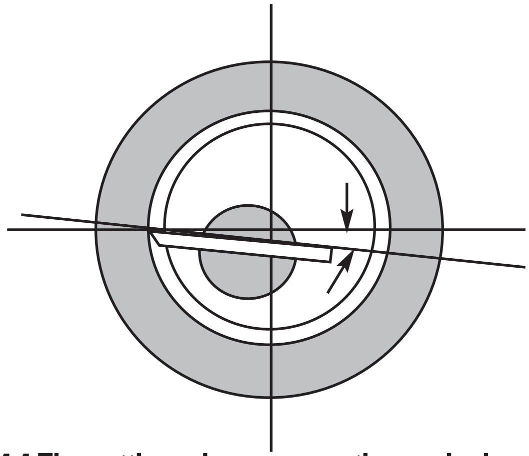


Figure 14.4 The cutting edge engages the work piece along a line in the mounted plane of the lathe centers

For straight longitudinal cuts, you can hold the cutter close up, therefore more rigidly, if it's at a 90° angle to the bar. For machining ends of a bar, however, you need a boring bar that holds the cutter at an angle or angles so the cutter extends beyond the end of the bar (Figure 14.5). 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.

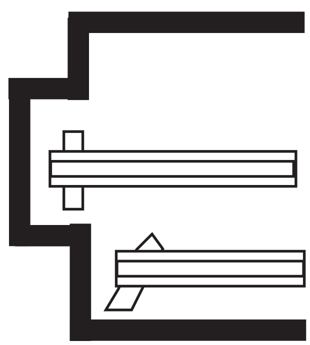


Figure 14.5 To machine ends of a bar, use a boring bar that angles the cutter so it extends beyond the bar

After the last finish cut, it is common to reverse the feed and take one last, fine cut with the tool coming out of the work. This last cut, taken without movement of the cross-feed, avoids a slightly undersized hole because you compensate for any spring in the bar.

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 14.6) 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 for different thread types. the cutter lifts clear of the thread for removal (Figure 14.7). Before cutting an internal thread, bore the workpiece to the exact inside diameter.

Figure 14.6 Use a center or thread gauge to correct cutter alignment error when squaring the cutter with the workpiece.

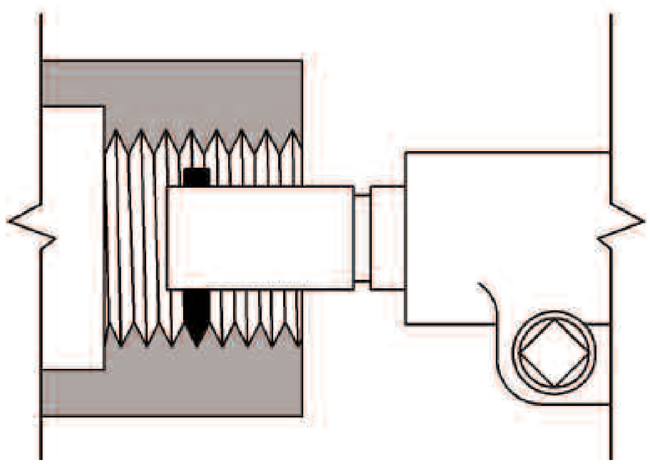


Figure 14.7 There must be enough end clearance for the cutter to lift clear of the thread.

Because the feed of successive cuts is toward, not away from, the operator, the thread-cutting set is reversed. 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.

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 14.8). If you don't have recommended clearances, it is safe to cut a nut thread (internal thread) 0.005" to 0.010" per inch larger in the screws outside diameter.

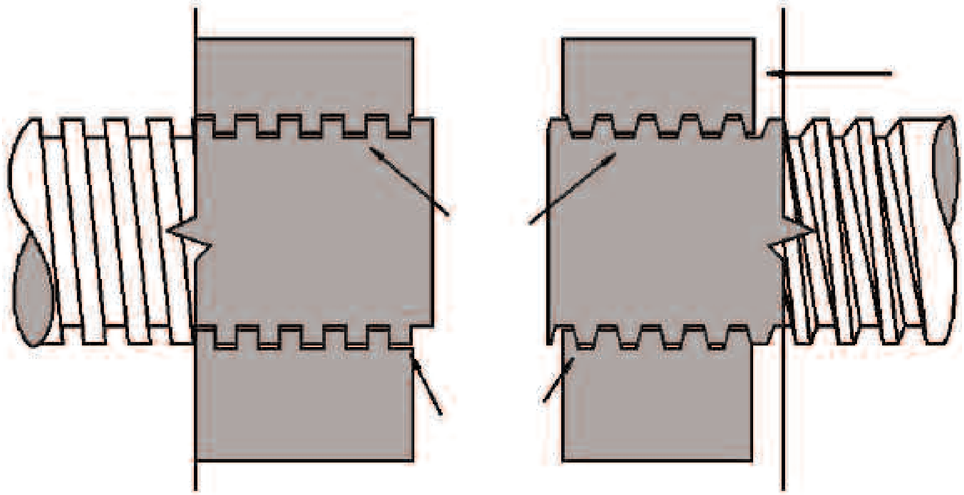


Figure 14.8 Use different clearances between nut and screw for different thread types.

Chapter 15

Changing Gears on Your MI-1220 LTD

To change gears on the MI-1220 LTD follow these steps.

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

1. 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 and C gear shaft in its bracket by turning the gear shaft counterclockwise (Figure 13.1). This allows the shaft to slide freely along the bracket for easy gear removal and replacement.

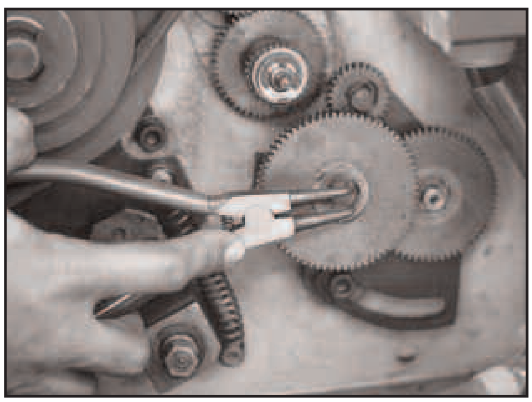


Figure 15.1 Remove all C clips, nuts, and gears

2. Select the proper A-D gear combination from the list outside the pulley box door.

3. Use the Allen wrench to loosen the bolt at the bottom of the bracket assembly for full swing and easy gear replacement (Figure 13.1).

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

5. Place the selected C gear, flange side in, on the B and C gear shaft.

6. Place the selected B gear, flange side in, on the B and C gear shaft. Replace the C clip.

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

Figure 15.2 Slide the B and C gear shaft until the C gear meshes with the D gear

8. Place the selected A gear, flange side in, on the A gear shaft and replace the C clip.

9. 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.

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

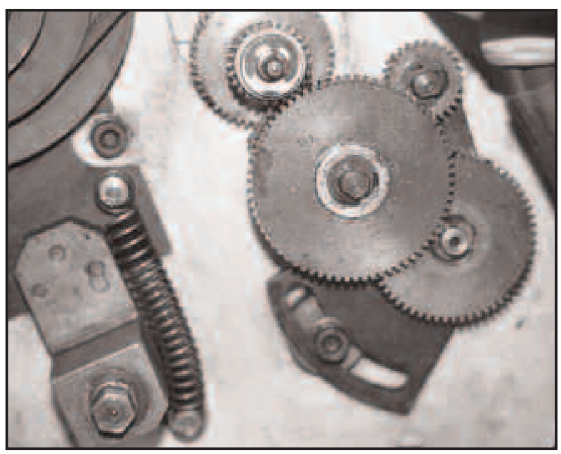


Figure 15.3 Engage the E gear between the C and D gears to reverse the leadscrew.

Chapter 16

Cutting Threads on Your MI-1220 LTD

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

Built for thread cutting, the MI-1220 LTD 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.

For most work, beginners use the Unified National Standard, which is a V-form thread slightly flat on top and at the root. Pitch numbers, such as 18 or 24, usually refers to screw threads meaning 18 or 24 threads per inch (TPI).

Thread charts on the inside of the door of your machine and shows both inch and metric measurements. The inch chart shows the TPI from 6 to 120. The metric shows the distance from thread crest to crest from 0.50 to 4 mm.

For right-hand threads, start the threading or chasing 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.

To make sure your cutter is on dead center, place a credit card or shim between the cutter point and workpiece (Figure 16.1). When the tool is on dead center, the credit 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.

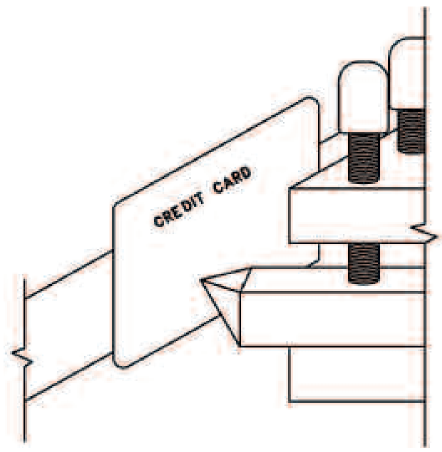


Figure 16.1 Check dead center with a credit card.

Set the compound perpendicular to the line of centers and rotate it 29-1/2° to the right .

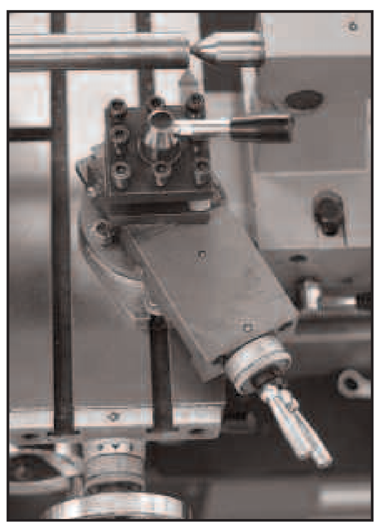


Figure 16.2 With the compound perpendicular to the line of centers, rotate it 29.5 degrees to the right.

Place the thread gauge on the point of the threading tool and feed the tool toward the work-piece (Figure 16.3). 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.

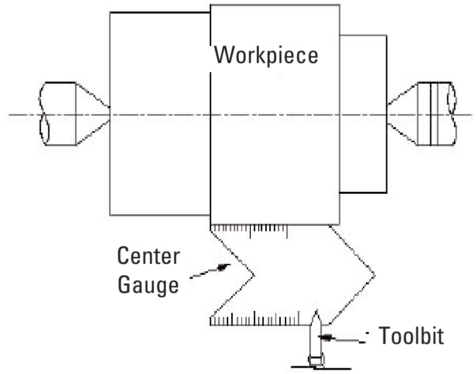


Figure 16.3 Using a center gauge, set the threading tool perpendicular to the work piece.

Cutting Right-hand Threads

Place the leadscrew selector in position to feed the cutter from right to left, toward the headstock. 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 halfnut lever carefully pushing the handle down. 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.

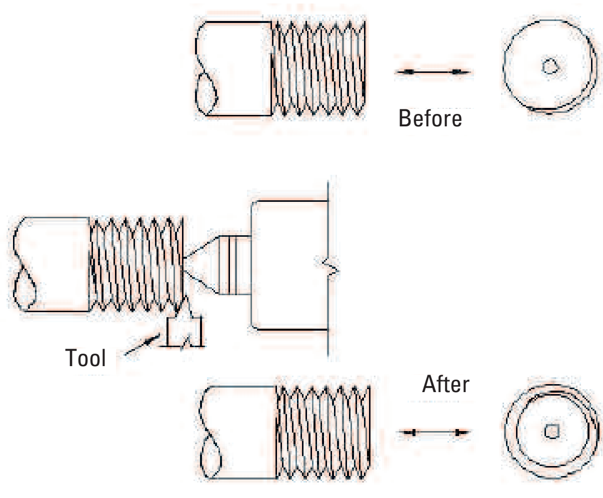


Figure 16.4 Chamfer the end of the thread to protect it from damage.

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

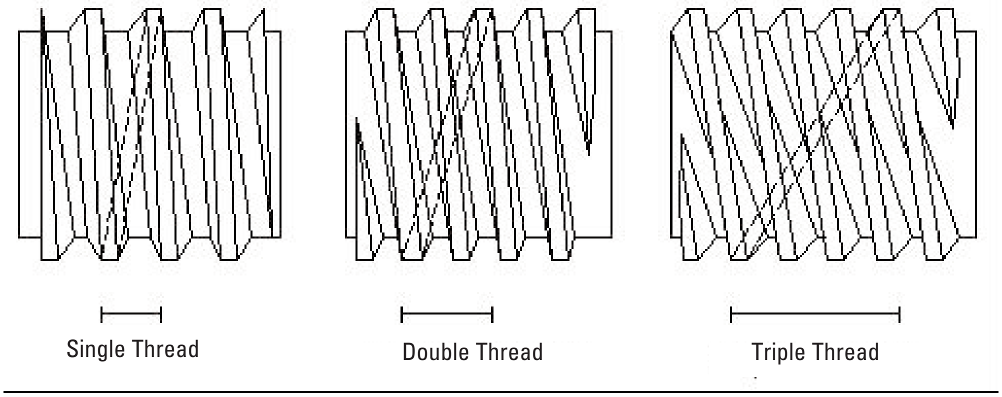


Figure 16.5 When cutting multiple threads, increase the lead to make room for succeeding threads.

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 piece in the chuck and not remove it for testing.

After returning the workpiece to the setup, continue taking 0.001-0.002" cuts. Then check the fit between each cut. 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 to protect it from damage. To chamfer is to take a 45° cut off the end of the bolt.

Using the Threading Dial

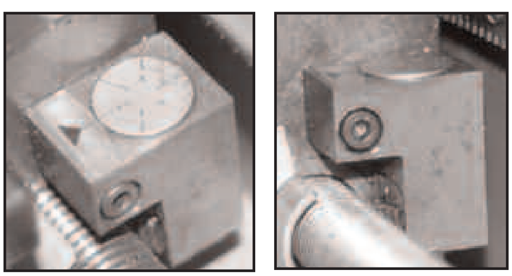


Figure 16.6 Threading dial

The threading dial performs the function of indicating the proper time to engage the half-nut so that the cutting tool will enter the same groove of the thread of each successive cutting pass. This allows the half-nut to be disengaged resulting in an easier method of threading. At the end for each consecutive pass the half-nut can be disengaged and the carriage can be moved back to the start of the thread without stopping the spindle or reversing the motor. Disengage the half-nut, back the cutter away from the workpiece and move the cutter back to the beginning of the thread with the hand wheel. Turn the cutter the desired amount and you are ready for the next threading pass. The threading dial is marked with lines numbered 1,2,3,4,5 and 6 and a single reference line on the housing of the dial. The indicator table shows the selection for the different thread pitches. Find the desired thread pitch under the "TPI" column and engage the half-nut at the proper numbers shown on the "SCALE" column of the table. "1-6" means the half-nut can be engaged on any of the numbered lines 1 through 6. "1,4" means that the half nut can be engage on 1 or 4 only. "1" indicates that the half-nut can be engaged on 1 only. For any thread pitch not listed on the chart, cut a test thread and engage the half nut on 1 or 4 only. If the test cut is not successful, the thread dial annot be used and the instructions from the previous section should be followed. Cutting of metric threads cannot be done with the thread dial.

TPI	SCALE	TPI	SCALE	TPI	SCALE	TPI	SCALE	TPI	SCALE
6	1,4	10	1-6	14	1,4	22	1,4	30	1-6
7	1	11	1	16	1,4	24	1,4		
8	1,4	12	1,4	18	1-6	26	1,4		
9	1-6	13	1	20	1-6	28	1,4		

Table 16.1 Indicator Scale

Cutting Multiple Threads

Cut multiple threads 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.

What Not To Do When Cutting Threads

Do not disengage the powerfeed direction 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 workpiece from the center, leaving the lathe dog attached, then test the thread. If it does not fit properly and you have to remove another chip or two, place the woprkpiece back in the centers exactly as it had been, then remove the chips and test again. Repeat until finished.

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.

Cutting Threads on a Taper

Cut threads on a taper the same as on a straight shaft, except in the setup of the tool. Set the threading tool at 90° to the axis of the taper, rather than at 90° to its surface (Figure 16.7).

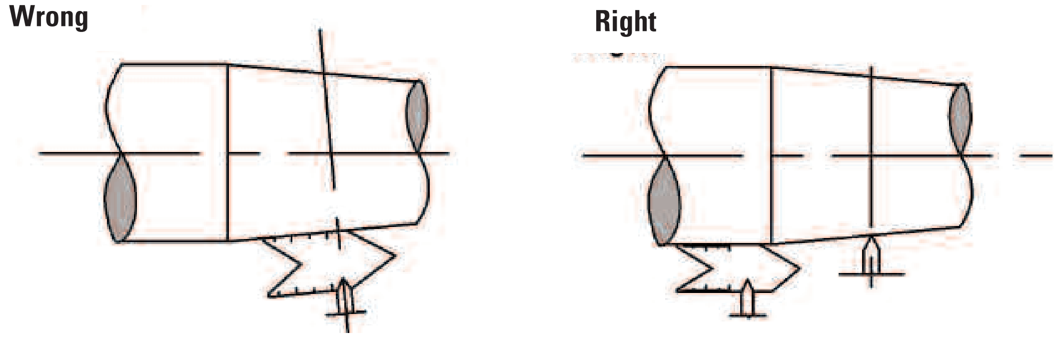


Figure 16.7 When cutting a thread on a taper, set the threading tool at (missing text)

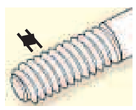
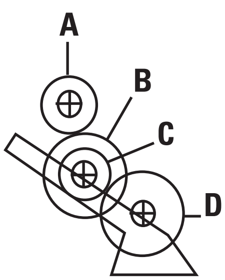
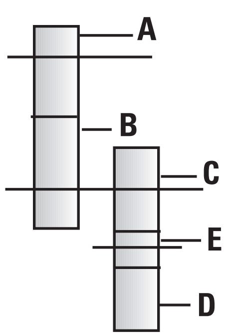
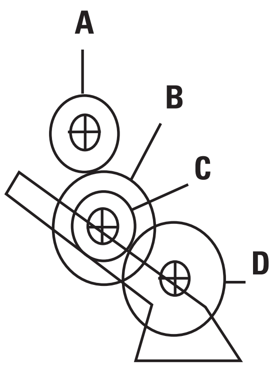
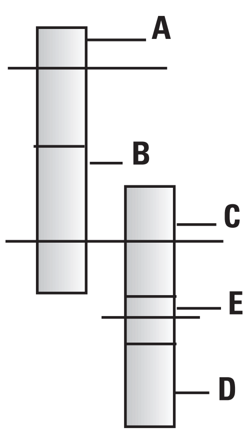
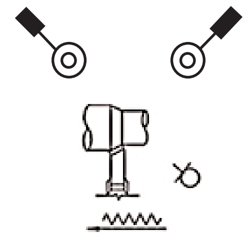
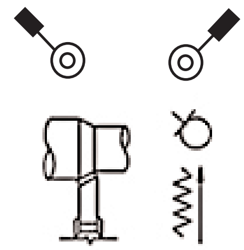
	INCH THREADS			INCH THREADS			INCH THREADS		
	I 0 II	I 0 II	A X C B D 27 X 27 70 60	I 0 II	I 0 II	A X C B D 27 X 27 70 60	I 0 II	I 0 II	A X C B D 49 X 42 32 56
MM 	30	15	70 X 40 32 45	32	16	70 X 40 32 48	0.50	1.00	49 X 42 32 56
	28	14	70 X 40 32 42	36*	18*	70 X 40 32 54	0.60	1.20	49 X 45 32 50
	26	13	70 X 40 32 39	40*	20*	70 X 32 32 48		1.25	45 X 49 48 32
	24	12	70 X 40 32 36	48*	24*	70 X 30 36 48	0.70		49 X 42 32 40
	22	11	70 X 40 32 33	56*	28*	42 X 50 36 56	0.75	1.50	63 X 42 32 48
	20	10	70 X 40 32 30	60*	30*	70 X 30 48 54	0.80		49 X 60 50 32
	18	9	70 X 45 27 36	64*	32*	70 X 30 48 60		1.75	49 X 63 48 32
	16	8	70 X 45 27 32	80*	40*	70 X 30 48 60	1.00	2.00	63 X 49 32 42
	14	7	60 X 60 27 32	96**	48**	39 X 45 60 48	1.25	2.50	60 X 49 32 32
	12	6	70 X 60 27 32	120	60	42 X 30 54 48	1.50	3.00	63 X 56 32 32
				333	167	30 X 27 60 63	2.00	4.00	70 X 63 30 32

Table 16.2 Threading Chart for the MI-1220 LTD





A X C / B D	I 0 II	I 0 II	I 0 II	I 0 II
27 X 27 / 70 60	0.0011	0.0022	0.001	0.0003
70 X 40 / 32 45	0.0126	0.0252	0.0015	0.0030
70 X 40 / 32 42	0.0135	0.0270	0.0016	0.0032
70 X 40 / 32 39	0.0145	0.0290	0.0017	0.0034
70 X 40 / 32 36	0.0157	0.0314	0.0018	0.0037
70 X 40 / 32 33	0.0171	0.0343	0.0020	0.0041
70 X 40 / 32 30	0.0189	0.0377	0.0022	0.0044
70 X 45 / 27 36	0.0210	0.0149	0.0025	0.0049
70 X 45 / 27 32	0.0236	0.0471	0.0028	0.0055
60 X 60 / 27 32	0.0269	0.0529	0.0032	0.0063
70 X 60 / 27 32	0.0314	0.0628	0.0037	0.0074

Table 16.3 Feed Rates for the MI-1220 LTD

Chapter 17

Milling

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

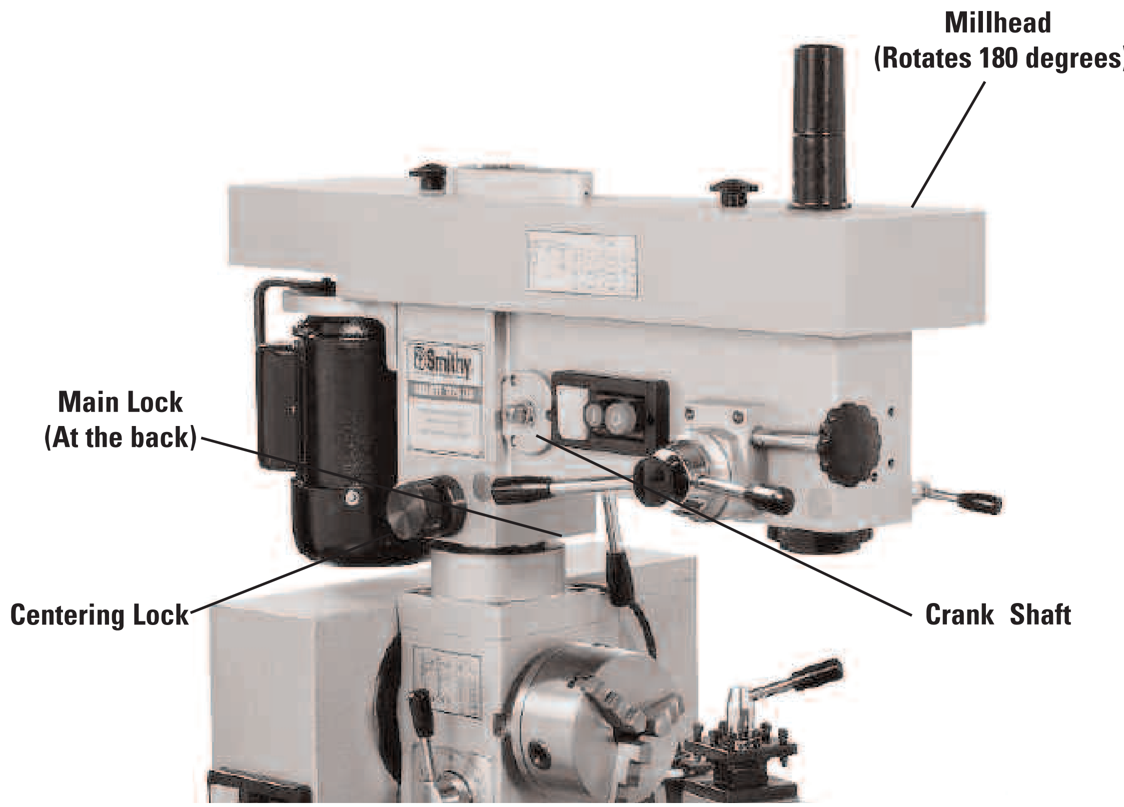


Figure 17.1 MI-1220 LTD Milling/Drilling parts

Machinists use mills to machine flat surfaces, both horizontal and vertical, and to make shoulders, grooves, fillets, keyways, T-slots, and dovetails. They can also make curved and irregular surfaces and machine accurate holes. Its variety of machining operations and high metal-removal rates rank the mill in importance with the lathe.
The millhead rotates 180 degrees and adjusts up and down. 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 long-feed handwheels 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 rotate the mill head, loosen the centering lock on the front of the mill head and the mail lock on the back.

Push the mil head in the desired direction. Lock the main and centering locks to hold the head into position

To recenter the mill head, push the head into the approximate centered position. Move the head back and fourth slightly while tightening the centering lock. The head will work its way into the centered position.

The mill head may be raised to accomodate larger work pieces. Unlock both locks and slide the crank handle over the square ended shaft on the front of the mill head. Crank to the desired height and lock into position.

Holding Milling Cutters

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

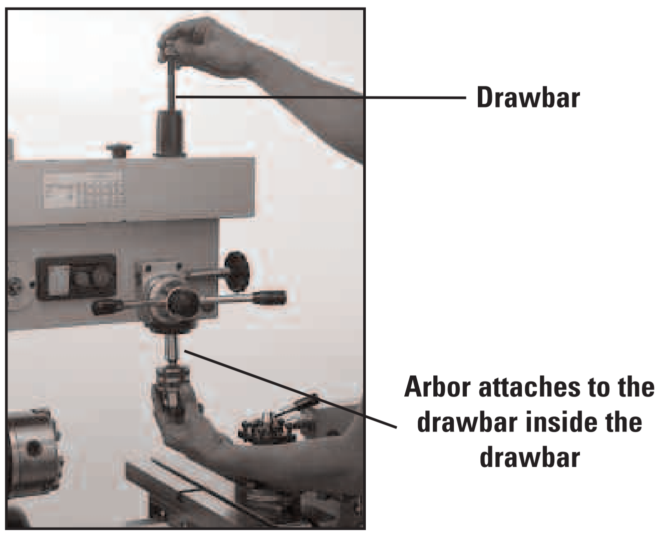


Figure 17.2 MI-1220 LTD milling attachments

Arbors

Arbors come in different sizes and lengths, with one end tapered to fit the bore in the end of the machine spindle. The arbor of theCB1220 XL LTD, which has an MT3 taper, is driven by the friction between the arbor and spindle. The arbor stays in place by means of a drawbar screwed into the end of the arbor from the top of the spindle.
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.

Collets and Holders

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

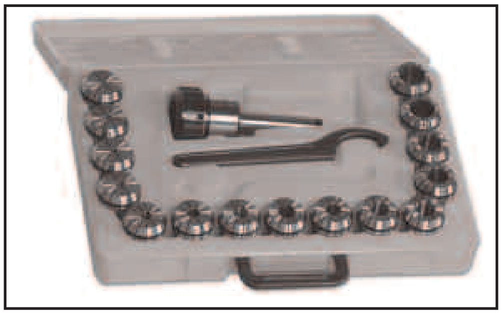


Figure 17.3 Spring collets, which fit into the mill spindle, hold straight-shanked end mills.

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:

1. Loosen the locknut on the drawbar about two turns.

2. Hit the end of the drawbar with a dead blow hammer, releasing the arbor or adapter from the spindle hole.

3. Hold the arbor or adapter so it won't fall out of the spindle when the drawbar is removed.

4. Unscrew the drawbar and remove the arbor or adapter.

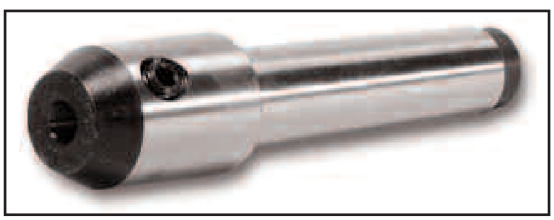


Figure 17.4 End mill holders also receive straight-shanked end mills.

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

1. Remove the drawbar.

2. Extend the mill spindle to expose the outer taper drift slot.

3. 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.

4. Insert the drift in the slot.

5. Holding the adapter with one hand, use a non-marring 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.

Miling 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.

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*** 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).

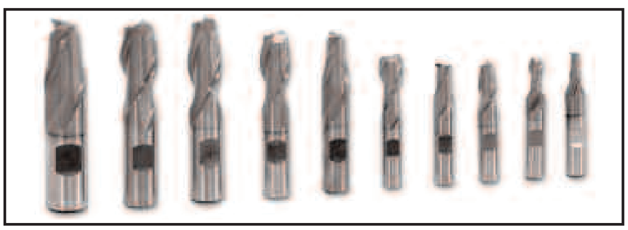


Figure 17.5 Two-flute end mills have two teeth that cut the center of the mill

• ***Multiple flute end mills*** have three, four, six, or eight flutes and may be single or double-ended. Multiple-flute mills are center cutting or non-center cutting. Don't use non-center 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, key seat cutters, and shell end mills.

• ***Ball end mills*** (Figure 17.6) cut slots or fillets with a radius bottom, round out pockets and bottoms of holes, and do die sinking and die making. Four-fluted ball end mills with center cutting lips are available.

Figure 17.6 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 pro-vide 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*** which mill wide, flat surfaces, have a hole for mounting on a short arbor. The center of the shell is recessed to provide space fro screw or nut that fastens the cutter to the arbor. The teeth are usually helical, and diameters are as large as 6”.

Figure 17.7 Shell end mills mill wide, flat surfaces and mount on arbors

• ***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 the periphery and face. Most of the cutting takes place on the periphery. They are similar to, but larger than, shell end mills.

Plain Milling Cutters

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

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

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

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

Side Milling Cutters

Similar to plain milling cutters, side milling cutters also have teeth on one or both sides. 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.

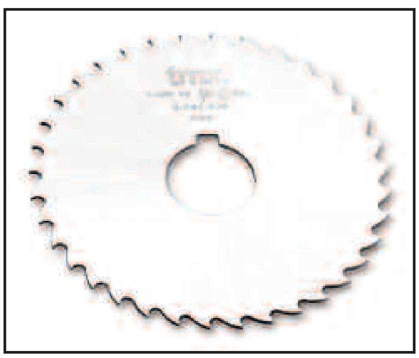


Figure 17.8 Side milling cutters are similar to plain milling cutter, but they have teeth on one or both sides

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

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

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

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 screw heads. Their sides are straight and parallel and offer no side relief.

Angle Milling Cutters

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

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

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

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.

Flycutters

With one or more single-point toolbits or cutters, flycutters (Figure 17.9) 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 (Section Seven).

You can also use flycutters for boring.

Note: When the tool revolves, the cutting tool becomes almost invisible, so be careful.

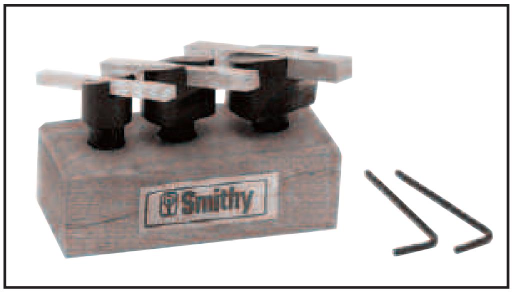


Figure 17.9 Flycutters take light face cuts from large surface areas.

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 20.6), and be careful to keep cast-iron dust from getting between the lathe and carriage ways.

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.

Speeds and Feeds for Milling

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. 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:

rpm = 12 x CS (fpm) / D" x it

where:
CS = cutting speed
fpm = feet per minute
D" = diameter of the cutter in inches, and
it = 3.14

For metric measurement, use this formula:

rpm = CS (mpm) x 1000 / D (mm) x it

where:
CS = cutting speed
mpm = meters per minute
D (mm) = diameter of the cutter in millimeters,
it = 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 5.3.

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. 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 how you should perform milling operations.

Down milling

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

Feed rates. Your feed rate should be as high as your machine, cutting tool, workholding method, and workpiece can tolerate while giving a good finish. Feed rate is usually given in inches per minute (ipm). You determine feed rate by the speed of the cutter in rpm and the number of teeth in the cutter.

There are many factors to consider in selecting the feed per tooth, and there is no easy formula to follow. Here are several principles to guide you:

• Use the highest feed rate conditions allow

• Avoid using a feed rate below 0.001" per tooth

• Harder materials require lower feed rates than softer materials

• Feed wider, deeper cuts more slowly than narrow, shallow cuts

• Slower feed rates gives a better surface finish

• Never stop the feed before finishing the cut.

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

ipm = ipt 5 N 5 rpm

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

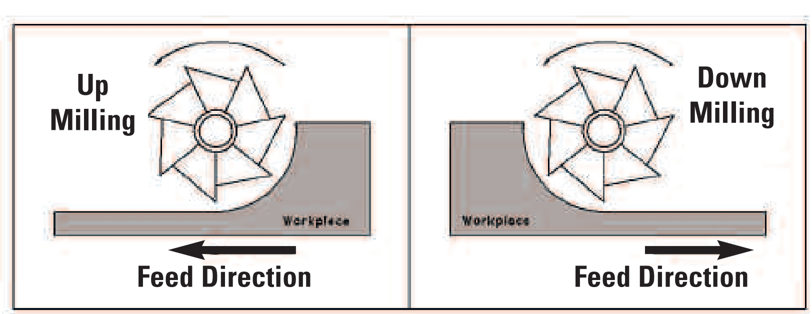


Figure 17.10 In up milling (left), the work piece feeds into the cutter in the opposite direction of the cutter's revolutions. In down milling (right), the work piece feeds into the cutter in the same direction as the cutter is turning.

Material		Brinell Hardness	High-Speed-Steel Cutters	Carbide Cutters
Free-machining low carbon steel resulphurized	1111 1112	100-150 150-200	120-160 120-180	400-600 400-900
Free-machining low carbon steel leaded	10L18 12L14	100-150 150-220	100-225 110-250	250-500 250-600
Plain low-carbon steels	1006 1026	100-125 125-175	80-150 80-140	300-600 250-500
Plain medium-carbon steels	1030 1095	125-175 175-225	80-140 60-110	250-500 225-400
Plain high-carbon steels	1060 1095	125-175 175-225	70-120 60-110	250-450 225-400
Tool Steels	W1-W7 H20-H43 D1-D7	150-200 200-250 200-250	80-120 40-85 30-60	300-350 175-300 100-200
Stainless Steel	302 430F	135-185 135-185	70-100 100-140	225-350 350-450
Gray Cast Iron	ASTM Class 20 Through Scale Under Scale	110-160	140-200 130-225	350-700 400-800
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

Table 17.1 Recommended Cutting Speeds for Milling (fpm)

Common Milling Operations

Milling Flat Surfaces

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

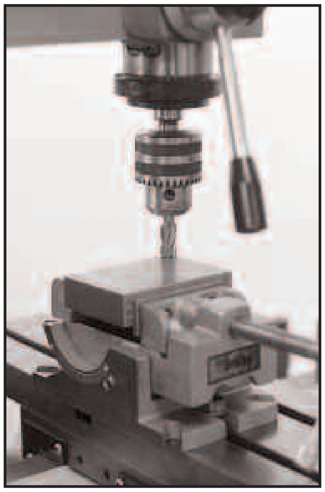


Figure 17.11 One way to mill a flat surface is by plane milling

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 face mill large surfaces.

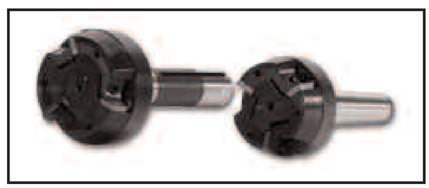


Figure 17.12 Inserted tooth face milling cutters face mill large surfaces

Bevels and chamfers are cut at an angle to the main work-piece surface. A bevel cut (Figure 17.13) goes from side to side, completely removing the perpendicular edge.
A chamfer removes only part of the perpendicular edge.

Figure 17.3 A bevel cut goes from side to side, completely removing 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 in a fixture held in a vise.

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.

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.

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.

Chapter 18

Workholding

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

NO IMAGE. NO AVAILABLE ACCESSORY FOR SET-UP.

Figure 18.1 Workholding set up (Missing)

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. Another method is to use the face of the spindle plate, chuck or taistock as a quick reference surface.

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.

Dividing Heads

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 18.2). An indexing plate is set in from the handle. By counting how many turns of the handle it takes to turn the workpiece a certain number of degrees, you can make cuts at different angles. This is how to cut gears.

NO IMAGE. NO AVAILABLE ACCESSORY FOR SET-UP.

Figure 18.2 Dividing head set up (Missing)

Rotary Tables

A rotary table 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 work piece. A typical rotary table is graduated in degrees and fractions.

The index plate in 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.

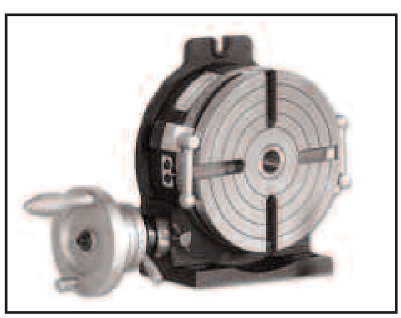


Figure 18.3 Rotary tables let you cut gears, precision holes, and curved slots.

Chapter 19

Troubleshooting

Powerfeed and Thread Cutting

Powerfeed does not move carriage

Cause	Solution
• Carriage locked	• Unlock carriage
• Speed selector not engaged	• Select speed I or II
• Leadscrew lever not engaged	• Move lever to the left or right
• Gears not meshing or teeth missing	• Check gears and mesh
• Half-nut not fully engaged for threading	• Keep half nut engaged for threading
• Long feed not engaged	• Move longitudinal feed lever down

Cut is not smooth

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

Thread is not smooth

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

Tool is not cutting “on thread”

Cause	Solution
• Half nut not engaged at proper time	• Check chart

Carriage/Milling Table

Powerfeed doesn't move table

Cause	Solution
• Carriage	• Unlock Carriage
• Speed Selector not engaged	• Select Speed I or 11
• Leadscrew lever not engaged	• Move lever to the left or right
• Gear not meshing or teeth missing	• Check and adjust gears
• Crossfeed not engaged	• Move crossfeed lever down

Horizontal movement in cross-slide table

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

Vertical movement in cross-slide table

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

Carriage moves smoothly in only one direction

Cause	Solution
• Debris on way or gib	• Remove debris
• Burr on gib	• Remove burr with fine file
• Gib improperly tensioned	• Loosen gib and re-tension

Cross-slide handwheel turns during cutting operations

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

Too much backlash in the cross slide

Cause	Solution
• Loose screw holding crossfeed nut	• Tighten screw,
• Worn brass nut	• Replace nut
• Loose spanner nuts	• Adjsut Spanner Nuts
• Loose Brass Nut	• Put shim between the stud on the nut and side of the hole

Cause	Solution
• Too much space between bearing and dial	• Add shim washers

Lathe Turning

Cut is rough

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

Work has unwanted taper

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

Machine vibrates

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

Work stops turning but machine continues to run

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

Diameter of work is not consistent

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

Too much backlash in compound

Cause	Solution
• Loose spanner nuts	• Tighten Spanner Nuts
• Worn nut	• Replace nut

Machine slings oil from behind the chuck or in belt box

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

Milling

Tool chatters

Cause	Solution
• Gibs too Toole on cross slide, compound or carriage	• Readjsut Gibs
• Unused feeds not locked	• Lock all axes but the one moving
• Milhead not locked	• Lock millhead
• Quill too loose	• Tighten guill lock
• Tool not on center	• Center Tool
• Improper tool shape, too dull	• Reshape, sharpen, or replace tool

Deph of cut is not consistent

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

Drilling

Hole is off center or bit wandres

Cause	Solution
• Bit Dull	• Use Sharp Bit
• Bit not mounted correctly in check	• Remount Tool
• Bit Bent	• Replace Bit
• Chuck loose in spindle	• Remount chuck and arbor and remount
• Drawbar not secured	• Tighten Drawbar
• Debris on Spindle	• Clean debris and arbor and remount tool

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

Entrance hole is out of round

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

Bit turns erratically or stops

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

Chuck is difficult to tighten or loosen

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

Chuck wobbles

Cause	Solution
• Chuck loose on arbor	• Clean arbor and remount
• Drawbar not tight	• Clean spindle and replace drawbar

Drive System

Turn on machine and nothing happens

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

Chapter 20

Removing the Quill and Quill Feed Assembly

Caution

Have the owner's manual available when doing any machine maintenance. The items referenced in these instructions can be found in the parts section of the owner's manual.

MAKE SURE THE MACHINE IS UNPLUGGED BEFORE STARTING ANY MAINTENANCE PROCEDURES.

Note: There are hidden screw located under the feed dial, it will be necessary to remove the dial to get to the screw.

1. Release the tension on the quill retract spring by loosening the setscrew (109) that comes in underneath the spring housing. Be sure the quill is retracted into the millhead completely, and to hold on to the spring housing with a rag or a glove so it does not spin suddenly. Then remove the setscrew 110 on the dial side, opposite setscrew 109.

2. Unscrew and remove the spring cap (65).

3. The outer part of the spring is still under tension, and can be dangerous if pulled out of the housing. Now you will need to unhook the inner part of the spring from the screw (108) on the shaft (107). Use needle nose pliers and a flathead screwdriver while rotating the shaft to unhook the spring.

4. Remove the screw (108) with needle nose pliers from the side, and slide the spring housing out of the casting while making sure the spring stays inside the housing. * Please see note below.

5. Pull knob (91) outward to disengage the fine feed and remove the knob.

6. Remove setscrew (96) from the handle seat (89), then turn the feed handles until setscrew (86), spring (87), and ball (88) are facing down. Lock the quill in place and remove the setscrew (86).

7. Tap the dial with a non-metallic mallet to dislodge the spring and ball.

8. Use two screwdrivers between the dial face (98) and the feed housing (101) to pry the dial off the shaft.

9. Remove the 3 screws (99 and 100) and pull the feed assembly put of the casting.

10. Remove the setscrew (69) from the casting, support the quill with one hand and release the quill lock. Lower the quill out of the casting.

Assembly

1. Basic assembly is the reversal of the above steps.

2. It is important to make sure all parts are clean and properly lubricated where needed.

3. A thin coating of light grease should be applied to any sliding or rotating surfaces before assembly.

Note: If you should need to remove the spring, the easiest way to rewind the old spring is to make a mandrill from a piece of round stock about 1/2" in diameter. Drill and tap a hole in the rod for a screw similar to the end of the shaft (107) on the machine where the spring is to be installed. Put the mandrill in the lathe chuck and slip the spring over the screw head. Drill and tap a hole in the side of a lathe tool and install a screw similar in size to the screw in the spring housing. Put the tool in the tool post in a manner that the screw head can be hooked to the end of the spring. Slowly turn the lathe spindle by hand as you also feed the tool post inward to keep pace with the spring as it winds up and gets smaller and smaller in diameter. When the spring is wound to a size smaller than the spring housing, wrap wire around the housing to hold it wound while you remove it from the mandrill and install it into the housing.

Chapter 21

MI-1220 LTD Full Specifications

General Dimensions

Length	53-1/2″
Width	20″
Height	37″
Shipping Weight	480 lbs
Machine Weight	397 lbs
Crate Size	57-1/2″ x 22-3/4″ x 38″
Footprint	54-1/2″ x 32″
T-Slot Size	7/16″
Accuracy	+/- 0.001″
Powerfeed (X-Axis)	Yes
Powerfeed (Y-Axis)	Yes
Powerfeed (Z-Axis)	No
Table Size	5-7/8″ x 16-3/4″
Threading Dial	Yes

Lathe Specifications

Distance Between Centers	20″
Dial Calibration on Crossfeed	.002″
Dial Calibration on Toolpost	.002″
Dial Calibration on Leadscrew	.002″
Dial Calibration on Longfeed Rack	.02″
Dial Calibration on Tailstock	.002″
Feed Rates	.0011-.020 (Y-axis) .0022-.040 (X-axis)
Headstock Taper	MT4
Lathe Chuck Bore	1.17″
Lathe Chuck Diameter	5″
Lathe Chuck - Max. diameter work piece	5″
Lathe Chuck - Min. diameter work piece	1/8″
Lathe Chuck Mount	Bolt-On
Lathe Chuck Type	3 Jaw Self Centering
Spindle Bore	1.03″
Spindle Speeds	Six (160-1600 RPM)
Swing Over Bed	12″
Swing Over Work Table	6-3/4″
Tailstock Offset	19/32″
Tailstock Taper	MT3
Tailstock Barrel Travel	2″
Threads-Inch	SAE 6-120 TPI

Threads-Metric	0.5 to 4 mm
Toolpost Travel	3-1/4″
Toolbit Size	1/2″
X-Axis Travel (w/tailstock installed)	26″
Y-Axis Travel	8-1/2″

Mill Specifications

Column Diameter	3-1/8″
Dial Calibration Drill-Coarse Feed	0.042″
Dial Calibration Mill-Fine Feed	0.042″
Drawbars Size (included)	12 mm, 3/8″
Drill Chuck Size (included)	1/2″
Drill Chuck Arbor Size (included)	MT3/JT33
Feed Rates	.0011-.020″ (Y-Axis) .0022-.040″ (X-Axis)
Head Rotation	180 Degrees
Head Travel	2-3/4″
Quill Diameter	2-3/4″
Quill Travel	3-1/8″
Spindle Center to Front of Chuck	8-3/4″
Spindle Center to Lathe Spindle Flange	11″
Spindle Center to Support Column	12-1/8″
Spindle to Table Distance (min-max)	6-1/4″-13″
Spindle Speeds	9 (315-2000 RPM)
Spindle Taper	MT3
Tool Size Limits	1″
X-Axis Travel	12-1/16″
Y-Axis Travel	8-1/2″

Electrical

Amperage	11 Amps Lathe & Mill
Horsepower	3/4 hp Lathe, 3/4 hp Mill
Motor Type	A/C
Phase	Single
Voltage	110 Volts A/C

Chapter 23

Machine Warranty

30 Day Trial Offer

Try a Smithy for 30 days. If, for any reason within that time, you decide to return your Smithy, just call our Customer Service department at 1-800-476-4849. We will help you arrange shipping back to us. When we receive the machine back, we'll refund your full purchase price. Please note: return shipping charges and any shipping damage from improper repacking is your responsibility.

Smithy Warranty

Smithy 3-in-1 and Dedicated Machines are warranted for two years (unless otherwise noted)to the original purchaser against defects in materials and workmanship. During that time, Smithy will replace any defective parts that are returned to our warehouse, free of charge. Upon receipt of the defective parts, Smithy technicians will arrange with you to send replacement parts immediately. This warranty does not cover parts that are worn out through the negligence on the part of the operator nor does it cover consequential damages resulting from defects in material or workmanship.

SmithyCNC warrants its machines and control systems for a period of one (1) year to the original purchaser from the date of purchase. If within one (1) year form the date of purchase a SmithyCNC machine and/or control system fails due to defect in material or workmanship, SmithyCNC will at their choice repair and/or replace components with new or remanufactured parts free of charge.

(Some have asked why SmithyCNC machines have a shorter warranty period than Smithy manual machines. There are several reasons, but the greatest factor is that, on average, CNC automated machine tools, are operated a   significantly greater number of hours per day than the average manual machine. Also, by comparison, most of our competitors selling benchtop CNC machines only offer a six (6) months warranty. Whereas SmithyCNC machine have a full one (1) year warranty.)

Most warranty repairs and/or replacements are handled routinely, but sometimes request for warranty service many not be appropriate. This warranty does not apply to defects due directly or indirectly to misuse, abuse, negligence, accidents, repairs, or lack of routine maintenance. This warranty is also void if the serial number of the machine or SmithyCNC control system has been removed or has been altered or modified.

In no event shall Smithy be liable for indirect, incidental or consequential damages for the sale or use of the product. This disclaimer applies to both during and after the term of this warranty.

We do not warrant or represent that the merchandise complies with the provisions of any law or acts unless Smithy Co. so warrants. In no event shall Smithy's liability under this warranty exceed the purchase price paid for the product. Legal actions brought against Smithy Co. shall be tried in the State of Michigan, County of Washtenaw.

Smithy Co. shall in no event be liable for death, injuries to persons or property for incidental, contingent, special or consequential damages arising from the use of our products.

This is Smithy Co.'s sole warranty and any and all warranties that may be implied by law, including any merchantability or fitness, for any particular purpose, are hereby limited to the duration of this written warranty.

This warranty gives you specific legal rights, and you may also have other rights, which vary from state to state. Some states do not allow the exclusion or limitation of incidental or consequential damages, so the above limitation or exclusions may not apply to you.

Telephone Support (Service engineers are available 8 am to 5 pm EST)

Service and Parts

Tel No. 1-800-476-4849
Fax No. 1-734-913-6663
Email Address: sales@smithy.com

Software and Programming Consultancy Services

In addition to our customary technical support for the machines and controls, we also provide technical consulting support to our customers by providing engineering and G-code programming services. The standard rate for these services is $28.00 per hour. Our principal objective is to support you and to increase your productivity while reducing the machining cost. Give us a call for such support as and when required.

Tel No. 1-800-476-4849
Fax No. 1-734-913-6663
Email Address: sales@smithy.com