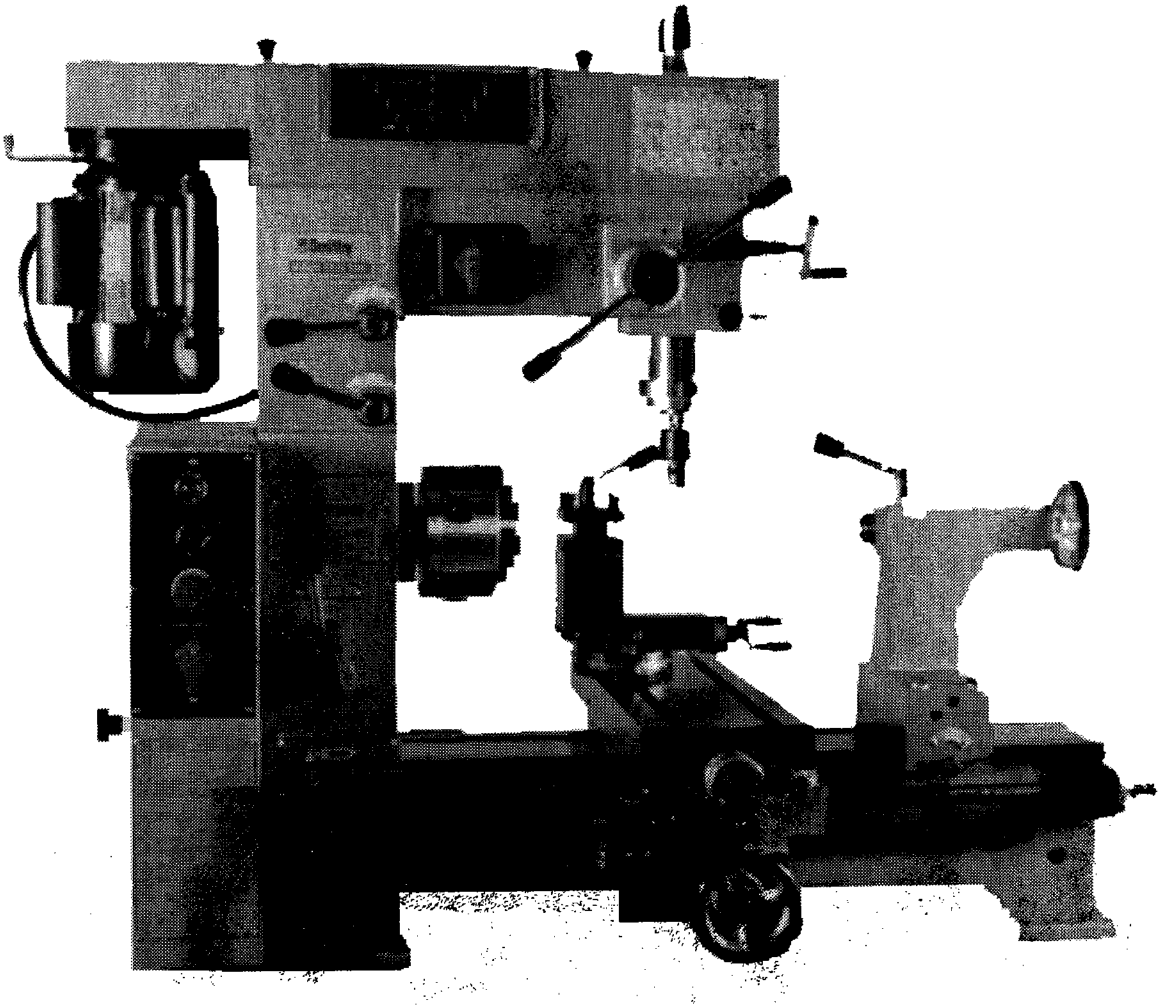
Smithy®

Operator's Manual



MI-1720CNC
Lathe•Mill•Drill

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

© 1999 by Smithy Co.

1-800-476-4849

Table of Contents

One: Introduction *Page(s)*

1.1 Customer information 7
1.2 Specifications 6

Two: Safety 8

Three: Caring for Your Machine 9

Four: Basic Parts 10-12

Five: Uncrating and Setting Up

5.1 Moving the machine 13
5.2 Uncrating and positioning the machine 14
5.3 Selecting a location 14
5.4 Cleaning and lubricating the machine 15-16
5.5 Setting lathe speeds 17
5.6 Setting mill speeds 17
5.7 Adjusting belt tension 18
5.8 Adjusting the gibs 18
5.9 Reducing backlash 19
5.10Running In 20

Six: Turning

6.1 Turning speeds 21-22
6.2 Gear ratios 22-23

Seven: Metalcutting Theory

7.1 Tool sharpness 24
7.2 Heat 24-25

Eight: Grinding Cutter Bits for Lathe Tools

8.1 High-speed-steel cutters 26-27
8.2 Materials other than steel 27
8.3 Bits for turning and machining brass 27
8.4 Special chip craters and chipbreakers 27
8.5 Using a center gauge to check V-thread forms 27
8.6 Acme or other special threads 28
8.7 Carbide-tipped cutters and cutter forms 29

Nine: Setting Up Lathe Tools *Page(s)*

9.1 Cutting tool height 29
9.2 Turning tools 29
9.3 Threading tools 29
9.4 Cut-off, thread-cutting, and facing tools 30
9.5 Boring and inside threading tools 30

Ten: Setting Up with Centers, Collets, and Chucks

10.1 Centering 31-32
10.2 Mounting work between centers 32
10.3 Using a clamp dog 32-33
10.4 Using faceplates 33
10.5 Setting up work on a mandrel 33
10.6 Steady rests and follow rests 33-34
10.7 Setting up work in a chuck 34
10.8 Mounting work in a four-jaw independent lathe chuck 34
10.9 Mounting work in a three-jaw universal chuck 36

Eleven: Lathe Turning

11.1 Rough turning 38-39
11.2 Finish turning 39
11.3 Turning to shapes 39
11.4 Machining square corners 39-40
11.5 Finishing and polishing 40
11.6 Taper turning 40
11.7 Boring a tapered hole 41

Twelve: Lathe Facing and Knurling

12.1 Facing across the chuck 42
12.2 Knurling 42-43

Thirteen: Cutting off or parting with a lathe 44

Fourteen: Lathe Drilling Boring

14.1 Reaming 45
14.2 Boring 45-47
14.3 Cutting internal threads 47
14.4 Cutting special-form internal threads 47

Fifteen: Changing Gears 48

Sixteen: Cutting Threads on the Midas 1720CNC *Page(s)*

16.1 Threading Terms 49-50
16.2 Cuttin right-hand threads 50
16.3 Using the treading dial 51
16.4 Cutting Multiple threads 52
16.5 What not to do when cutting threads 52
16.6 Finishing off a threaded end 53
16.7 Cutting threads on a taper 53

Seventeen: Milling

17.1 Holding Milling Cutters 55-56
17.2 Milling Cutters 57-59
17.3 Using Cutting fluid 60
17.4 Tool Gringing 60
17.5 Speeds and Feeds for Milling 61-63
17:6 Common Miling Operations 64

Eighteen: Workholding

18.1 Mounting to the table 65
18.2 Using a vise 65
18.3 Dividing heads 65
18.4 Rotary tables 66

Nineteen: Troubleshooting 67-71

Twenty: Parts Lists with Schematics 72-109

1.2 Specifications—Midas 1720 CNC

General dimensions: 42" long 22" wide 36" high
Crate size: 44" long 23" wide 38" high
Weight: shipping 600 lb, machine 500 lb
Power: dual 3/4 hp, 110 V AC (0.55 Kw)
Mill/drillhead swivels
Calibrated dials
    Leadscrew: 0.002"
    Cross slide: 0.002"
    Tailstock: 0.001"
    Mill and drill: 0.040"

Lathe

Swing: over bed 17", over table 7"
Distance between centers: 20"
Spindle bore: 1.102"
Headstock taper: Morse taper #4
Tailstock taper: Morse taper #3
Speeds:Seven 160-1360 rpm
Thread pitches
    Inch-thread: 40-120 tpi
    Metric: 0.75 - 6 mm
Hardened, ground dovetail ways
Travel: cross slide 7.9 ", longitudinal 18.1"
5" three-jaw chuck with inside and outside jaws
Leadscrew Pitch: 6 tpi
Powerfeed rates .0002" - .014" rpm
Tailstock taper: Morse taper #3
Tailstock barrel travel 3.14"

Mill

Millhead taper: Morse taper #3
Spindle nose to table: 7.7" to 12.0"
Spindle to lathe faceplate: 11.2"
End Mills up to 1.1'
Face Mills up to 3"
Table: 6" wide 7-3/4" long
T-slot width: 7/16"
Quill diam: 3"
Quill travel:4.33"
Speeds: 16 (120 - 3000 rpm)
Working height: 6-1/4" min, 13" max
Drawbars
    Standard: 3/8–16 tpi

SECTION ONE

INTRODUCTION

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

The purpose of this manual is to give the machinist, beginning or advanced, the information he needs to operate the Smithy Midas 1720 CNC. It will teach you about the machine's parts and how to care for them. In fact, education is our primary goal. We'll explain how to grind cutters, set up lathe tools, hold workpieces, and do all basic machining operations.

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

If you have any questions not covered in this 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 5:00 pm Eastern Time. You can also find Smithy on the internet at www.smithy.com. Check for service updates and service bulletins.

We are always interested in your suggestions to improve our products and services. Feel free to contact us by phone or in writing. If you have comments about this operator's manual, or if you have a project you'd like to share with other Smithy owners, contact the Communications Director, Smithy Co., 170 Aprill Drive, PO Box 1517, Ann Arbor, MI 48106-1517.

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

1.1 Customer information

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

Model #______________________________________

Serial #______________________________________
(on the back of the lathe bed)

Purchase date__________________________________

Delivery date__________________________________

Sales technician_______________________________

SECTION TWO

SAFETY

Your workshop is only as safe as you make it. Take responsibility for the safety of all who use or visit it. This list of rules is by no means complete, and remember that common sense is a must.

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

- Ground the machine. TheMidas 1720 CNC has three-conductor cords and three-prong grounding-type receptacles. Never connect the power supply without properly grounding the machine.

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

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

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

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

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

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

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

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

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

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

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

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

Remember: your safety depends largely on your practices.

SECTION THREE

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:

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

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

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

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

- Use a good 10W 30 wieght non-detergent oil on your machine.

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

- 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 be unused for some time are BoeShield, developed by the Boeing company and CRC lubricants. There are also speciality oils that may be purchased for your machine, way oil for the ways and table of the machine and turbine fluid or 10 wt hydraulic oil for the headstock.

SECTION FOUR

BASIC PARTS OF THE Midas 1720 CNC

To learn the operations of your machine, you have to know the names and functions of its basic units.

- 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 toolpoint and workpiece from one end of the machine to the other, the ways must be absolutely true and accurately aligned to the line of centers and to one other.

- 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 rigidly and move it along the bed for different operations. It locks into place by tightening the carriage lock under the cross-slide handwheel.

- 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 calibrations on the front of the base are numbered in degrees from 60° right to 60° left.

- Cross slide. The T-slotted cross slide moves crosswise at 90° to the lathe axis by manual turning of the cross-feed screw handwheel. It also serves as the milling table.

- 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 use it as a drill press or push it in to use the fine feed. Use the fine-feed handwheel to move the quill up and down.

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

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

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

- Mill belt tensioner . To adjust the lathe belt tensioner, pull the handle forward to tighten the belt, back to loosen it.

- 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 workpiece and drives lathe chucks and other workholding devices. All attachments (three-jaw chuck, four-jaw chuck, faceplate, etc.) bolt to the spindle flange either directly or via an adapter plate.

- 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.

- 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.

- Micrometer control and calibration. Just inside the handles of the tailstock crossfeed, drill press, compound feed, and leadscrew are collars calibrated in inches. The dial on the compound feed is also calibrated in millimeters. The compound feed and crossfeed are calibrated in two thousandths, the tailstock in thousandths, the leadscrew in two thousandths, and the drill press in forty thousandths.

These micrometer dial collars can move independently around the handle shafts. This independent motion is called float. The Midas 1720 CNC has floating dials on the cross slide, tailstock, and leadscrew. They let you zero the collars at any point and read the feed travel from that point on the dial for increased accuracy.

- 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 fit into collets that attach through the spindle via drawbars.

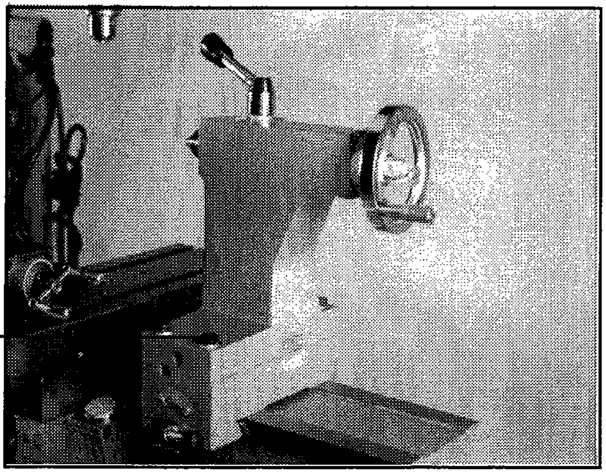
- Half-nut lever. This lever transmits power to the leadscrew when rotated down.

- Power Cross-feed. Pull out on the knob to engage the cross-feed and push in to disengage.

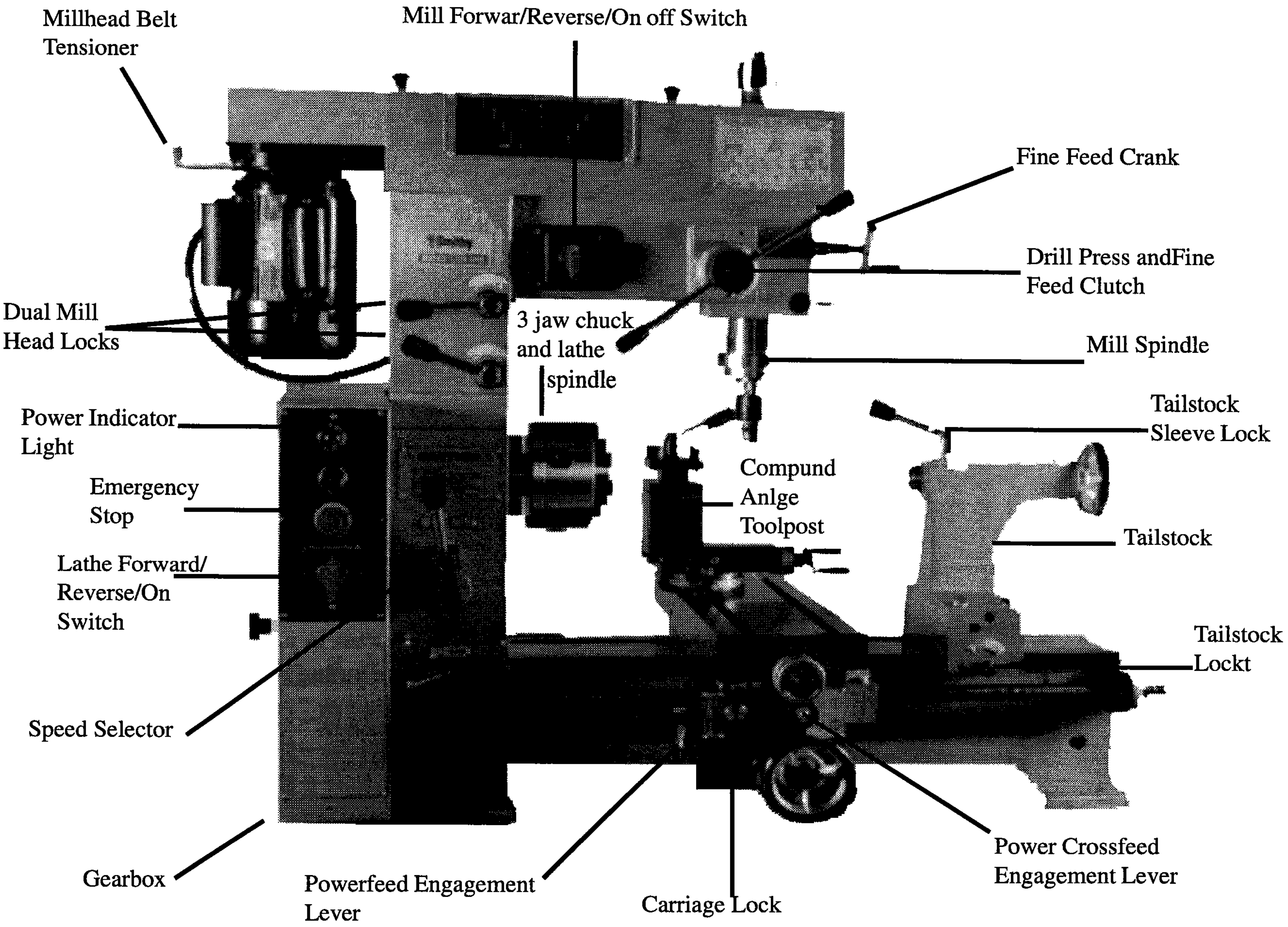
- Powerfeed speed selector . The two-speed selector for powering the leadscrew is on the front of the headstock. The leadscrew turns twice as fast in the II position as in the I position.

- 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 handwheel. The scale of offset calibrations on the back of the tailstock is in millimeters.

To offset the tailstock, loosen the four base-locking bolts. To offset to the left, loosen the left adjusting bolt and tighten the right. To offset to the right, loosen the right adjusting bolt and tighten the left. (Figure 4:1).



4.1 To offset the tailstock loosen the four bolts at the base of the tailstock.


Millhead Belt Tensioner
Mill Forwar/Reverse/On off Switch
Fine Feed Crank
Drill Press andFine Feed Clutch
Dual Mill Head Locks
3 jaw chuck and lathe spindle
Mill Spindle
Power Indicator Light
Tailstock Sleeve Lock
Emergency Stop
Compund Anlge Toolpost
Tailstock
Lathe Forward/ Reverse/On Switch
Tailstock Lockt
Speed Selector
Gearbox
Powerfeed Engagement Lever
Carriage Lock
Power Crossfeed Engagement Lever

SECTION FIVE

UNCRATING AND SETTING UP THE MI-1720CNC

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 tool. Follow these guidelines whenever you use your Smithy machine:

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

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

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

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

- Use a good 10W 30 wieght non-detergent oil on your machine.

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

- 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 be unused for some time are BoeShield, developed by the Boeing company and CRC lubricants. There are also speciality oils that may be purchased for your machine, way oil for the ways and table of the machine with turbine fluid or 10 wt hydraulic oil for the headstock.

5.1 Moving the machine

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

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

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

5.2 Uncreating and Setting Up Your Machine

The machine is assembled, inspected, and ready to go on its stand. It's wrapped in a water and greaseproof cover, strongly barced and crated. Your basic tool pack is included in your crate.

The metal bands that encricle the crate are under tension. Wearing eye protection and golves, cut the metal bands with tin snips. Be careful-the cut edges are sharp. The bands secure the crate top to the base.

After removing the straps, you will need two people to remove the crate cover. With one person standing at each end, grip the bottom of the crate lid. Lift straight up to remove the crate cover. Do not dispose of your crate cover you may need it to transport your machine in the future.

Once the crate cover is removed, you will need to remove the machine from the skid and place it on a work bench. There are four holes located in the bed of the Midas 1720CNC. Place a sturdy 3/4" pipe/rod through these holes. It is best to use a engine hoist or some other lifting device with the aid of lifting straps/chain to move your machine from the ground level to a work bench. Be sure to have have some friends or family members with you to work as a spotter.

Once your machine is lifted onto a workstation, it is important that you secure it to your workstaion throught the foot pads of your machine.

5.3 Selecting a location

There are several major considerations for selectiong a location for your Smihy.

Operation is from the apron side, so allow at leat 40-48" 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.

Place the machine on a solid foundation-concreate, 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 well as at it s end and above it for later additions, attachments, and or accessories. PRovide clearance on the left end for bar stock to be fend through the spindle. If you are considering placing more than one machine in an area, allow endough floor space to feed long bar stock to ecah machine.

5.4 Cleaning and lubricating the MI-1720CNC

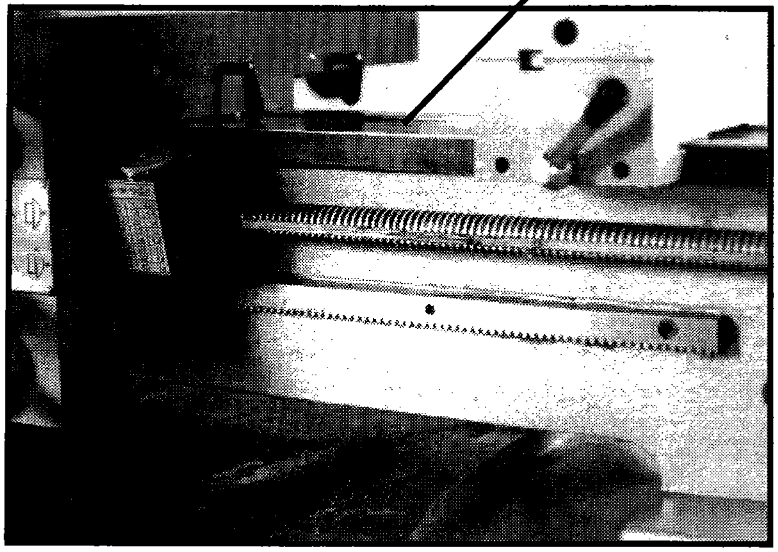
Smithy machines are shipped with a protective grease coating called cosmoline. Use WD-40 or a noncorroseive kerosene to remove the cosmoline.

Once you have your Midas 1720CNC set up and postioned correctly, you 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 trorough and complete, follow this routine:

Oiling the Ways

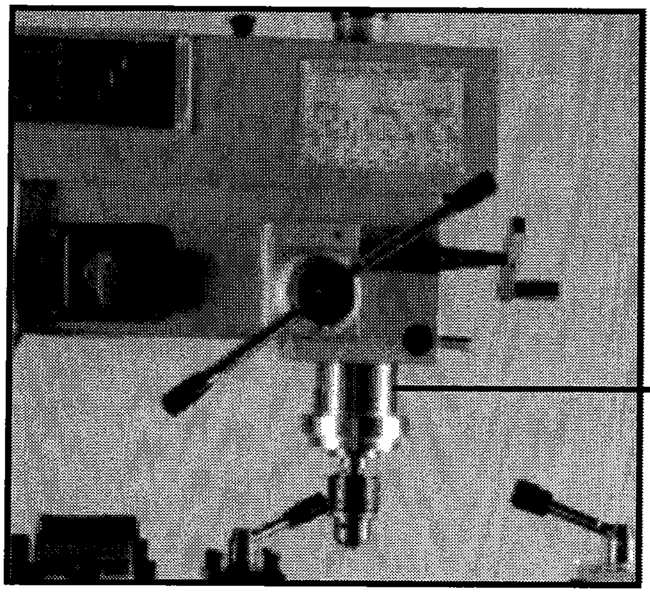
Run the carraige as far to the left as possible. Put a few drps of oil on the ways. Run the carriange to the extreme right and repeat. You may want to use the waylube an special oil formulated for the ways.



Apply a ligtht layer of oil to the ways.

Oiling the Millhead Quill

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



Place a few drops of oil here and use the drill press handles to move the quill up and down to work the oil.

Smithy machines are shipped with a protective grease coating. Use a non-corrosive kerosene to remove the cosmolione.

Once you have the Midas-1720CNC set up and postioned correctly, you are ready for lubricating. You must do the caarefully and thoroughly before starting the machine. ;USe a pressure oil can and a supply of good-quality SAE 10 weight oil.

The be thorough and complete follow this routine.

Oiling the carriage

Lubricate the oil buttons in the cross-feed table. There are two buttons on each side of the crossslide table (Figure 5:1). Using your oil can, oil each of these buttons.

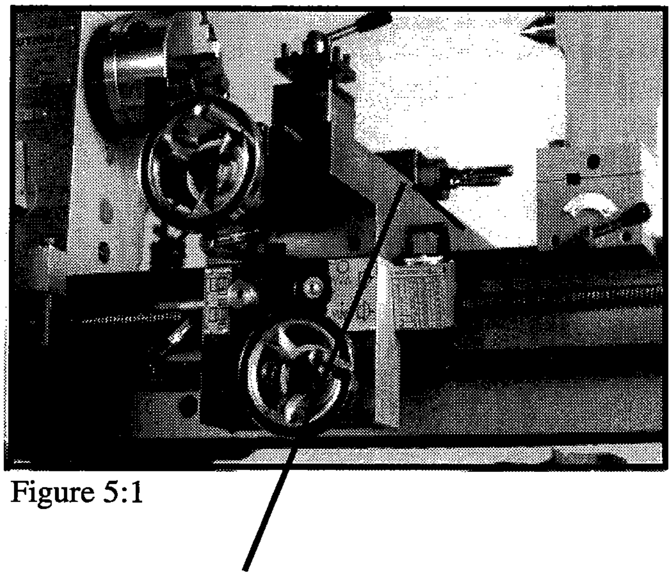


Figure 5:1

While oiling the buttons on your crosslide table, put a few drops of oil on the compound and crosslides feed-screws. Also the compound angle toolpost has two oil buttons that should be oiled as well.

Oiling the Tailstock

There are three oil buttons on top of the tailstock that need to oiled. (Figure 5:2)

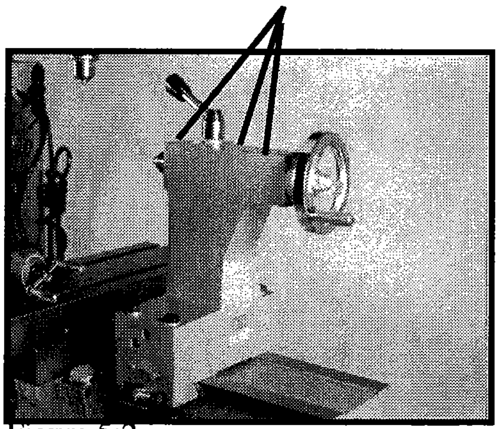
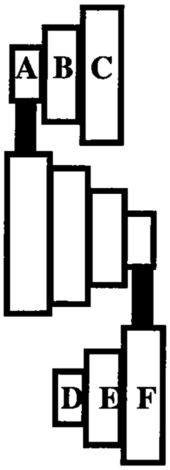


Figure 5:2

5.5 Setting Lathe Speeds on the MI-1720CNC

Changing belt positions on the MI-1720CNC changes speeds. Your machine comes with two v-belts, one long one and one shorter belt. Use the longer belt between the lathe spindlepulley and the middle pulley. The shorter belt is placed the motor pulley and the middle pulley. Use the the chart at the left to place your belts and determine your lathe speeds. Remember, always make sure your motor is disengaged before changing your belts.

Figure 5:3



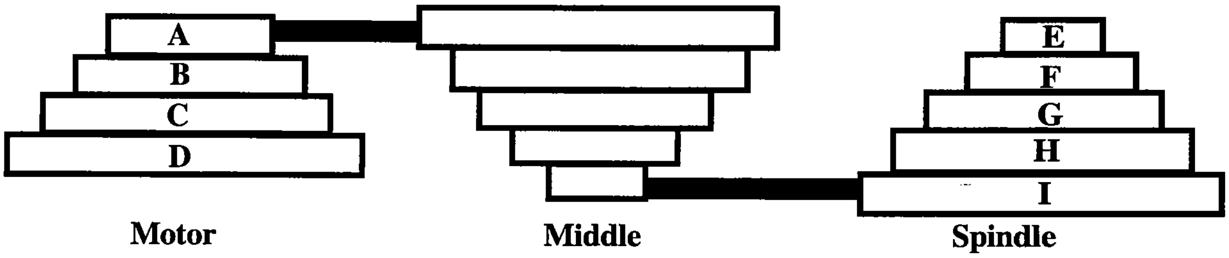
Setting Lathe (RPM)

A-F	160
A-E	300
A-D	375
B-F	470
C-F	600
B-E	870
C-D	1360

5.6 Setting Mill Speeds on the MI-1720CNC

The mill changes speed in the same manner that the lathe does, by changing the belts. To set mill speeds, use the two v-belts that come with your machine and use the chart below to determine the desired mill speeed.

Setting Mill/Dril Speeds (RPM)



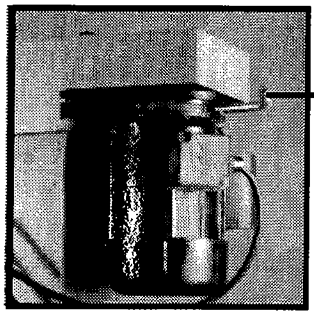
A-I	**B-I**	**A-H**	**C-I**	**A-G**	**D-I**	**B-H**	**A-F**
120	**200**	**310**	**350**	**400**	**450**	**530**	**600**
B-G	**C-H**	**B-E**	**D-G**	**C-F**	**D-F**	**C-E**	**D-E**
660	**900**	**1380**	**1450**	**1670**	**2140**	**2350**	**3000**

Figure 5:4

5.7 Adjusting belt tension on the MI-1720CNC

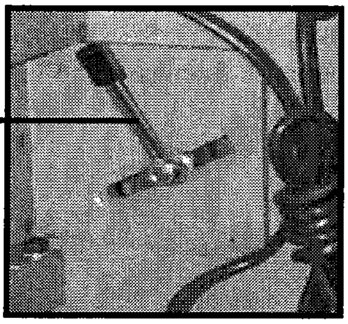
The MI-1720CNC has two belt tensioners installed by the factory, one for the millhead and one for the pulley box. You will see a "L-shaped lever" positioned at the top of the millhead motor. To losen the tensiton, turn the lever counter-clockwise.

On the back side of your machine there a handle directly above the lathe motor on the pulley box. Turn this counter clockwise as far as it will go to losen the belts in the pulley box.



Mill Belt Tensioner

Figure 5:5



Lathe Belt Tensioner

Figure 5:6

5.8 Adjsuting the Gibs

The Midas has a tapered gib on the crossfeed and the straight gibs o n the carriage, the compound tool post and the tailstock. Gib adjustment effects the tool rigidity and accuracy of the cuts.

As the gibs tighten, the effort it takes to move the controls increases. Adjsut the gibs according to the work you are doing and personal preference. Wha is importatnt is to adjust them evenly. The tigher the gib, the more accurate it will be. Removing and polishing the gibs also improves the accuracy.

Crossfeed

The crossfeed gib is located under the crossfeed table and to the right. There is a screw at the rear of the gib to lock it into place and a screw at the front of the gib to move it in and out (tighten or loosen). Loosen the rear screw and then use the front screw to tighten or lossen the gib as desired. Check the crossfeed for desired movement. Tighten the rear screw to lock the gib into position.

Carriage and Tailstock

The carriage and tailstock each have two gib adjustment points. Each point contains a locking set screw and an adjusting set screw. Remove the locking set screw from each ajustment point. Tighten the ajusting screws all the way, locking up any lateral movement. Then back the screws off 1/8" to /4" of a turn and check the movement.

The effort to move the carriage or tailstock should be the same in both directions. If it is not, the gib is not adjusted evenly. If there is more resistance felt moving to the right., the right gib is tighter than the left and visa versa. As you work with the adjustments, you will feel the differerence even gib tensions makes on the ease of movement.

When everything is set, reinstall the locking set screws to hold the ajustments in place.

Compound Tool Post

There are two gib adjustment screws on the tool post. They extend out the side of the tool post and are locked inplace with a jam nut. Adjustment procedure is the same as the carriage and the tailstock.

5.9 Removing Backlash

Some backlash on the crossfeed and longfeed is normal, but to keep it to a minimum the thrust bearings on each feed screw must be kept tight. The bearings are held in the proper position by two spanner nuts. The nuts for the crossfeed are located on the front end of the screw under the crossfeed dail (M18717). The handle and dial must be removed to asjust these nuts. The nuts for the longfeed are lcoated in the transmission box (part NO. M10144) just below the powerfeed engagement lever. They may be accessed from under thetransmission box.

Adjusting the spanner nuts to assure minimum backlash is the same for both screws. Using a pin punch, loosen the outside locking nut and then the inside adjusting nut. Using the pin punch, slowly tighten the inner adjusting nut until a light drag is felt on the screw when it is rotated. Tighten the outer locking nut with the punch while frequently checking the ease of rotation of the screw. If the rotation of the screw becomes tight, the adjusting nut has turned with locking screw. This can be corrected by loosening the adjustment nut slightly.

5.10 Running in the MI-1720CNC

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

Follow these steps:

Make sure that the Forward/Off/Revesr switch for the lateh mtor and the mill motor are both in the off position.

If not already installed, screw the red stop button into the end of the stop switch. Pushing the red button in will kill all power to the machine. The red button will stay in until it is reset. To reset the switch, turn the red button clockwise until it pops out about 1/2 inch into the operation position.

Close the door on the end gearbox. The machine will not operate with the door open.

Plug the machine into a grounded 20-amp circuit.

Start the mill motor by placing the mill switch into "1" position. After a few minutes, switch the mill motor "O" (Off) and allow the motor to stop. Repeat the procedure in the "2" position then turn the mill motor off.

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 know 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.

Place powerfeed selector in position III, the slowest speed.

Make sure all other powerfeed controls are in the disengaged position: Powerfeed engagment lever moved to the right, half nut lever pushed down, and the crossfeed button pushed in.

Manually place the longfeed and the crossfeed in the center of their travel.

Start the lathe motor by placing the lathe switch into the "1" position.

Engage the powerfeed by moving the engagement lever to the left until the leadscrew starts to trun. Pull the half nut handle up to the activate the longfeed, push it back down to disengage. Pull the crossfeeed button out to activate the longfeed, push it back dwon to disengage. Pull the crossfeed button out to activate the crossfeed, push it back down to disengage.

> NOTE: If the crossfeed screw turns but there is no table movement, pull the table toward the operator slightly while the power crossfeed is engaged. There is a void spot in the crossfeed thread to prevent the table from being ran too far back (crashing the table). Some early machines did not have this safety feature. Instructions are available through Smithy Company to modify the crossfeed screw on such machines.

Turn the lateh switch to "O" and all wthe lathe to stop. Place the lathe switch to "2" amd repeat the powerfeed run-in test.

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

SECTION SIX

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.

6.1 Turning speeds

When metal cuts metal at too high a speed, the tool burns up. You can machine soft metals like aluminum at fast speeds without danger or trouble, but you must cut hard steels and other metals slowly.

You must also consider the diameter of the workpiece (Figure 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 Midas 1720 CNC 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. Figure 5.4 shows belt positions for various speeds.

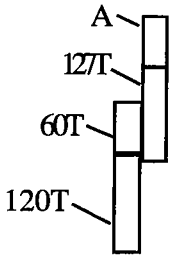
6.1 Cutting Speeds for Various Diameters

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

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.

Figure 6.2
Cutting Speeds for the MI-1720CNC

Cutting Speeds

	Travel per Rev	A	24	30	36	42
		I	0.2	0.25	0.30	0.35
		II	0.1	0.12	0.15	0.175
	MM	III	0.05	0.063	0.075	0.088
	Travel Per Rev	I	0.008	0.010	0.012	0.014
		II	0.004	0.005	0.006	0.007
	INCH	III	0.002	0.0025	0.003	0.0035

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

6.2 Gear ratios

The lathe lets you use various indicated gear combinations to cut the desired number of threads per inch (tpi), or the metric equivalent, or to advance the tool a specified amount each revolution (feed rate expressed as inches per revolution [ipr]).

The MI-1720CNC has pick-gear gearboxes (*Figure* 14.1); gears are picked and placed to change the gear ratios. The gearbox mechanism determines the leadscrew's rotation rate in relation to the spindle's for threading, turning, and facing. To change the fee rate, replace the gears per *Figure* 6.2.

SECTION SEVEN

METAL CUTTING THEORY

7.1 Tool sharpness

Instead of being the all-important factor in determining tool performance, keenness of the cutting edge is just one of many 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 toolholder you use. The MI-1720CNC have a four-sided turret toolpost that accommodates four high-speed-steel (HSS), carbide-tipped, or indexable carbide turning tools.

7.2 Heat

The energy expended at the lathe's cutting point converts largely into heat, and because the energy expended is great, the heat is intense. Before 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.

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

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

In everyday lathe operations like thread cutting and knurling, always usa a cutting oil or other lubricant. On such work, especially if the cut is light and lathe speed low, dipping a brush in oil occasionally and holding it against the workpiece will provide sufficient lubrication. For continuous, high-speed, heavy-duty production work, however, especially on tough alloy steels, using a cutting oil or coolant will increase cutting efficiency. It's essential if you're using a non-HSS cutting tool. When you use coolant, direct it against the cutting point and cutter. Consider installing a coolant system if you don't have one.

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

rpm = CS × 4 / D″

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

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

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

rpm = 300 sfm × 4 / 1

rpm = 1200 / 1

rpm = 1200 or next slower speed.

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

rpm = 50 sfm × 4 / 1

rpm = 200 / 1

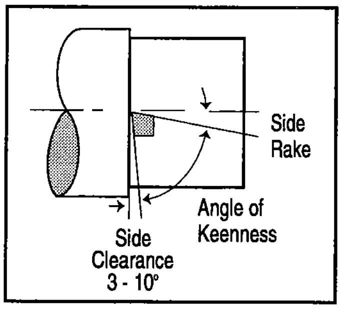
rpm = 200 or next slower speed.

The 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.

SECTION EIGHT

GRINDING CUTTER BITS FOR LATHE TOOLS

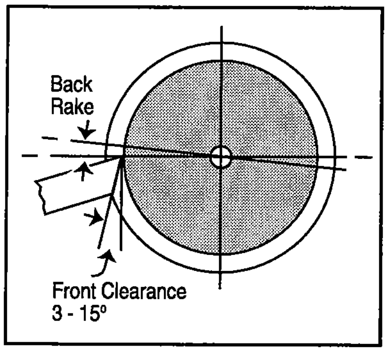
8.1 High-speed-steel cutters



8.1 Keenness angles vary from 60° to 90°.

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.



8.2 The edge weakens if front clearance is too great.

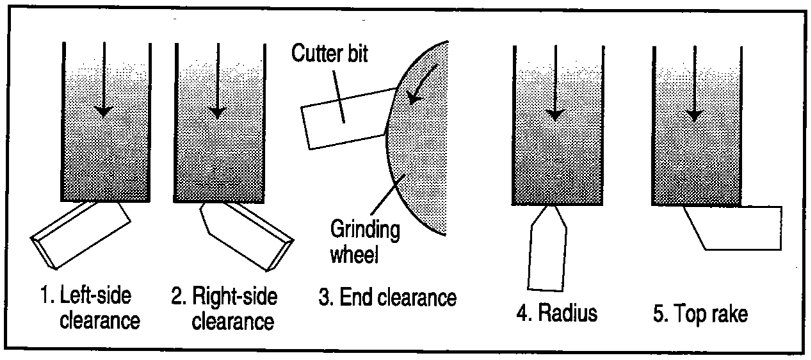
In grinding HSS cutter bits, you have five major goals:

- 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).

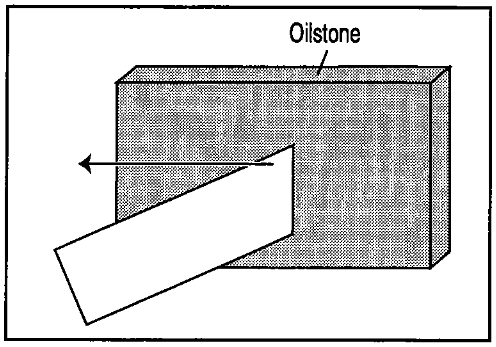
Front clearance must always be sufficient to clear the work. If it is too great, however, the edge weakens and breaks off (*Figure* 8.2). Side and back-rake requirements vary with the material used and operation performed. Back rake is important to smooth chip flow, which is needed for a uniform chip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

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



8.3 Grinding sequence for an unground cutter bit.

the cutter away from the cutting edge across the oilstone (*Figure* 8.4).

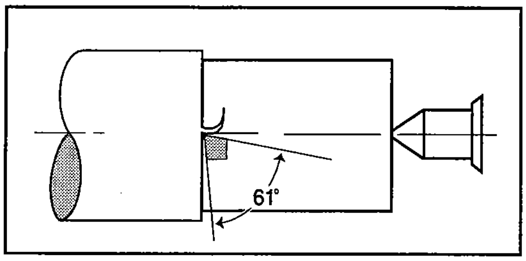


8.4 When honing, draw the cutter away from the cutting edge across the oilstone.

8.2 Materials other than steel

As pointed out earlier, when grinding HSS cutters, we determine cutting angles primarily by strength requirements, not keenness requirements. Angles and rakes for general industrial shop use are established. In machining steel, the softer the steel, the keener the angle of the cutting edge. For soft steels, angles as acute as 61° are possible (*Figure* 8.5).

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



8.5 With soft steels, 61° angles are possible.

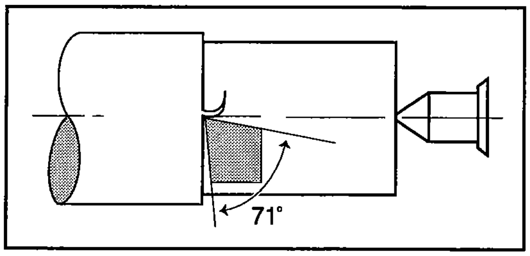
8.3 Bits for turning and machining brass

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

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

8.4 Special chip craters and chipbreakers

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



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

8.5 Using a center gauge to check V-thread forms

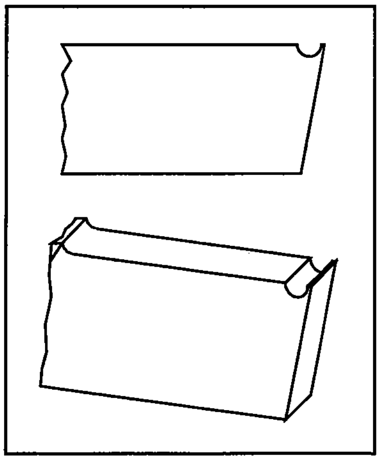
It may be convenient to grind a standard cutter bit for thread cutting, especially for cutting standard 60° V-threads. When grinding an ordinary square cutter into a thread-cutting tool, take care to ensure a true thread form. The easiest way is to use an ordinary center gauge for a standard V-thread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

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

8.6 Acme or other special threads

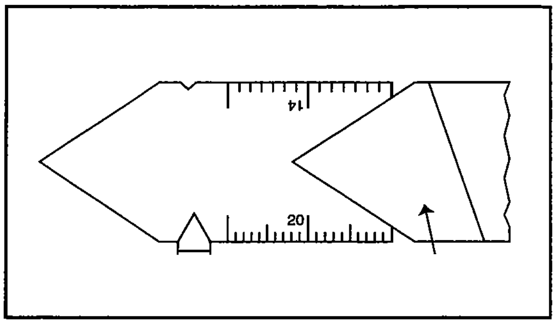
Thread gauges are available for all standard threads. Before grinding such cutters, ascertain the correct pitch angle of the particular thread profile. For example, the pitch of an acme thread is 29° to a side, and the toolpoint is ground back square to an exact thread profile that requires a different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

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



8.7 A crater starts the chip curling smoothly.

8.7 Carbide-tipped cutters and cutter forms



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

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.

8.9 Carbide Types and Cutting Tool Applications

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

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

SECTION NINE

SETTING UP LATHE TOOLS

After selecting a cutter, insert it in the toolholder. Allow the cutter bit to project just enough to provide the necessary clearance for the cutting point. 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.

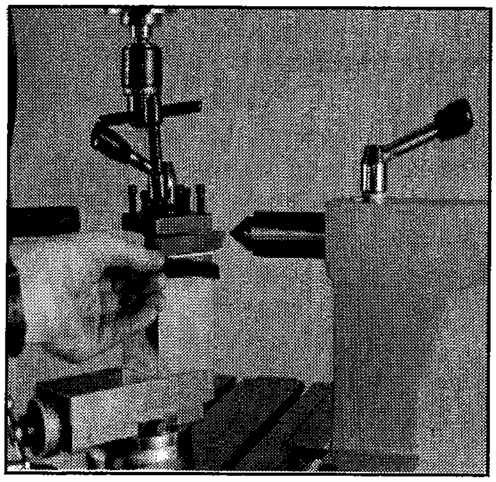


Figure 9.1 Placing shims under the tool can correct tool height.

9.1 Cutting-tool height

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

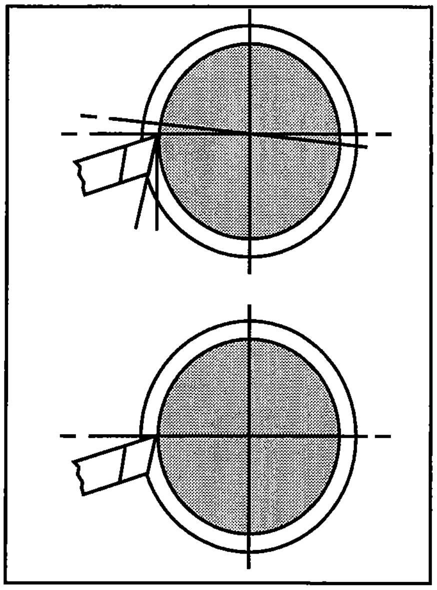


Figure 9.2 The harder the steel (top) the less above center you

9.2 Turning tools

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

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)

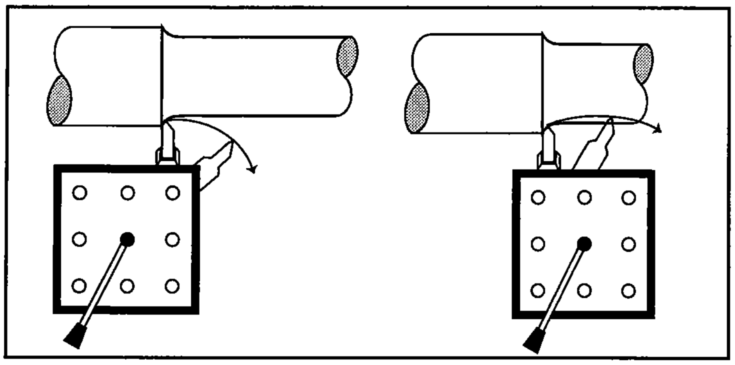
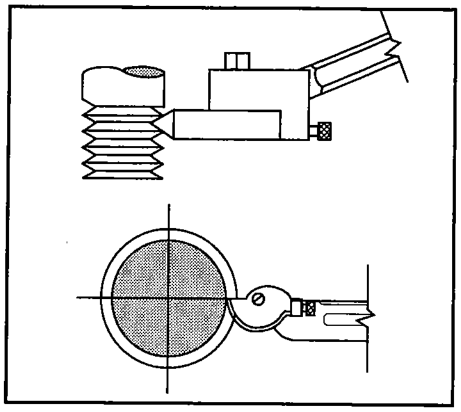


Figure 9.3 The tool will swing out of the cut (left) rather than hog into the work (right) if you reach a stalling point. Note the tool is in the right-hand side.

9.3 Threading tools

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



9.4 Threading tools engage the work on dead center.

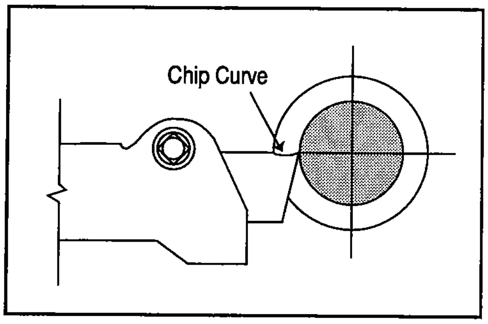
9.4 Cut-off, thread-cutting, and facing tools

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

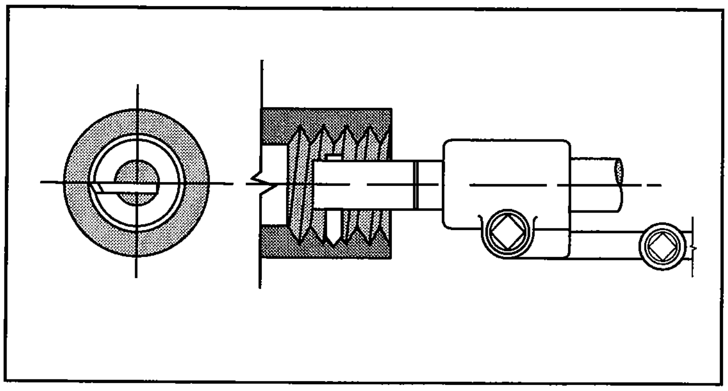
9.5 Boring and inside threading tools

For boring and inside threading, the cutter point engages the work on dead center (*Figure* 9.6). For greater cutting efficiency, position the bar while parallel to the line of lathe centers sufficiently below center to give the cutter a 14-1/2° approach angle. For internal threading, grind the top face of the cutter to compensate for this angle, giving a flat, true-form top face.

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



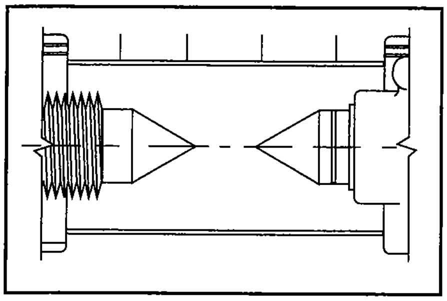
9.5 Feed the cutter on dead center for cutoff, thread cutting, and facing.



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

SECTION TEN

SETTING UP WITH CENTERS, COLLETS, AND CHUCKS



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

Before setting work up on centers, make sure the spindle and tailstock centers align accurately. Do this by inserting a cen-ter 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 cor-rect any lateral alignment error by adjusting the tailstock set-over screws (*Figure* 4.8).

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

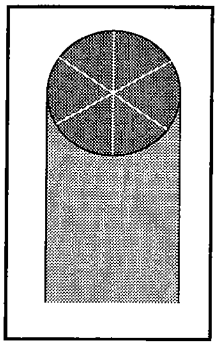
10.1 Centering

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.

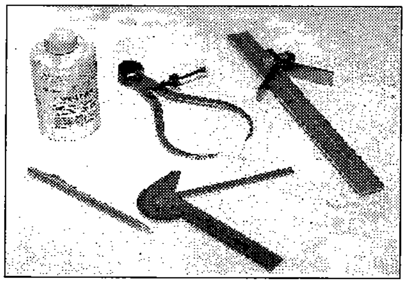
Start your lathe on the slowest speed. Bring the tool into the cutting position against the center of the workpiece. Feed the tool from the center of the stock outward, toward yourself, using the hand crossfeed. One or two light cuts is usually enough to true up an end roughened by the hacksaw. After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

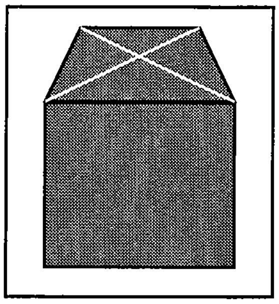
After locating the center of each end, drive a starting depression for the drill into the stock with a center punch. Check centering accuracy by placing the workpiece between the spindle and tailstock centers. Revolve the headstock slowly against the tip of a tool or a piece of rigidly held chalk. The 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.



10.2 Centering on round stock.

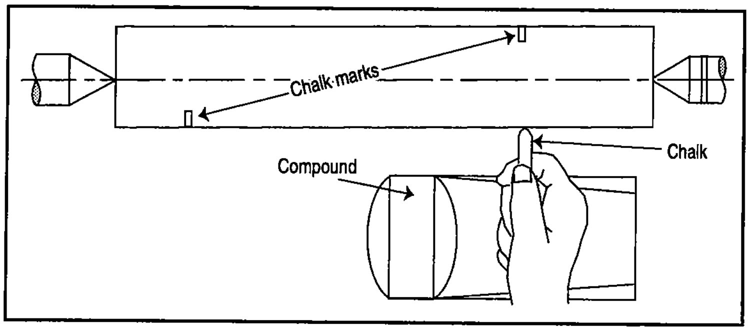


10.3 Use centering instruments include calipers and dividiers.

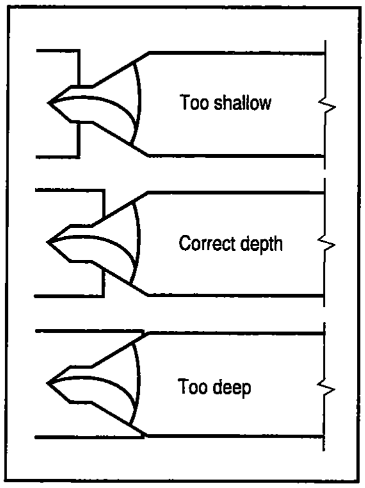


10.4 Centering on square or rectangular stock.

Next, drill and countersink the centers to conform to the profile of the lathe centers. This is



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



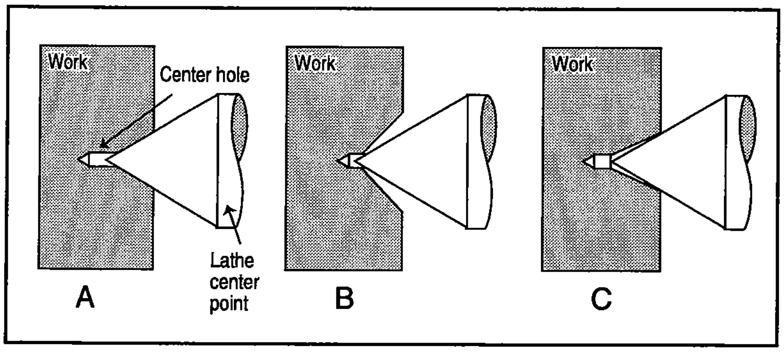
10.6 The correct depth of center is illustrated above. If it's too deep (bottom), only sharp outer edges will contact the center.

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).

10.2 Mounting work between centers

Remove the chuck from the lathe, bolt the faceplate to the spindle flange (*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 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.

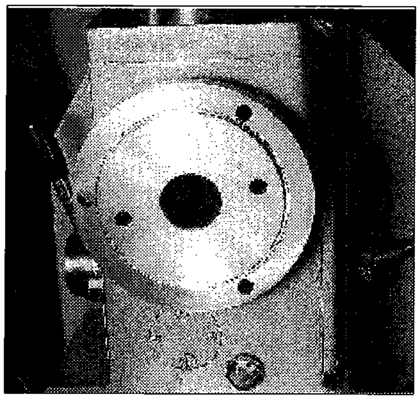


10.7 Counterbore centers with a drill to a 60° point so they fit the lathe centers (A). Too obtuse (B) or too acute (C) a counterbore will give insufficient bearing, prevent accuracy, and destroy the lathe centers.

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

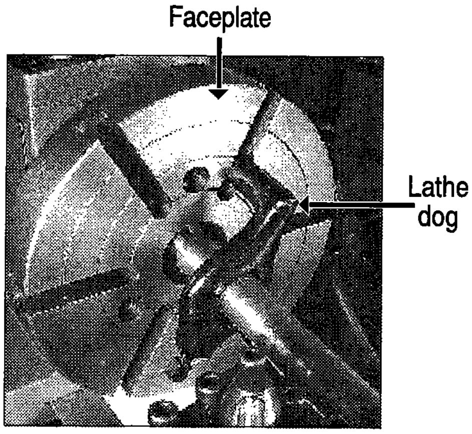
10.3 Using a clamp dog

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



10.8 Bolt the faceplate to the spindle flange.

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.



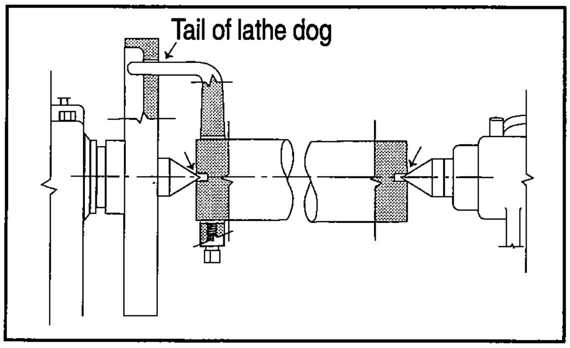
10.9 Fasten a lathe dog to one end of the workpiece.

10.4 Using faceplates

For work setup, faceplates serve two purposes. First, they drive workpieces held between centers. Second, they hold workpieces shaped so you can't chuck them or mount them on centers.

Faceplates for driving workpieces on centers are generally small. They're notched and slotted to receive the tail of the lathe or clamp dog, bolt drive, or other driving tool (*Figure* 10.9). Faceplates for holding workpieces (irregularly shaped casting, machine, or die parts, for example) are usually larger and of varied design. They may be T-slotted, drilled all over, or slotted and drilled. Workpieces mount on such faceplates with T-slot or standard bolts, strap clamps, angle plates, or other standard setup tools.

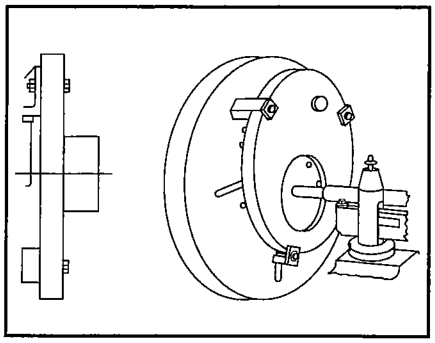
Note: Before starting to machine work set up on centers, check to see the lathe dog tail is free in the faceplate slot so it won't lift stock off its true line of centers, as in Figure *10.10. Also, be sure lathe centers fit closely into the center holes to eliminate side play but not so tightly they 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).*



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.

10.5 Setting up work on a mandrel

You can machine cylindrical or bored pipe work or cored castings too long to fit in a chuck by mounting them first on a mandrel (*Figure* 10.12). Then mount them between centers. The solid mandrels, which are driven into the hole of the workpiece, must be tight enough to turn the workpiece against the tool without slippage. Oil them lightly before driving them into the workpiece. Otherwise, the workpiece may freeze to the mandrel, making it impossible to remove the mandrel without damaging both workpiece and mandrel. When removing a mandrel, drive it back out of, instead of through, the hole.

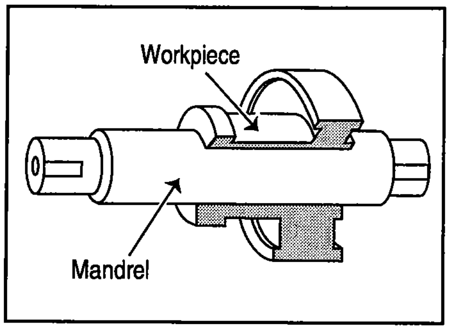


10.11 Counterweights can help with unbalanced setups.

You can purchase hardened steel mandrels, which have a slight (0.003") ground taper and an expanding collar, to facilitate mounting and demounting (*Figure* 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.

10.6 Steady rests and follow rests

Rests are for setting up (1) work that is relatively long in proportion to its diameter or (2) work whose dead end must be left free for boring or other operations. You can also use rests to machine slender

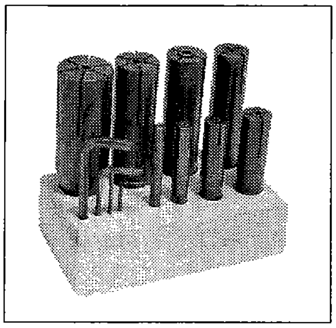


10.12 Mount workpieces too long for a chuck on a mandrel.

shafts that are apt to spring out of alignment from the thrust of the tool. The purpose of a rest is to support the workpiece and maintain it in accurate alignment for machining. Rests are classed as steady rests or follow rests.

Steady rests. Steady rests mount on the lathe bed (*Figure* 10.14). Clamped over the ways, they provide three bearing surfaces. These surfaces bear down lightly but rigidly against the surface of the shaft and keep it from moving out of the line of the lathe centers. You can place a steady rest anywhere along the bed where it will best support and steady the workpiece without interfering with the operation.

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



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

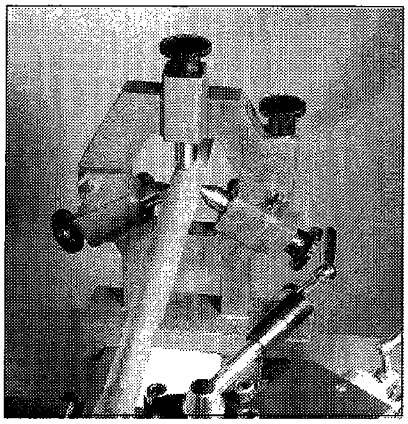
Follow rests. Long or slender shafts that are apt to spring out of alignment by the thrust of the cutting tool often require a follow rest (*Figure* 10.15). Follow rests mount on the carriage of the lathe and move with the tool, backing up the workpiece opposite the point of the tool thrust. They have two adjustable supporting jaws, one holding the work to keep it from climbing up on the tool and the other behind the work to counter the thrust of the tool.

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

10.7 Setting up work in a chuck

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

For ordinary use, there are two standard types of headstock chucks. The four-jaw independent lathe chuck has four holding jaws that can operate independently and adjust to hold round, square, eccentric, or odd-shaped work (*Figure* 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).



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

10.8 Mounting work in a four-jaw independent lathe chuck

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

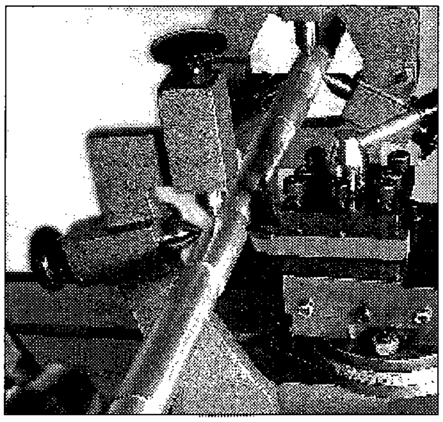
To place work in a chuck, follow these steps:

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

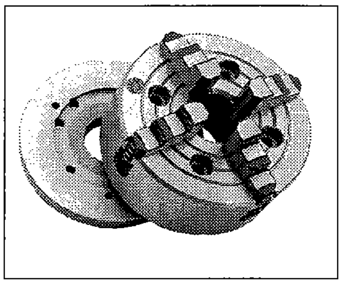
- Place the work in the chuck and grip it. Turn up the opposing jaws a uniform number of turns with the key provided. This will hold the work in position. Then bring in the other pair of opposing jaws the same way.

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

- Guided by the chalk marks, readjust the jaws until a chalk line will carry completely around the work. Then tighten all the jaws securely.



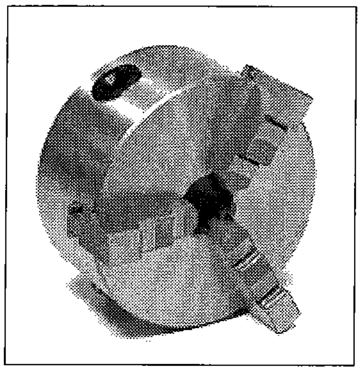
10.15 Follow rests mount on the lathe carriage and move with the tool.



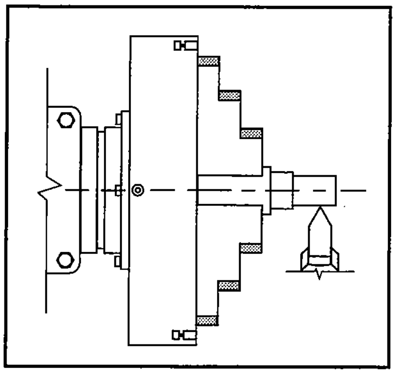
10.16 Four-jaw independent lathe chucks hold round, square, eccentric, or odd-shaped workpieces.

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.

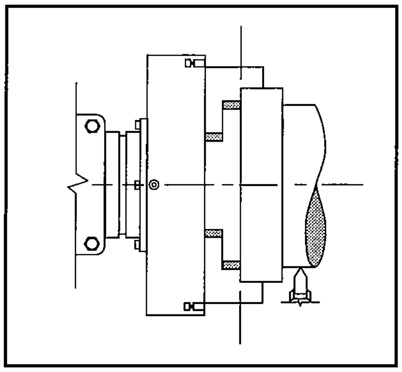
When making several identical pieces, after completing each workpiece release only two adjoining jaws, leaving the others to hold the center. The jaws of the four-jaw independent chuck are reversible. You can insert them with high steps to the inside or outside.



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



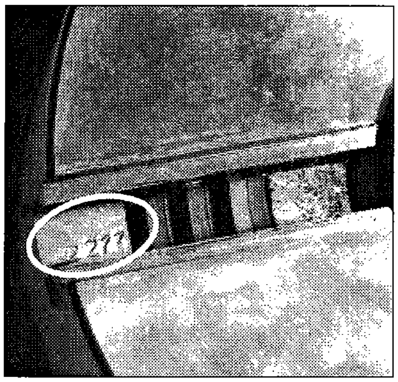
10.18 For short, small-diameter workpieces, insert the jaws with high ends to the center.



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

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

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

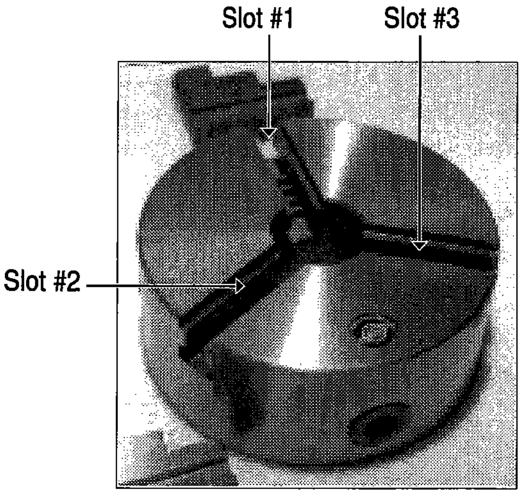


10.20 A serial number is stamped in the #1 slot of the three-jaw chuck.

10.9 Mounting work in a three-jaw universal chuck

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

- On three-jaw chucks, the key moves all the jaws at once.
- You need not center or check for concentricity because these chucks center automatically.
- Jaws are not reversible. Each chuck comes with two sets of jaws. One is for setups with high steps toward the inside (inside jaws), the other for mounting in the chuck with high steps to the outside (outside jaws).
- When installing the chuck jaws on a three-jaw chuck, install them in numerical order and counterclockwise rotation.



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.

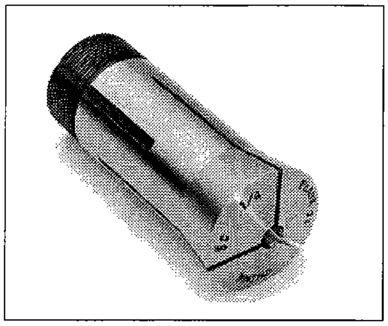
Each jaw is stamped with a serial number and jaw number (#1, #2, or #3). The slots in the chuck are not numbered, but there is a serial number stamped at the #1 slot (*Figure* 10.20). With the #1 slot in the 12:00 position, the #2 slot is at 8:00 and the #3 slot at 4:00 (*Figure* 10.21).

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.

10.10 Collets and collet attachments

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



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 spindle's MT5 taper to the collet's MT3 taper. A drawbar holds the collet in place.

10.11 Toolpost grinders

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

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

SECTION ELEVEN
LATHE TURNING

11.1 Rough turning

In turning a shaft to size and shape where you have to cut away a lot of stock, take heavy, rough cuts to get the work done in the least time. With the MI-1720cnc 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.

Remember caution must be taken to not run the powerfeed past their limits of travel. As part ofthe notrmal operation, procedures, run each axis through the entire length of the proposed machining operation before engaging the powerfeed to assure there is suficient travel to accomplish for the desired task. Failure to so so could result in running the power feed to the end of its mechnical 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.

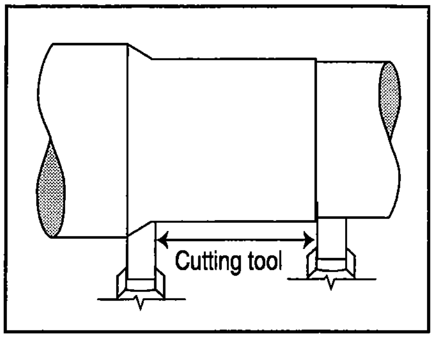
Engage the longitudinal feed by moving the powerfeed engagement lever left and then pulling the half nut lever up. Always cut deeply enough to reach below the scale on oxidized bars or iron castings. Hard, oxidized surfaces dull tools rapidly.

11.2 Finish turning

After you've rough-turned the workpiece to 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 machined surface.

Check the diameters carefully with a caliper or micrometer to be sure you are working to proper dimensions. Remember: the diameter will reduce twice the thickness of the cut.

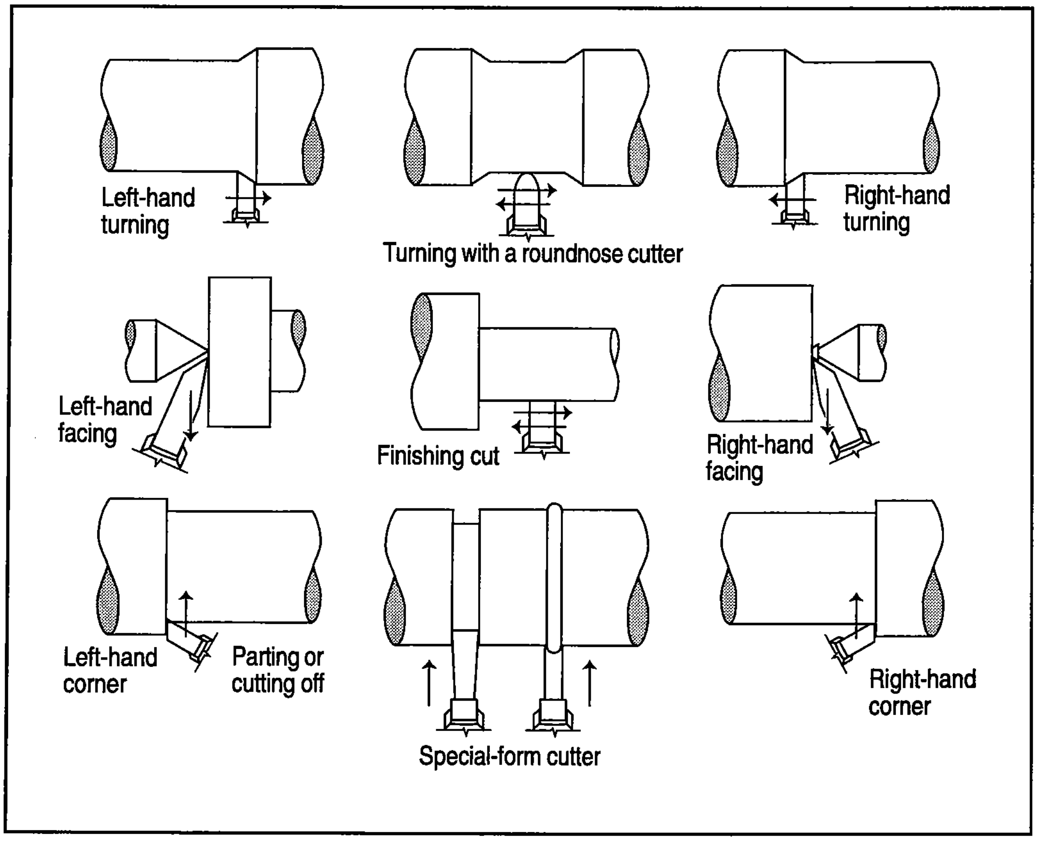
For rough turning, most machinists prefer a deep cut and a comparatively fine feed, but the reverse is true for finishing cuts. They usually use a very light crossfeed and a coarse transverse feed with a cutting edge wider than the feed per revolution. In *Figure* 11.2, the left-hand tool illustrates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



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

11.3 Turning to shapes

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



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

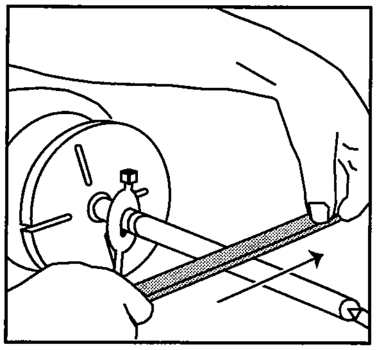
11.4 Maching Square Cornors

To machine an accurate cornor, follow these steps:

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

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

- 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.

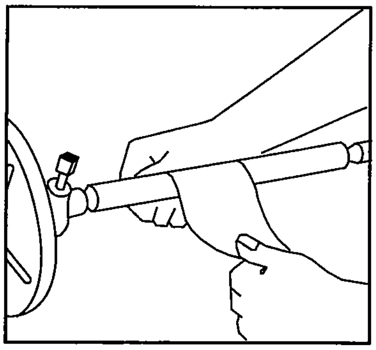


11.4 With a file, take full strokes at an oblique angle; never hold the file still.

11.5 Finishing and polishing

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

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



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

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* 11.5).

11.6 Taper turning

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

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

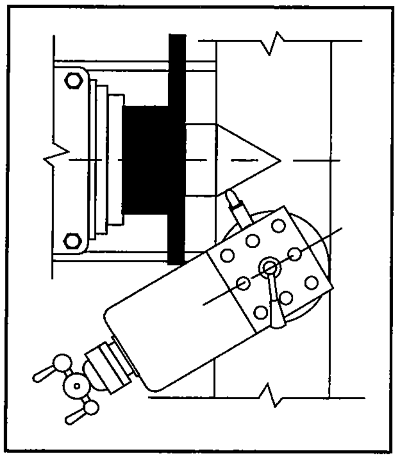


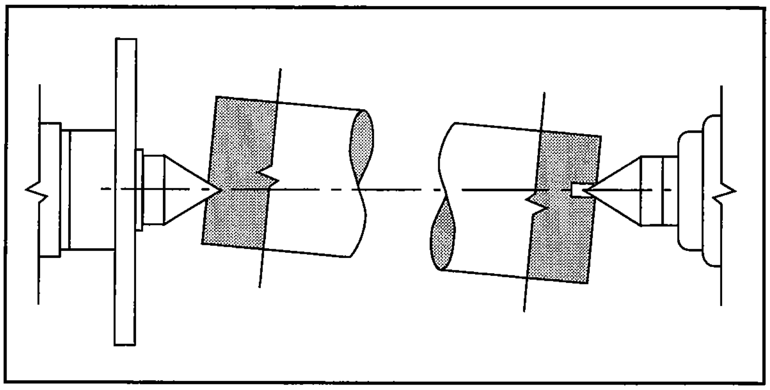
Figure 11.2 Tapers cut with the compound rest are usually short, abrupt angles.

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* 11.7). The tailstock has a set-over scale calibrated both forward and backward from the straight turning or zeroing point for measuring set-over distances.

To offset the tailstock, 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.

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.

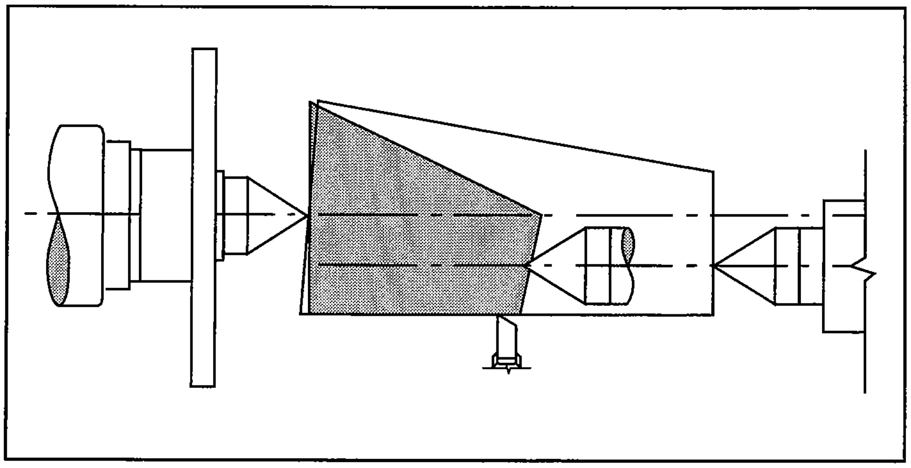
The distance of tailstock setover needed to machine any given taper depends on three factors:



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

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

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



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

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

11.7 Boring a tapered hole

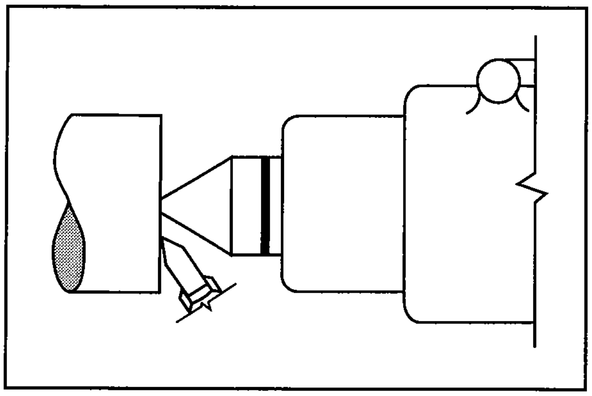
Boring a tapered hole involves setting the compound at the desired degree of taper and feeding the tool with the compound rest. Make sure the compound rest is set at half the degrees of the angle of the completed taper hole. You can also use a taper attachment to bore a tapered hole.

SECTION TWELVE
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).

12.1 Facing across the chuck



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

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 . 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.

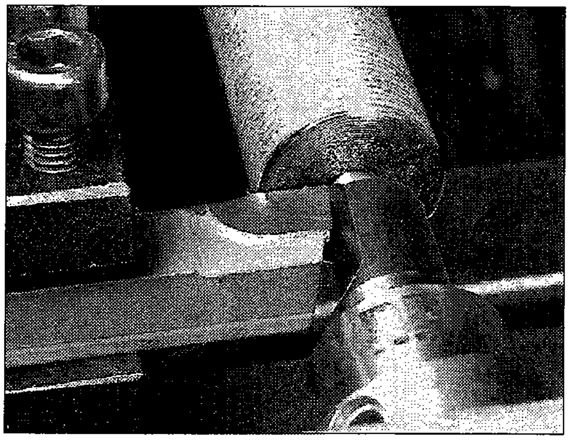
If you must finish the ends of the shaft, use a half-center (*Figure* 12.2). This lets you extend the tool across the entire face of the work.

To use the powerfeed for facing, place the powerfeed selector into position III before the lathe is turned on. Once teh cutter has been positioned as per the above paragraph, move the leadscrew lever to the left and pull the crossfeed. Push the button in at the endo of the cut to stop the cutter travel.

12.2 Knurling

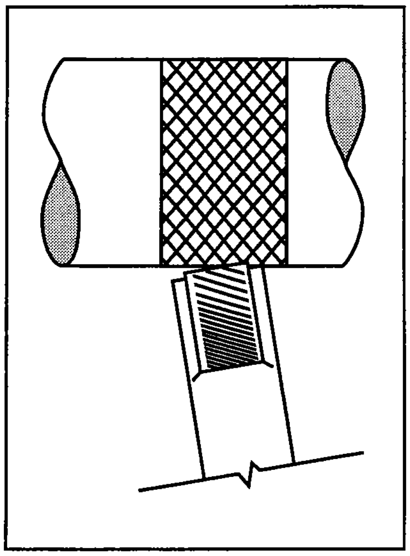
Strictly speaking, knurling is not a machining operation because no metal is cut. It is a forming operation in which patterned knurls are pressed into the work, depressing and raising the surface of the metal into a pattern. As with all other forming operations, your work can be no better than the pattern,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

To make a true, uniform knurl, maintain uniform pressure on
knurls. Select a self-centering knurling tool that equalizes
e on the knurls automatically and is strong enough to
end and side thrusts. Operate the lathe at the slowest
m).



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

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



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

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.

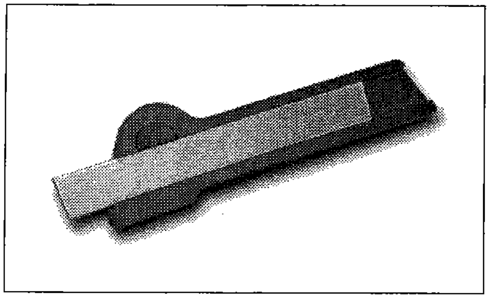
Use plenty of oil, lubricating both knurl and workpiece. Then start the lathe and engage the automatic feed, moving the knurls across the portion to be knurled. When you reach the left scribe line, force the tool into the work another 1/64", reverse the lathe without removing the tool, and feed it back to the starting point. Feed both ways using the automatic longitudinal feed. Once across, each way, usually makes a good knurl.

SECTION THIRTEEN

CUTTING OFF OR PARTING WITH A LATHE

You can cut off in a lathe only when holding one end of the work rigidly, as in a chuck. It is not practical for long workpieces held between centers because the workpiece is not supported closely with a rest and the free section is long enough to sag and pinch the blade. Cutting off requires a tight lathe without excess play in the spindle, compound, carriage, or toolpost. Looseness will almost certainly cause chatter. Cutting off also requires a narrow cutting edge with ample (5–10°) side clearance, which should feed into the work slowly to prevent hogging in. Once considered a difficult, costly operation, cutting off became much simpler with development of narrow tools with special cut-off blades (*Figure* 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 workpiece, just beyond its center. The tool should feed to the workpiece on exact center, slowly and evenly with the crossfeed. If the tool hogs in and the spindle stops rotating, turn off the motor and reverse the spindle by hand before backing the tool out with the crossfeed.



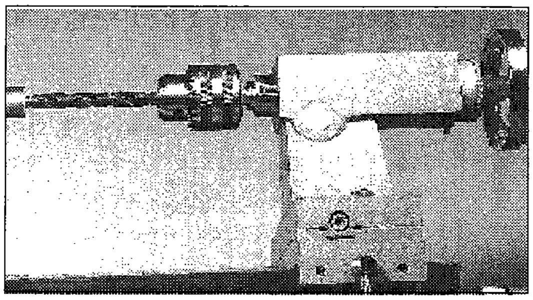
13.1 Specially designed tools like this one make cutting off easier.

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.

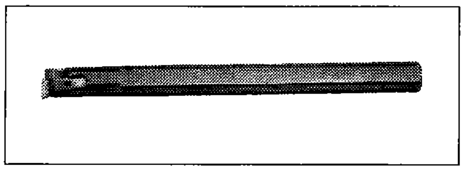
SECTION FOURTEEN
LATHE DRILLING AND BORING

You can lathe drill on the MI-1720CNC in two ways, holding the drill stationary and revolving the workpiece, or holding the workpiece stationary and revolving the drill. Holding the drill sta-tionary in a tailstock chuck gives a straighter hole (*Figure* 14.1). Without changing setup and recentering, 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.


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.


14.2 A tool with a smooth-ended bar won't mar the workpiece.

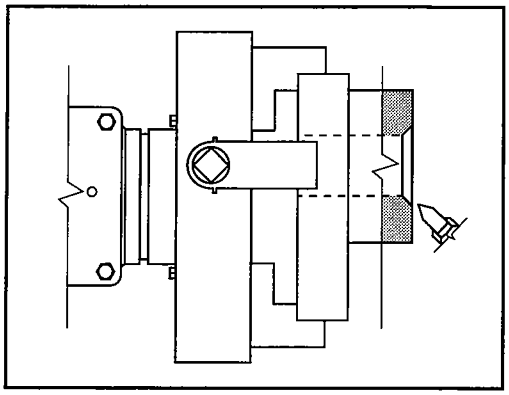
With the workpiece in the headstock and the drill in the tailstock chuck, feed the drill into the workpiece by advancing the tailstock ram. Do this by turning the tailstock handwheel. Make a locating center for the drill point, or even a countersunk center for large diameters, to keep the drill from creeping.

14.1 Reaming

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

Lathe reaming is usually done with 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.

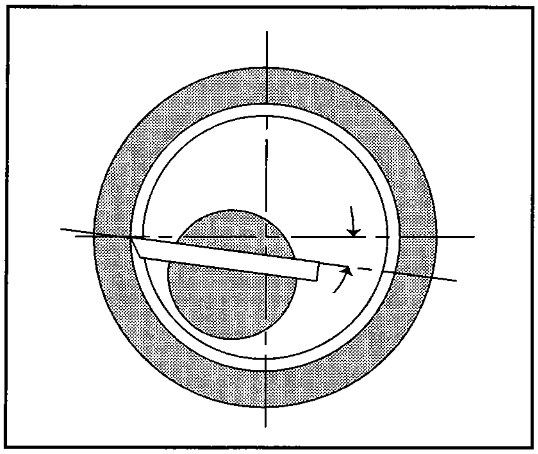
14.2 Boring


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

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, #K99-125).

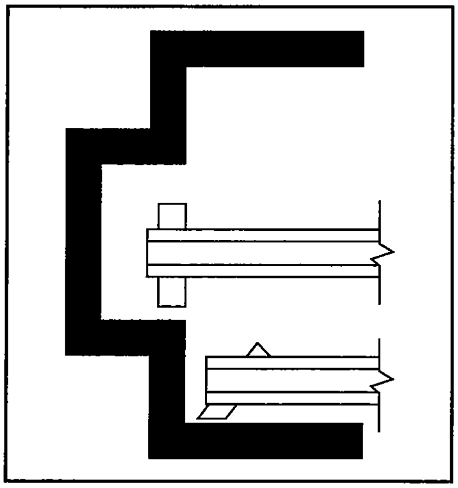
It is also wise to select tools with smooth-ended bars without a projecting nut or hardened edge that might mar the work (*Figure* 14.2). Most boring tools have only one cutting edge. There *are* double-end cutters, however, and they offer advantages in special instances. In grinding cutters, allow sufficient end rake to provide clearance from the internal diameter.



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

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 (*Figure* 14.3) before introducing the boring tool.

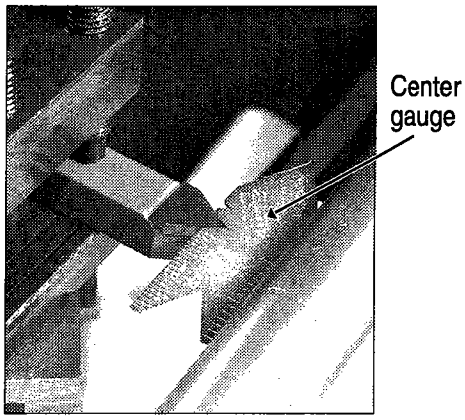
With the boring toolholder set up (in the toolpost or toolpost T-slot, depending on the type), select the largest-diameter boring bar whose cutter the bore will accept. Extend the bar from the holder just enough to reach the full depth to be machined and still allow tool clearance. Except when using the adjustable boring tool (usually for very-large-diameter work), feed the bar into the hole, parallel to the hole's axis. The cutting edge engages the work along a line in the mounted plane of the lathe centers with the bar positioned to give the cutter a top rake of approximately 14° from the radius at the cutting point (*Figure* 14.4). This takes into consideration the ground angle (top rake) of the cutter itself.



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

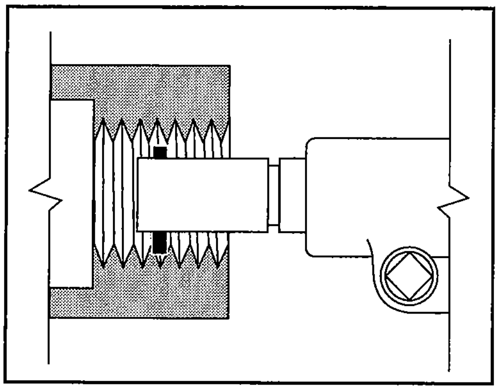
For straight longitudinal cuts, you can hold the cutter close up, therefore more rigidly, if it's at a 90° angle to the bar. For machining ends of a bar, however, you need a boring bar that holds the cutter at an angle or angles so the cutter extends beyond the end of the bar (*Figure* 14.5). For maximum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



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

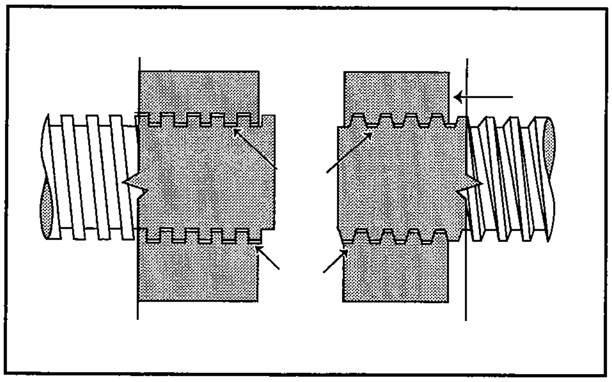
After the last finish cut, it is common to reverse the feed and take one last, fine cut with the tool coming out of the work. This last cut, taken without movement of the cross-feed, avoids a slightly undersized hole because you compensate for any spring in the bar.



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

14.3 Cutting internal threads

Internal thread cutting is like external thread cutting, except you have the clearance restrictions and tool problems of boring. You use the same toolholders, but the cutters have thread forms and are fed at thread-cutting ratios of feed to spindle revolutions.



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

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 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.

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.

14.4 Cutting special-form internal threads

You can cut internal forms in all the thread forms used for external threads. There is only one factor that calls for special attention in cutting special-shaped internal threads: the difference of clearances between the nut and screw recommended for different thread types (*Figure* 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 screw's outside diameter.

SECTION FIFTEEN

CHANGING GEARS

To change geras on the MI-1720CNC, follow these steps. You will need a 17 mm wrench and a 5 mm allen wrench.

Remove the nuts from A and D gear shafts, loosen the B/C gearshaft by turning it counter closckwise with the allen wrench. This lets the shaft slide freely along the bracked for easy gear removal.

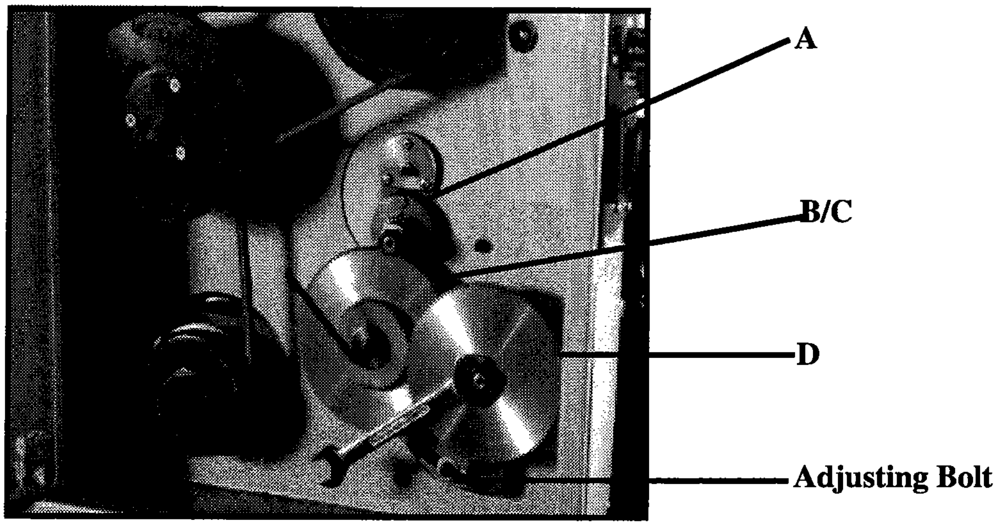


Figure15.1 SlideB/C gearshaft until theC gear meshes with the D gear

Slide the B/C gearsahft until the C gear meeshes proberly with the D gear and tighten in with the allen wrench. (Figure 15:1).

Place the selected A gear on tha gearshaft and replace the nut and washer.

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

SECTION SIXTEEN
CUTTING THREADS ON THE MIDAS 1720CNC

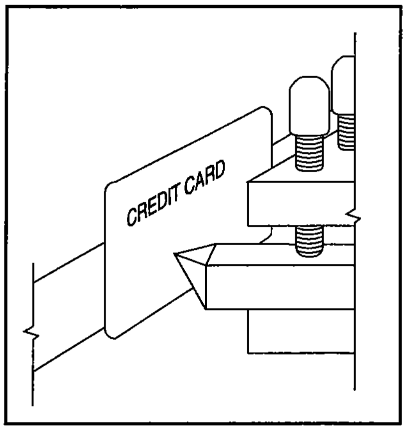
16.1 Threading terms

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

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

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

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



16.1 Check dead center with a credit card.

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-1720CNC 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.

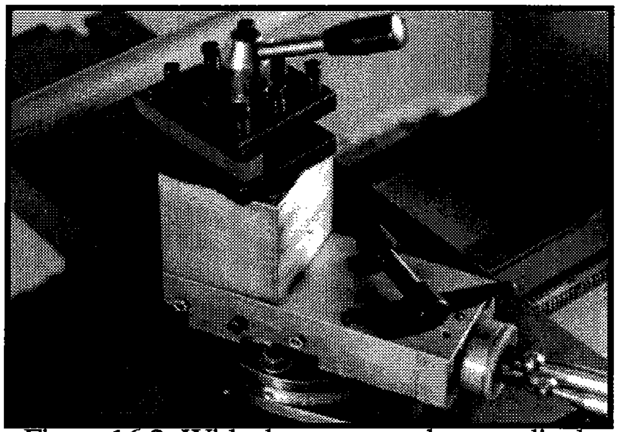


Figure 16.2 With the compound perpendiculat to the line of centers, rotate it 29.5 degrees to the right.

For most work, beginners use the Unified National Standard, which is a V-form thread slightly flat on top and at the root. Screw threads are usually referred to by pitch numbers, such as 18 or 24, meaning 18 or 24 threads per inch (tpi). The MI-1720CNC cuts standard threads in pitches from 4 to 120 tpi and metric threads from 0.75 to 6 mm. (See Figure 16:9)

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

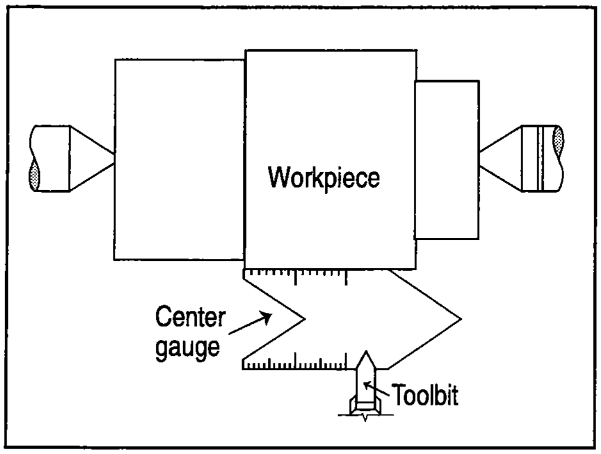
Thread charts on the front of your machine show both inch and metric measurements. The inch chart shows the tpi from 4 to 120. The metric shows the distance from thread crest to crest from 0.075 to 6 mm.

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.



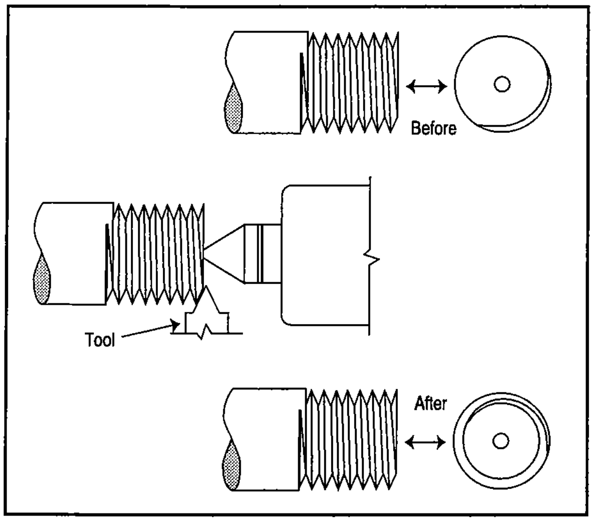
16.3 Using a center gauge, set the threading tool perpendicular to the work piece.

Set the compound perpendicular to the line of centers and rotate it 29-1/2° to the right (*Figure* 16.2). 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.

16.2 Cutting right-hand threads

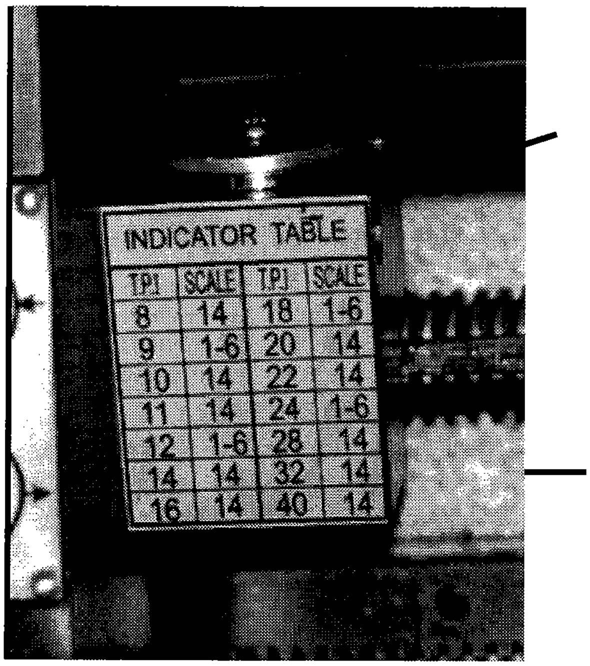
Now you are ready to cut right-hand threads. First, advance the tool so it just touches the workpiece and turn the compound calibration back to zero. Then, using the compound feed, feed in the tool 0.002". Next, turn on the lathe and engage the powerfeed lever by carefully turning it one-quarter turn to the left. 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.



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

16.3 Using the Threading Dial



INDICATOR TABLE

TPI	SCALE	TPI	SCALE
8	14	18	1-6
9	1-6	20	14
10	14	22	14
11	14	24	1-6
12	1-6	28	14
14	14	32	14
16	14	40	14

Figure 16:5 Note numbers on top of the threading dail when cutting threads.

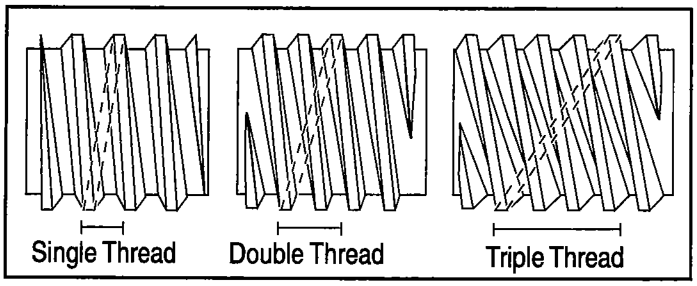
The threading dail performs the function of idicating the proper time to engage the half-nuts so that the cutting tool will enter the same groove of the thread of each successive cutting pass. This allows the half-nut to be disengaged resulting in an easier method of threading. At the end fo each consecutive 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 fo 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 houseing of the dial (Figure 16.5). The indicator table (Figure 16.6) on the front of the trheading dial show the selection for the different thread pitches. Find the desired thread pitch under the "TPI" column and engage the half-nut at the proper numbeers 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. 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 in not successful, the thread dial can not abe used and the instructions from the previous section should be followed. Cutting of metric threads cannot be done with the trhead dial.

INDICATOR SCALE

TPI	SCALE	TPI	SCALE	TPI	SCALE	TPI	SCALE
8	1,4	12	1-6	20	1,4	32	1,4
9	1-6	14	1,4	22	1,4	40	1,4
10	1,4	16	1,4	24	1-6		
11	1,4	18	1-6	28	1,4		

Fiqure 16.6 Threading Dial Indicator Scale

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.



16.7 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 (*Figure* 16.4).

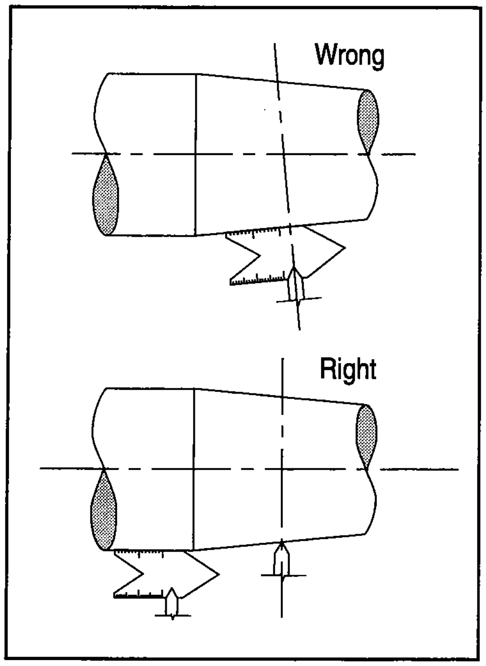
16.4 Cutting multiple threads

Cut multiple threads (*Figure* 16:7) 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.

16.5 What *not* to do when cutting threads

Do not disengage the powerfeed engagement 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



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

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 workpiece back in the centers exactly as it had been. Then remove the chips and test again. Repeat until you are finished.

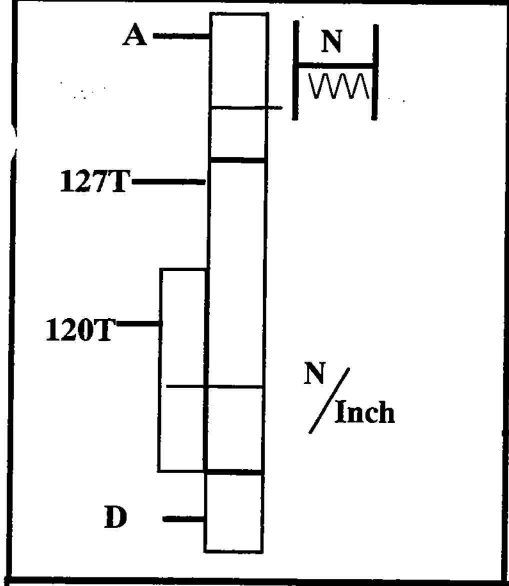
16.6 Finishing off a threaded end

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

16.7 Cutting threads on a taper

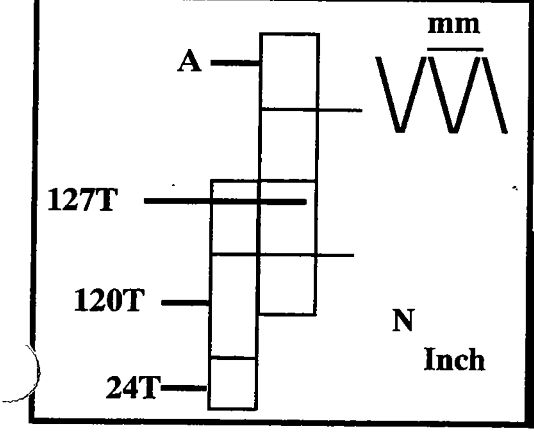
Cut threads on a taper the same as on a straight shaft, except in the setup of the tool. Set the threading tool at 90° to the axis of the taper, rather than at 90° to its surface (*Figure* 14.8).

Figure 16:9 Threading Charts



MIDAS 1720 CNC
THREAD CHART FOR
INCH THREADS

A \ D		24	27	30	33	36	39	42	48	60
72	I	4	4.5	5		6	6.5	7	8	10
	II	8	9	10	11	12	13	14	16	20
	III	16	18	20	22	24	26	28	32	40
24	I					18		24		30
	II		27	30	33	36	39	42	48	60
	III		54	60	66	72	78	84	96	120



MIDAS 1720 CNC
THREAD CHART FOR
METRIC THREADS

A	36	42	48	60	72
I	0.75		1	1.25	1.5
II	1.5	1.75	2	2.5	3
III	3	3.5	4	5	6

SECTION SEVENTEEN

MILLING

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

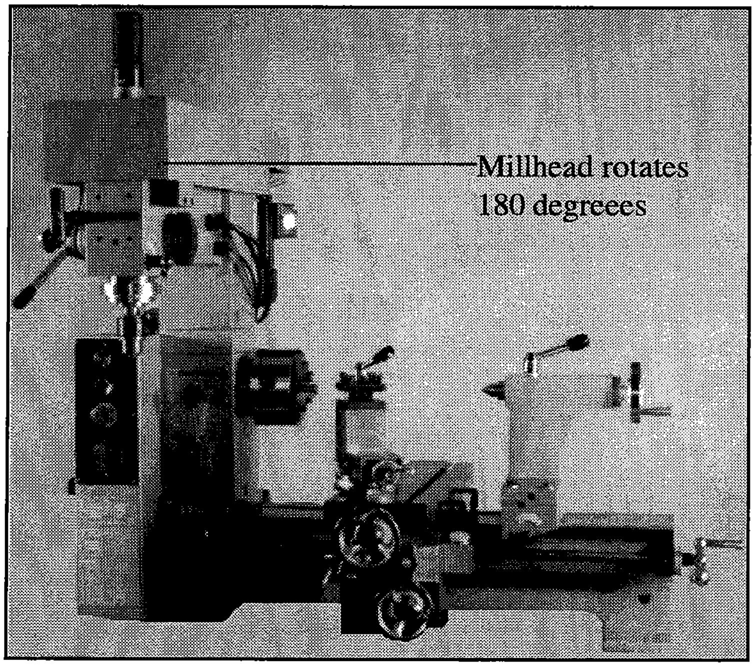
Machinists use mills to machine flat surfaces, both horizontal and vertical, and to make shoulders, grooves, fillets, keyways, T-slots, and dovetails. They can also make curved and irregular surfaces and machine accurate holes. Its variety of machining operations and high metal-removal rates rank the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

The millhead rotates 180 degrees. 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 millhead, loosen dual locks located on front of machine and push in the desired direction. Remmeber you millhead swivels 180 degrees, do not force it past its mechanicals stops. (Figure 17:1).

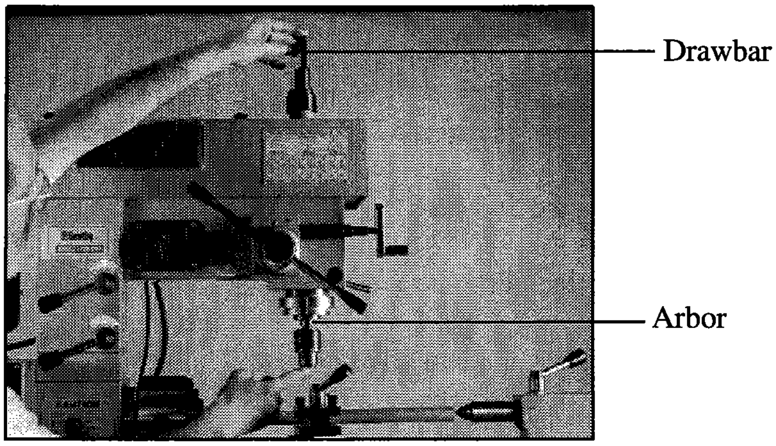
Figure 17:1 Millhead Rotated 90 Degrees



17:1 Holding Milling Cutters

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

Arbors

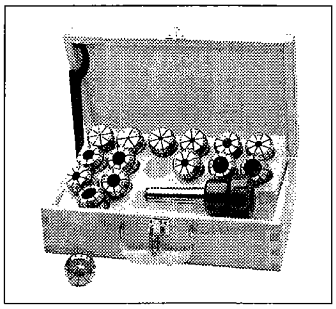


17:2 The arbor stays in place via a drawbar.

Arbors come in different sizes and lengths, with one end tapered to fit the bore in the end of the machine spindle. The arbor of the MI-1720CNC, 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 (*Figure* 17.1).

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

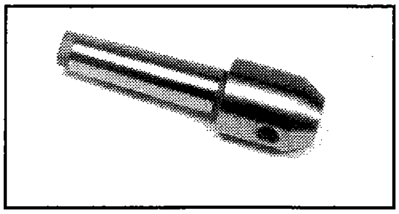
Collets and holders



17:3 Spring collets, which fit into the mill spindle, hold straight-shanked end mills.

Straight-shank end mills fit into spring collets (*Figure* 17.2) or end mill holders (*Figure* 17.3). Their precision-ground shanks go into the mill spindle. When you tighten a spring collet, its hole reduces in size and the collet grips the end mill shank evenly. 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.

Adapters



17:4 End mill holders also receive straight-shanked end mills.

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

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

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

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

- Remove the drawbar.
- Extend the mill spindle to expose the outer taper drift slot.
- Rotate the spindle to align outer and inner taper drift slots. You will be able to see the end of the adapter through both slots.
- Insert the drift in the slot.
- Holding the adapter with one hand, use a nonmarring hammer (rubber, dead-blow, or brass) to drive the drift into the slot. The taper on the tool will release and the adapter drop out.

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

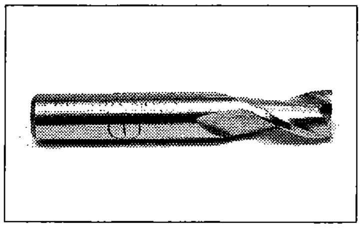
17:2 Miling Cutters

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



17:5 Two-flute end mills have two teeth that cut to the center of the mill.

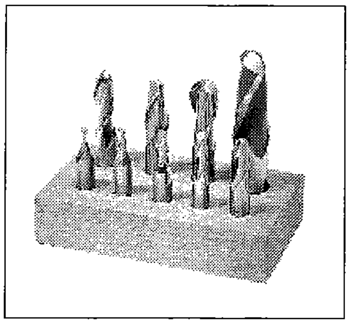
End mill cutters cut on their ends and sides. They are either solid (cut from a single piece of material) or shell (separate cutter body and shank). They have two, three, four, or more teeth and may do right or left-handed cutting. Their flute twist or helix may also be right or left-handed. Solid end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

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

- *Two-flute,* or center-cutting, end mills (*Figure* 17:5) have two teeth that cut to the center of the mill. They may feed into the work like a drill (called plunge milling), then go lengthwise to form a slot. Teeth may be on one end (single-ended) or both ends (double-ended).

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

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



17:6 Ball end mills cut slots or fillets with a radius bottom.

Ball end mills (*Figure* 17:7) cut slots or fillets with a radius bottom, round out pockets and bottoms of holes, and do diesinking and diemaking. Four-fluted ball end mills with center cutting lips are available.

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.

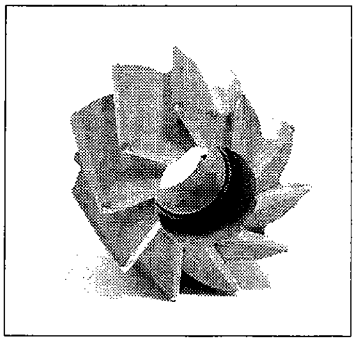
T-slot cutters cut T-slots. After machining a groove for the narrow part of the T-slot with an end or side mill, finish up with the T-slot cutter.

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

Shell end mills (*Figure* 17:7), which mill wide, flat surfaces, have a hole for mounting on a short arbor. The center of the shell is recessed to provide space for the screw or nut that fastens the cutter to the arbor. The teeth are usually helical, and diameters are as large as 6".

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

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



17:7 Shell end mills mill wide, flat surfaces and mount on arbors.

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.

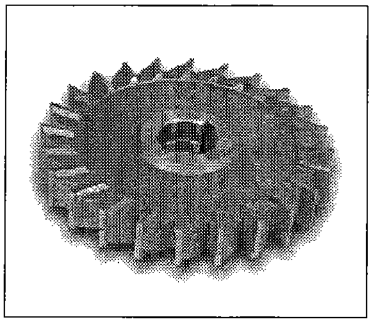
- *Light-duty* plain cutters for light cuts and fine feeds come in two forms. Narrow ones have straight teeth parallel to the cutter axis. Wide ones have helical teeth at a 25° angle. Features include ease of starting cuts, little chatter, and good surface finishes.

- *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 (*Figure* 17:8). 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.



17:8 Side milling cutters are similar to plain milling cutter, but they have teeth on one or both sides.

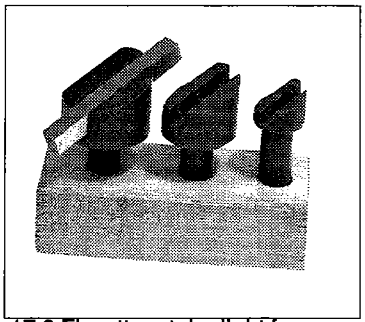
- *Plain* side milling cutters have straight teeth on the periphery and both sides. Side teeth taper toward the center of the cutter, giving side relief or clearance.

- *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 keyway and slotting cuts.

Slitting saws

Slitting saws do narrow slotting and cut-off operations.



17:9 Flycutters take light face cuts from large surface areas.

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

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

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

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

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.*

17:3 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.

17:4 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.

17:5 Speeds and Feeds for Millling

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

There is no exact right cutting speed for milling a particular material (*Figure* 21.1). Machinists usually start with an average speed, then increase or decrease it as appropriate. For light cuts, use the upper end. Use the lower end for heavy cuts and when you don't use cutting fluid.

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

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

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

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

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

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

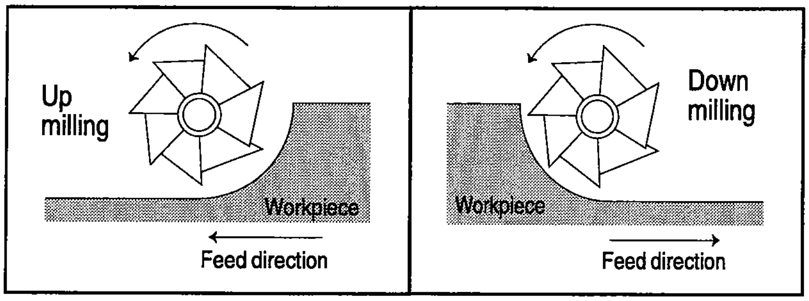
Up milling. In up milling, forces on the workpiece tend to pull it out of the vise or fixture holding it, so fasten it securely. These forces also push the workpiece away from the cutter, which eliminates backlash. Up milling is advised for milling cast iron, softer steels, and other ductile materials.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

17.10 Recommended Cutting Speeds for Milling (fpm)

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



17:4 In up milling (left), the workpiece feeds into the cutter in the opposite direction of the cutter's revolutions. In down milling (right), the workpiece feeds into the cutter in the same direction as the cutter is turning.

(ipm). You determine feed rate by the speed of the cutter in rpm and the number of teeth in the cutter.

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

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

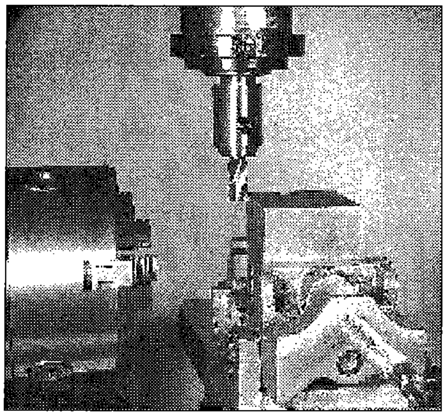
If you know the feed in inches per tooth, use this formula to calculate table feed rate in inches per minute (ipm):

ipm = ipt 5 N 5 rpm

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

17:6 Common milling operations

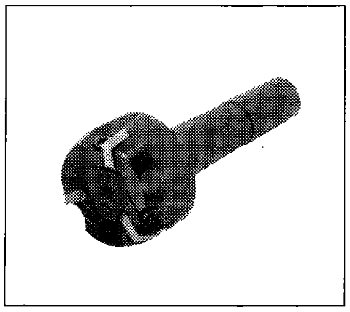
Milling flat surfaces



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

One way to mill a flat surface is by plane milling (*Figure* 17.11). Adjust the milling cutter vertically to give the needed depth of cut while the workpiece is held on the table and slowly feed it horizontally. Every tooth on the periphery of the cutter removes a chip every revolution. Milling wide, flat surfaces this way is called slab milling.

Another way to mill flat surfaces is by face milling. In this method, the cutter teeth operate at right angles to the cutter axis. Inserted-tooth face-milling cutters (*Figure* 17:12) face-mill large surfaces.



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.

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.

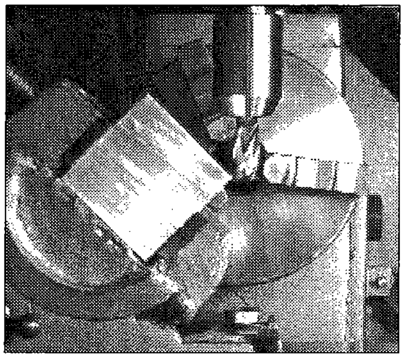
Plunge cutting is efficient for removing material quickly on low horsepower. Plunge the end mill a predetermined width and depth, retract it, then advance and plunge it again repeatedly. The maximum cutting force is in the machine's strongest (axial) direction.

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.



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

SECTION EIGHTEEN

WORKHOLDING

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

18.1 Mounting to the table

If you need to align the workpiece to the table, place it against stops that exactly fit the table's T-slots. Another way is to measure in from the edge of the table to the workpiece. Be sure the table and workpiece are clean and free of burrs. Another method is to use the face of the spindle plate, chuck or taistock as a quick reference surface.

18.2 Using a vise

Vise sizes are designated by the width of the vise jaw in inches or millimeters. Plain and swivel vises range from 3 to 10" (76 to 254 mm). 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.

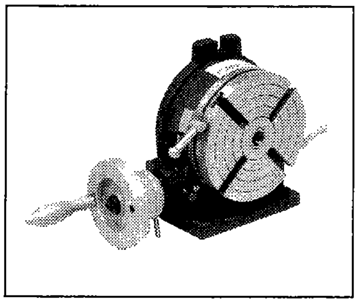
Using special fixtures. Clamp both workpiece and fixture securely in place. Be sure they are clean. Watch them carefully during machining; a loose fixture or workpiece can be disastrous.

18.3 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.

18.4 Rotary tables



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

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

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

SECTION NINETEEN

TROUBLESHOOTING

19.1 Powerfeed and thread cutting

Powerfeed does not move carriage

Cause	**Solution**
• Carriage locked	Unlock carriage
• Speed selector not engaged	Select speed I or II or III
• Leadscrew lever not engaged	Move lever to the left
• Gears not meshing or teeth missing	Check gears and mesh
• Half-nut fully engaged	Keep half nut engaged
• Sheard Pin on left enf of leadscrew	Replace Pin

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 (Figure 16.6)

19.2 Carriage/milling table

Powerfeed doen't move table

Cause	Solution
• Carriage	Unlock Carriage
• Speed Selector not engaged	Select Speed I, II or III
• Leadscrew lever not engaged	Move lever to the left
• Gear not meshing or teeth missing	Check and adjsut gears

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

19.3 Lathe turning

Cut is rough

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

Work has unwanted taper

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

19.4 Milling

Tool chatters

Cause	Solution
• Gibs too loole on cross slide, cmpoung or carriage	Readjsut Gibs
• Unused feeds nto 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

19.5 Drilling

Hole is off center or bit wandres

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

19.6 Drive system

Turn on machine and nothing happens

Cause	**Solution**
• End Door Open	Close and latch door
• Machine unplugged	Plug in machine
• Loose electrical connections	Tighten wiring connections

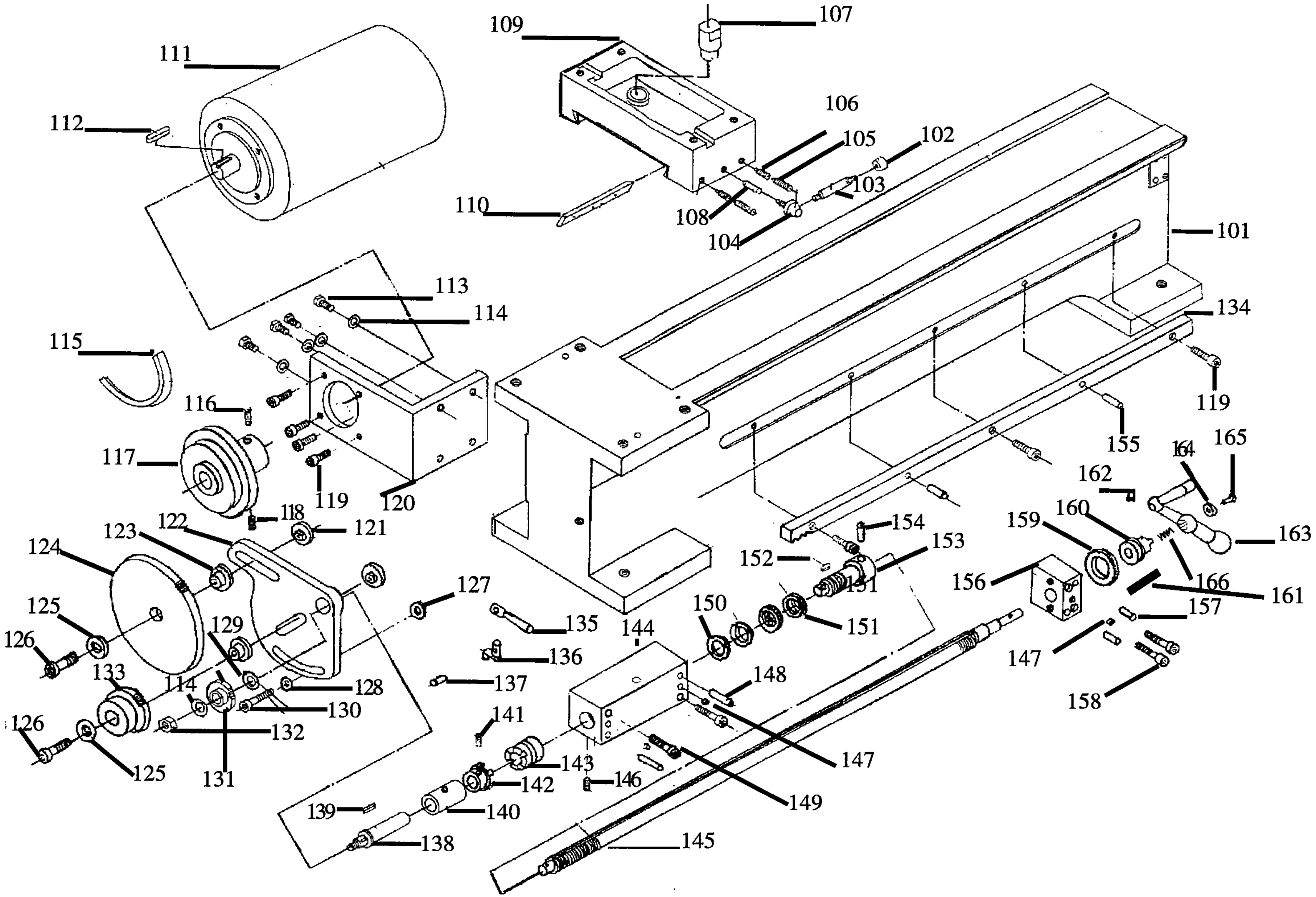
SECTION TWENTY

PARTS LISTS WITH SCHEMATICS

Diagram 1:0	Lathe Bed and Tailstock
Diagram 2:0	Headstock
Diagram 3:0	Control Panel and Pulley Box
Diagram 4:0	Lathe Idler Pulley
Diagram 5:0	Millhead
5:1	Mill Quill and Spindle
5:2	Mill Idler Pulley
Diagram 6:0	Mill Fine Feed/Drill Press
Diagram 7:0	Crossfeed and Apron
Diagram 8:0	CNC Mount (Y Axis)
8:1	CNC Mount (Z Axis)
8:2	CNC Mount (X Axis)
Diagram 9:0	Tailstock
Diagaram 10:0	Threading Dial
Diagram 11:0	Leadscrew Cover
Diagram 12:0	Steady Rest
Diagram 13:0	Follow Rest
Diagarm 14:0	Wiring Diagram

SCHEMATICS

Diagram 1:0 Lathe Bed and Tailstock Base



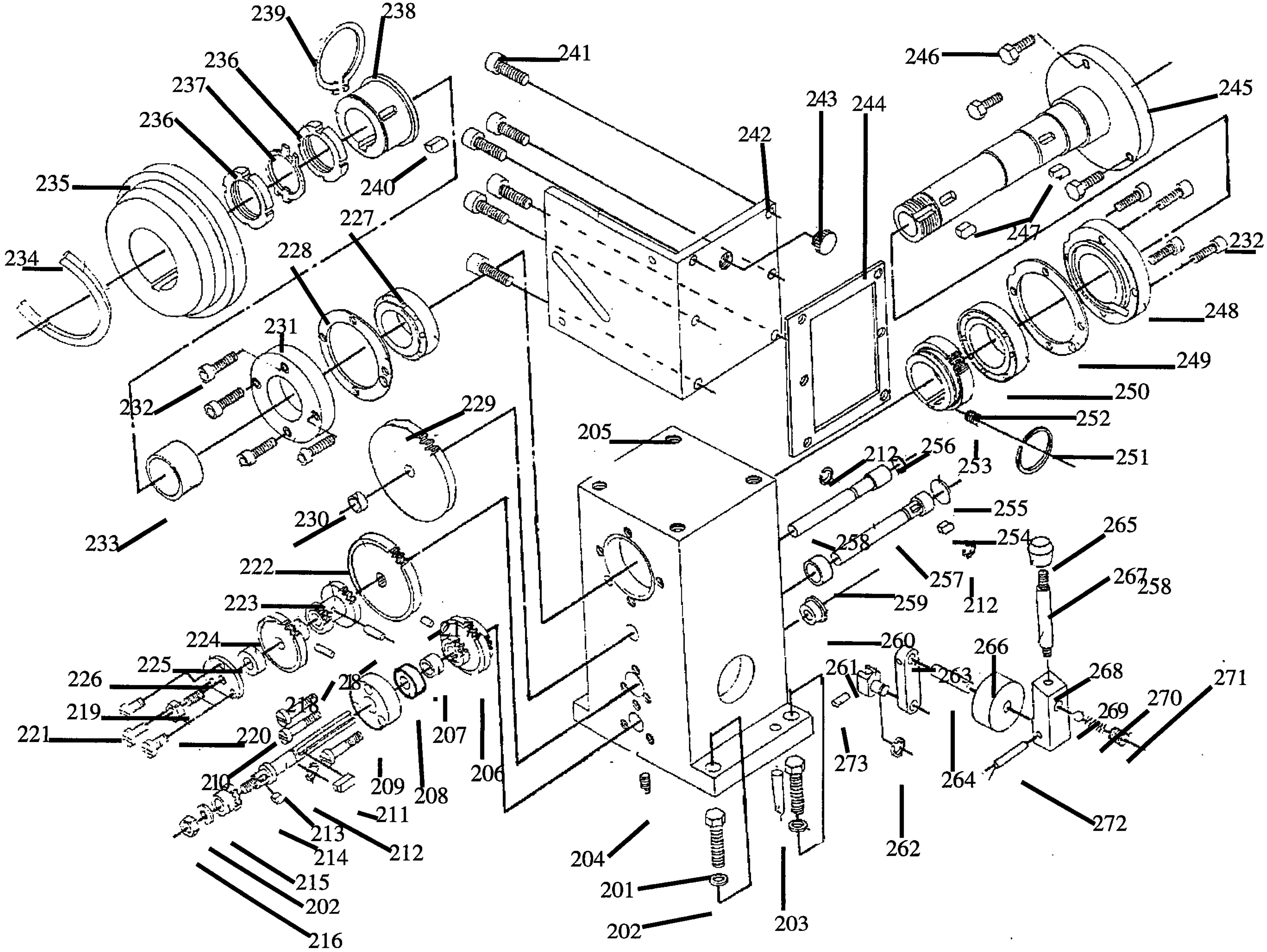
Midas parts list

Smithy Number	Picture Number	Description
M10101	101	bed
M10102	102	handle knob M6x20
M10103	103	handle
M10104	104	handle seat
S11825	105	hex socket set screw M8x25
S12268	106	set screw M8x14
M11107	107	adjusting block
M11108	108	pin
M11109	109	tailstock carriage
M11110	110	gib
M12111	111	motor YCYS7144
S21230	112	key A5x16
S12125	113	hex head bolt M10x25
S18106	114	washer ?16
M12115	115	belt Z630
S12088	116	set screw M8x8
M12117	117	pulley
S12068	118	set screw M8x6
S12269	119	hex socket head screw 6x20
M12120	120	motor mount
M12121	121	T-nut
M12122	122	bracket
M12123	123	T-bushing
See Below	124	B/C gear
M12124-127/120	124-127/120	B/C gear 127/120
M124124-120/60	124-120-60	B/C gear 120/60
S18110	125	washer M6
S12041	126	hex socket head screw 6x40
S18110	127	washer M6
S18110	128	washer M6
S18110	129	washer M6
S12021	130	hex socket head screw M6x30
M10131	131	gear
S18155	132	hex nut M10
M10133	133	gear

M10134	134	rack
M10135	135	handle A M10 x 80
M10136	136	eccentric lever
S22203	137	roll pin M3 x 20
M10138	138	transmission shaft
S21522	139	key A4x12
M10141	141	roll pin M 5 x 22
M10142	142	clutch jaw A
M10143	143	clutch jaw B
M10144	144	transmission box
M10145	145	leadscrew
S18096	146	hex socket set screw M6 x 8
C30050	147	oiler M6
S22555	148	taper pin B M5 x 5
S12050	149	hex socket head screw M6 x 5 0
M10150	150	spanner nut M17 x 1.5
M10151	151	ball bearing 8103 17x30x9 M
S21512	152	key A5 x12
M10153	153	shaft connector
S22442	154	roll pin M4 x 24
S22305	155	roll pin M5 x
M10156	156	leadscrew seat
S22655	157	taper pin A M6 x 55

This page is left blank intentionally blank.

Diagram 2:0 Headstock

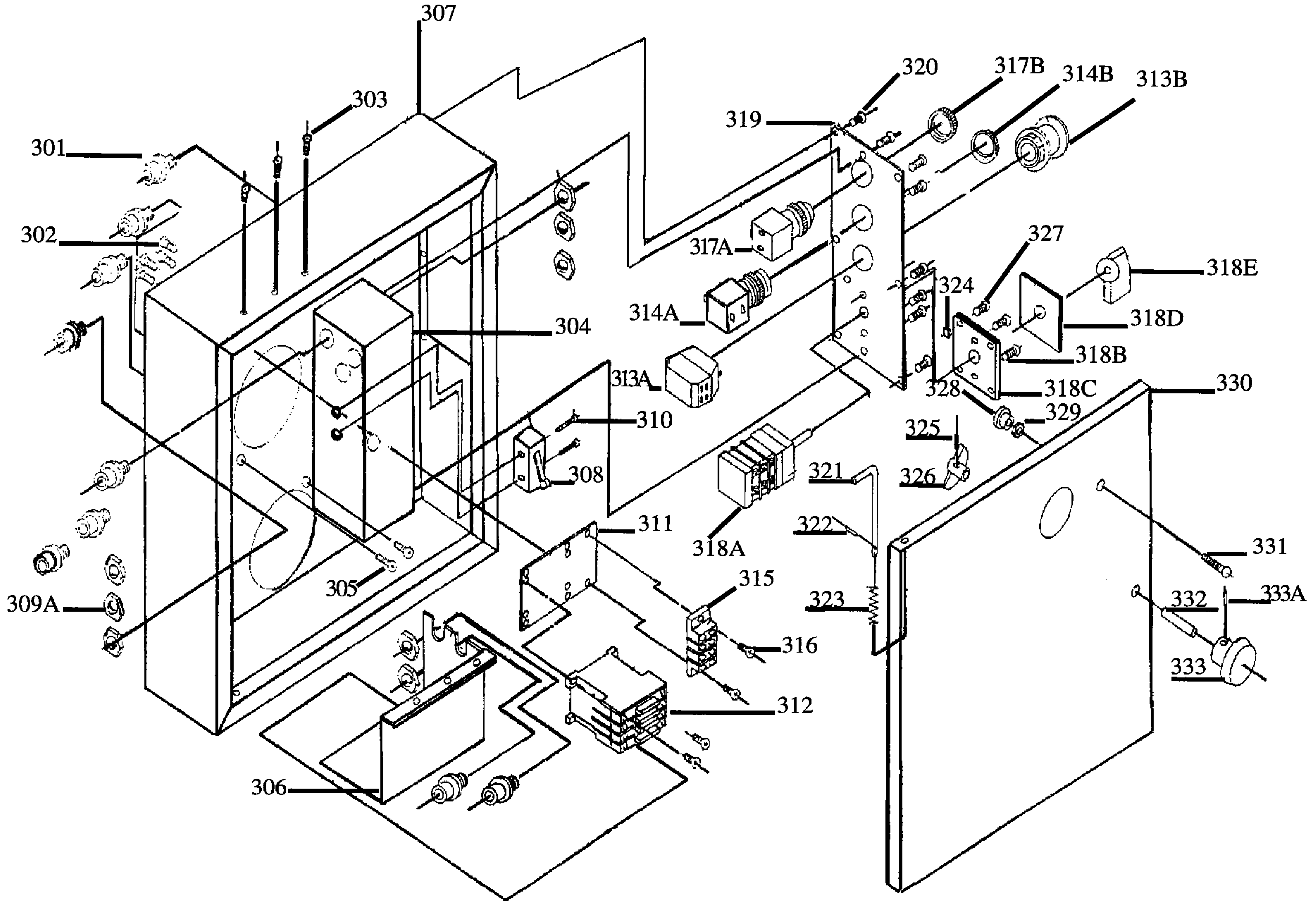


S12640	201	hex bolt M10x40
S18170	202	washer M10
S22104	203	taper pin M10 x 40
S11512	204	cone point set screw M5x12
M13205	205	headstock
M13206	206	triplex gear
M13207	207	brass bushing
M13208	208	O ring PD15x30x10
M13209	209	O ring seat
S11905	210	fillister slotted screw M5x12
S21290	211	key 5x25
M13212	212	spacer M14
M13213	213	shaft D
S21115	214	key 4x10
See Below	215	gear, A or D
M13215-24	215-24	A or D Gear, 24 Tooth
M13215-27	215-27	A or D Gear, 27 Tooth
M13215-30	215-30	A or D Gear, 30 Tooth
M13215-33	215-33	A or D Gear, 33 Tooth
M13215-36	215-36	A or D Gear, 36 Tooth
M13215-39	215-39	A or D Gear, 39 Tooth
M13215-42	215-42	A or D Gear, 42 Tooth
M13215-48	215-48	A or D Gear, 48 Tooth
M13215-60	215-60	A or D Gear, 60 Tooth
M13215-72	215-72	A or D Gear, 72 Tooth
M13215-120	215-120	A or D Gear, 120 Tooth
S18155	216	hex nut M10
S22038	217	roll pin M3 x 8
S22400	218	roll pin M4 x 20
S11125	219	flat point set screw M5x20
S18065	220	hex nut M5
S12512	221	pan head screw M5x12
M13222	222	gear C
M13223	223	gear C(2)
M13226	226	cover
M13227	227	taper roller bearing 40x8x18

M13226	226	cover
M13227	227	taper roller bearing 40x8x18
M13228	228	gasket (left)
M13229	229	duplex gear (B)
M13230	230	brass bushing
M13231	231	cover (left)
S11611	232	hex ssocket hd cap srewM5x6
M13233	233	spacer
M13234	234	V-belt Z710
M13235	235	pulley
M13236	236	spanner nut M40x1.5
M13237	237	locking washer M40
M13238	238	pulley spacer
M13239	239	external snap ring M63
S21102	240	key 10x25
S12215	241	hex socket cap screw M5x12
M13242	242	bracing plate
M13243	243	oil plug
M13244	244	O ring
M13245	245	spindle
S12310	246	hex head bolt M8x25
S21822	247	key 8x22
M13248	248	flange
M13249	249	gasket (right)
M13250	250	taper roller bearing 45 x 75 x19
M13251	251	locking ring
M13252	252	gear A
S12242	253	cone point set screw M8x10
S21220	254	key 5x14
M13255	255	hole plug
MI3256	256	hole plug
MI3257	257	shaft C
M13258	258	shaft B
M13259	259	brass bushing
M13260	260	oil glass
M13261	261	fork
M13262	262	spacer M12

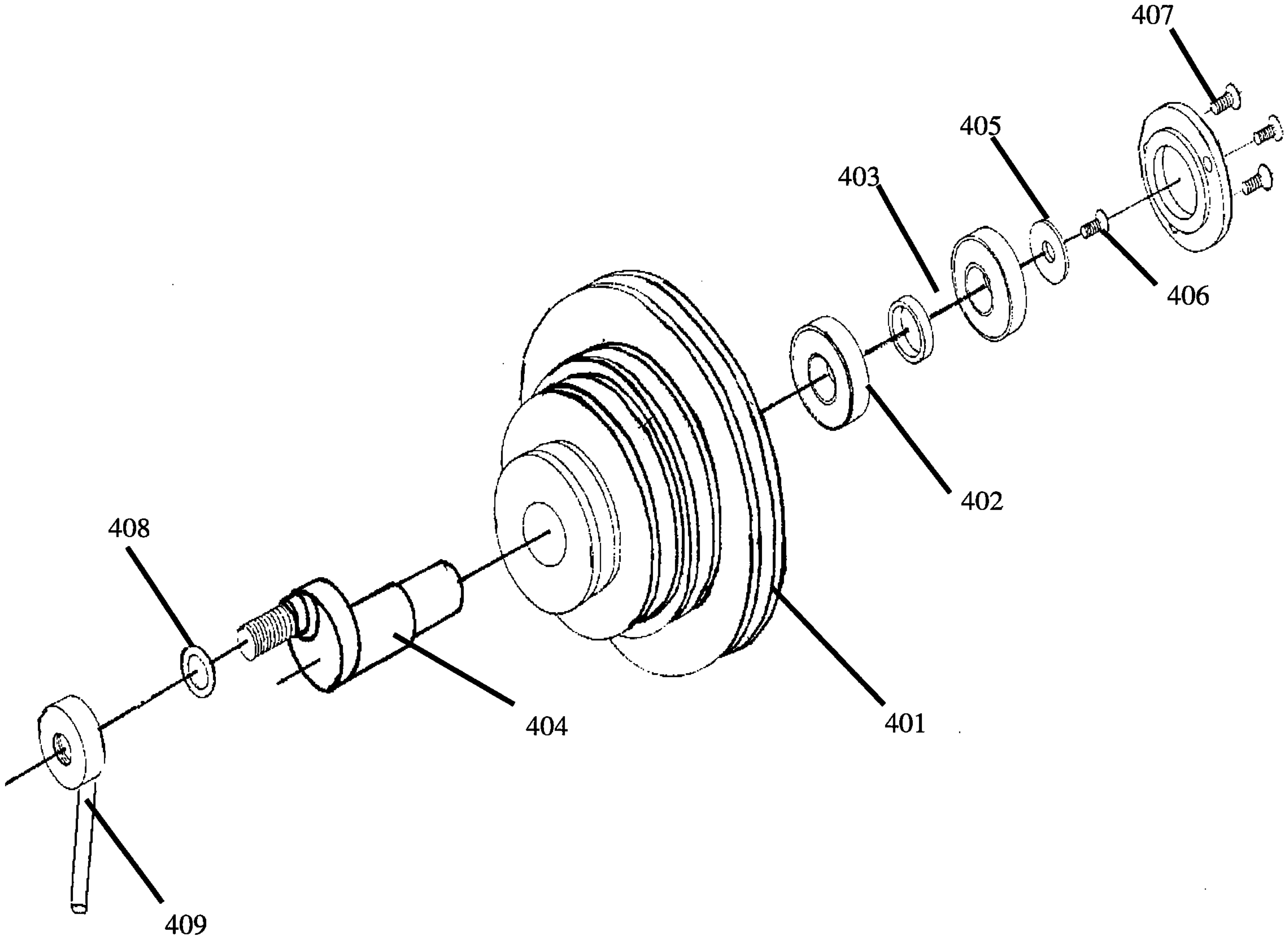
M13263	263	fork arm
S22124	264	straight pin ?B12 x 45
M13265	265	handle knob M10x32
M13266	266	spacer
M13267	267	handle
M13268	268	handle seat
C30126	269	steel ball 6.5
M13270	270	spring 5 x 0.8 x 15
S12208	271	flat point set screw M8x5
S22425	272	roll pin M4 x 25
S22518	273	roll pin M5 x 18

Diagram 3:0 Electrical Diagram



M14301	301	connection tube M10
S11416	302	round phillips screw M4x16
S11335	303	round phillips screw M4x8
M14304	304	electric box
S11941	305	hex socket head cap screw M6x10
M14306	306	electric housing
M14307	307	pulley box
Z40201	308	micro switch LX5-11N
M14309	309	connection tube
S11425	310	pan slotted screw M4x25
M14311	311	electrical panel
M14312	312	relay B16
Z40203	313A	killoff switch LAY3
Z40284	314A	start switch LA19
M14315	315	wire block
S11344	316	pan phillips screw M4x10
Z40379	317	light indicator AD11
M14318	318	on-off switch H25-10
M14319	319	electrical plate
S11344	320	pan phillips screw M4x10
M14321	321	pivot
S22212	322	straight pin ? A2 x 12
M14323	323	spring 9 x 0.8x 50
S18030	324	hex nut M4
S22314	325	roll pin ?3 x 14
M14326	326	latch piece
S12416	327	flat head phillips M4x16
M14328	328	micro switch lever
S18065	329	hex nut M5
M14330	330	door
S11535	331	round head screw M5x35
M14332	332	stud
M14333	333	knob A8x32
M14334	318D	on-off plate
M14335	318E	switch knob

Diagram 4.0 Lathe Idler Pulley



M15401	401	idle pulley
M15402	402	ball bearing 104 20 x 42x12
M15403	403	spacer
M15404	404	shaft
M15405	405	washer
S11105	406	flat slotted screw M5x10
M15407	407	cover
S18200	408	washer M 12
M15409	409	handle

Diagram 5:0 Millhead

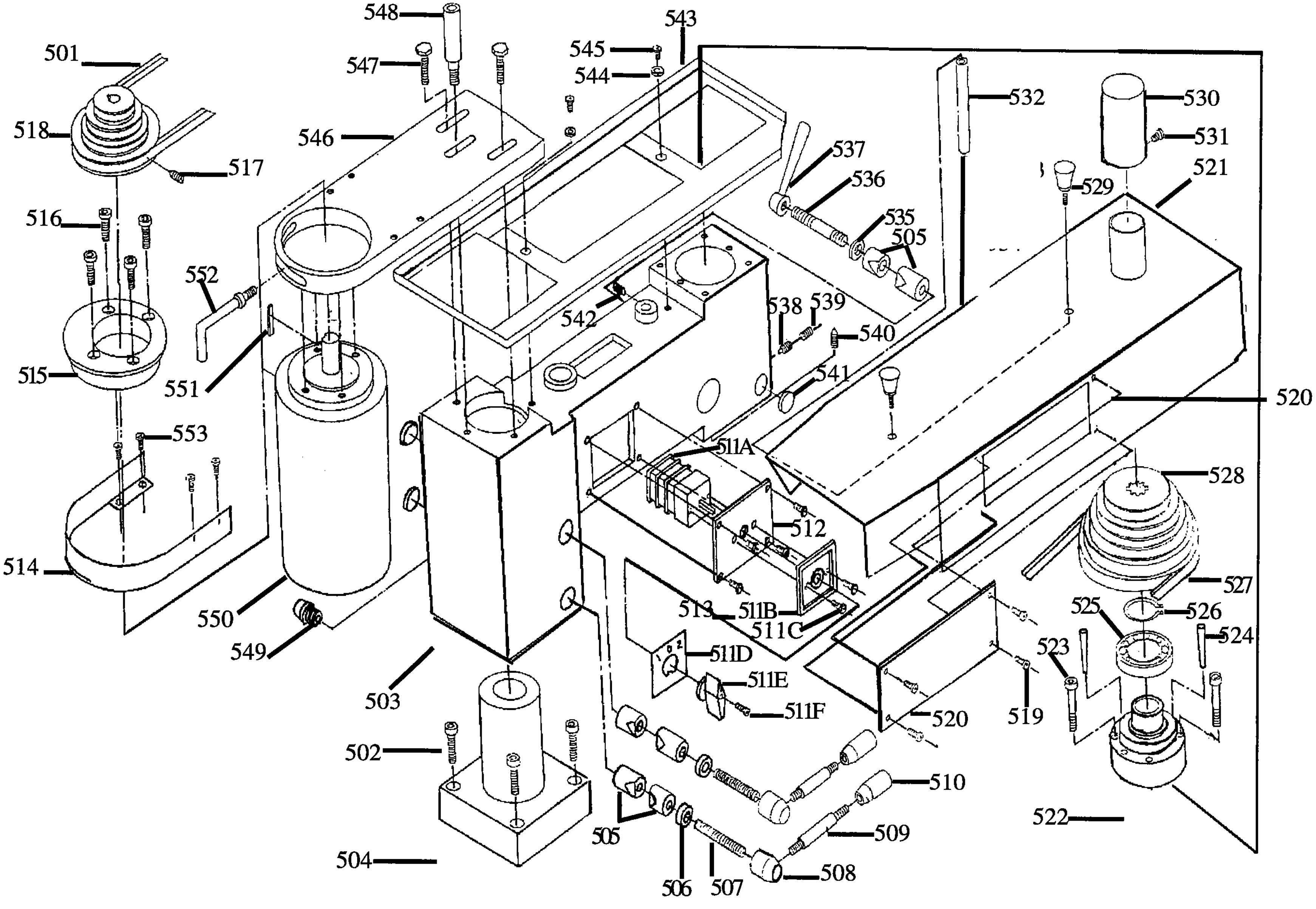


Diagram 5:1 Mill Quill and Spindle

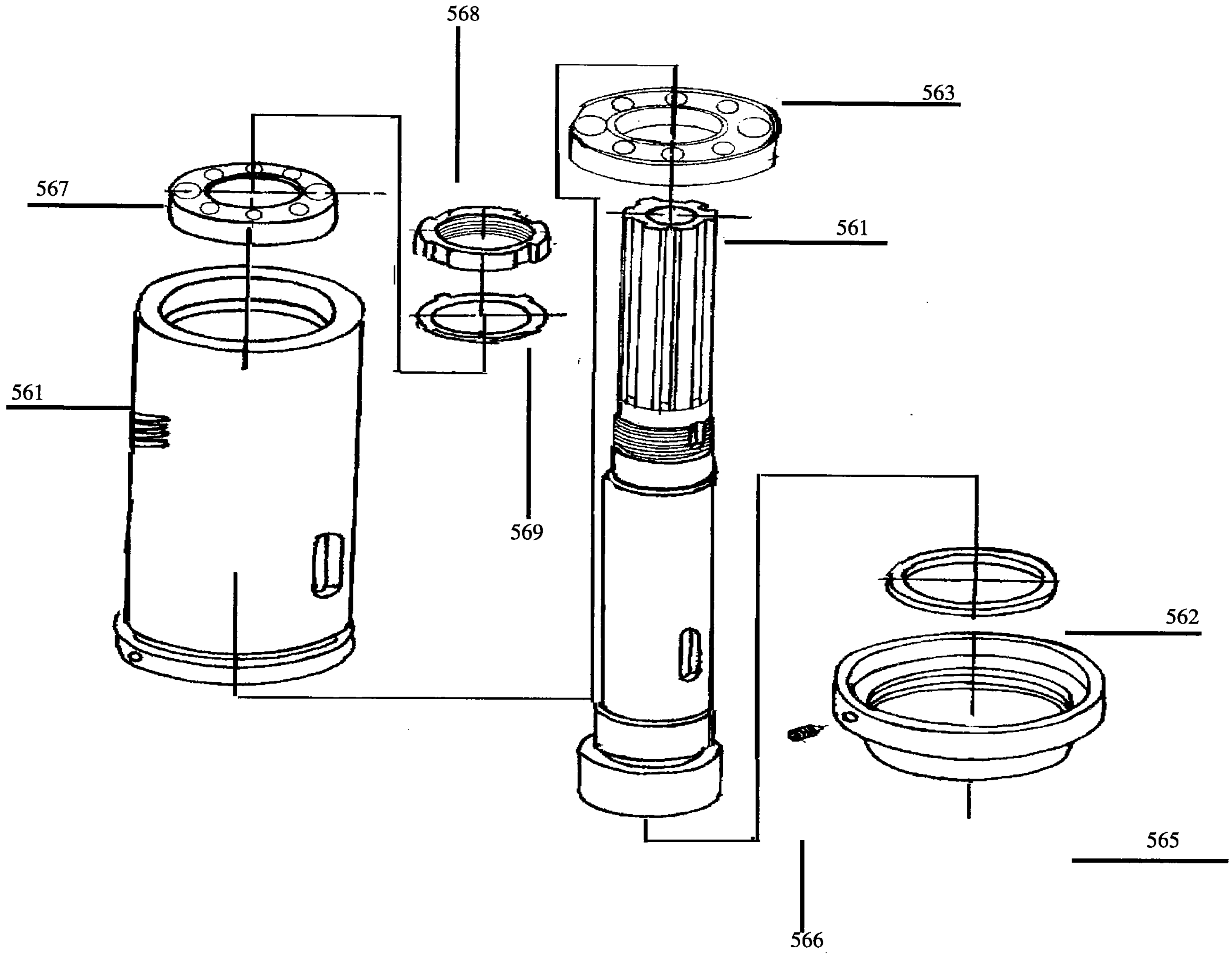
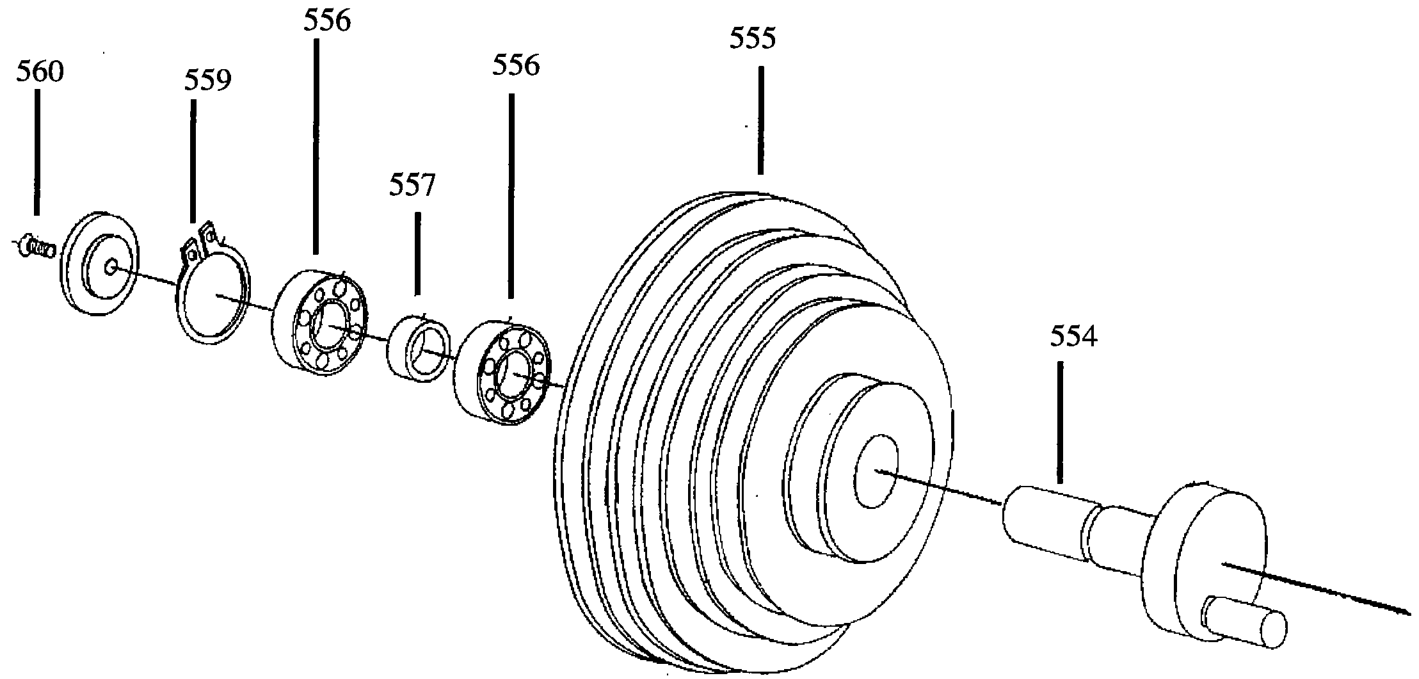


Diagram 5:2 Mill Idler Pulley

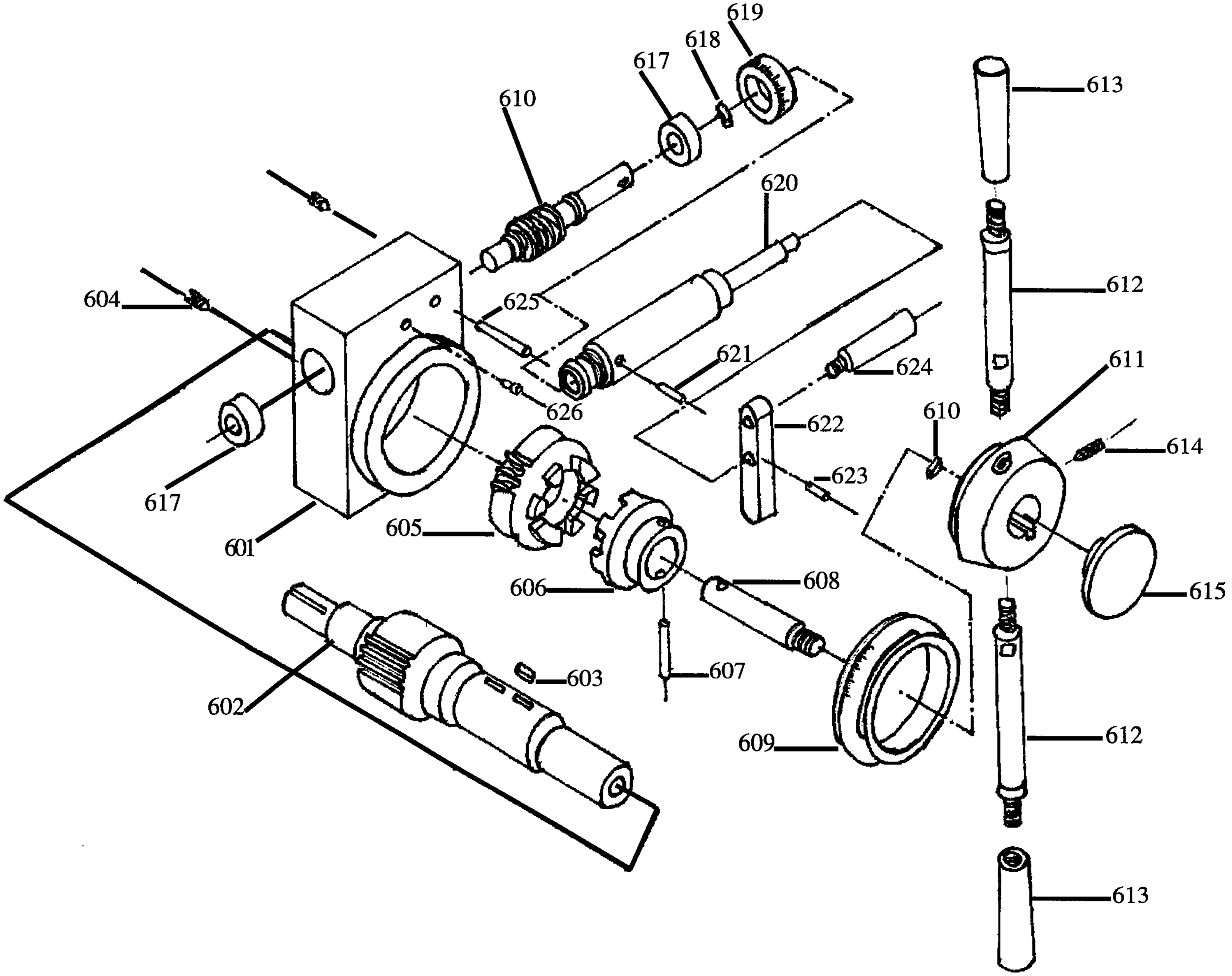


M16501	501	V-belt Z800
S12104	502	hex socket head cap screw M10
M16503	503	mill-drill head
M16504	504	mill-drill head support column
M16505	505	brake pad (pair)
M16506	506	brake spacer
M16507	507	stud AM10x80
M16508	508	handle seat
M16509	509	handle
M16510	510	handle knob BM10x80
M16511	511D	on-off switch plate
M16512	512	switch plate
S11058	513	flat slotted screw M5x8
M16514	514	frame cover
M16515	515	eccentric flange
S11340	516	hex socket head cap screw M6x20
S12252	517	cone point set screw M8x12
M16518	518	motor pulley
S11046	519	pan phillips screw M4x6
M16520	520	see-through window
M16521	521	mill-drill cover (upper)
M16522	522	pulley seat flange
S12041	523	hex socket head cap screw M6x40
S22545	524	taper pin M A5 x 45
M16525	525	ball bearing 108 40x 68 x15
S23040	526	external snap ring M 40
M16527	527	V-belt Z860
M16528	528	mill-drill pulley
M16529	529	knob CM8x25
M16530	530	cap
S11644	531	pan phillips screw M5x10
M16532	532	mill-drill cover post
M16535	535	brake spacer
M16536	536	stud AAAM10 x60
M16537	537	handle BM10 x 0

S12268	538	dog point set screw M8x14
S12812	539	flat point set screw M8x12
S12281	540	cone point set screw M8x20
M16541	541	hole plug
S12252	542	cone point set screw M8x12
M16543	543	mill-drill cover (lower)
S18110	544	washer M6
S11954	545	pan head screw M6x12
M16546	546	motor mount
S12620	547	hex head bolt M10x30
M16548	548	mill-drill cover post
M16549	549	connect tube (M16 2pcs, M18 1 c)
M12111	550	motor
S21230	551	key A5x16
M16522	552	tension handle
S11620	553	pan head screw M6x20
M16554	554	eccentric shaft
M16555	555	idle pulley
M16556	556	ball bearing 104 20x 42 x12
M16557	557	spacer
S23042	558	external snap ring M42
M16559	559	bearing cover
S11105	560	flat head srew M5x10
M16561	561	quill
M16562	562	gasket
M16563	563	ball bearing 35 x 62 x1 7
M16564	564	spline spindle
M16565	565	cover
S11612	566	cone point set screw M5x6
M16567	567	bearing 2007106 30 x 55 x 16
M16568	568	spanner nut M30x1.5
M16569	569	washer M30
M14318	511E	switch knob
S12212	511F	pan head screw M2 x 12
M14318	511A	switch, mill motor
M14336	511B	switch seat
S12315	511C	flat head screw M3 x 15

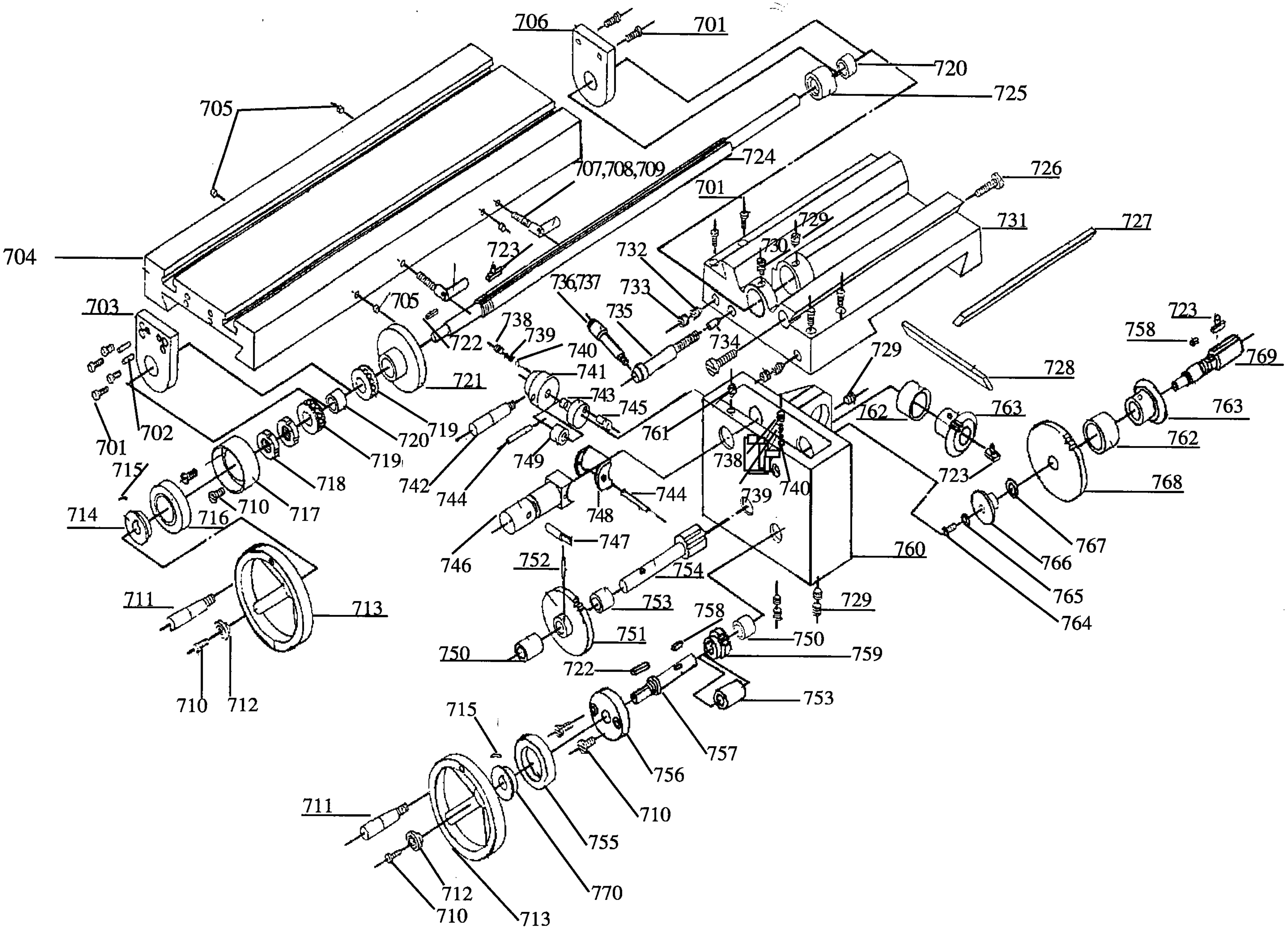
This page is left intentionally blank.

Diagram6:0 Mill Fine Feed/Drill Presss



M17601	601	quill feeding box
M17602	602	gear shaft
S21530	603	key 6x14
S18096	604	cone pointset screw M6x8
M17605	605	worm gear
M17606	606	clutch jaw
S22180	607	taper pin D M5 x 28
M17608	608	stud
M17609	609	dial
M17610	610	spring piece
M17611	611	dial seat
M17612	612	handle
M17613	613	handle knob BM8x40
S12816	614	cone point set screw M6x18
M17615	615	knurled knob BM10x40
M17616	616	worm
M17617	617	dial seat
M17618	618	spring piece
M17619	619	dial
M17620	620	worm shaft
S22400	621	roll pin M4 x 20
M17622	622	crank
S22441	623	roll pin M4 x 14
M17624	624	handle M8x40
S22260	625	roll pin M5 x 35
C30030	626	oiler M6

Diagram 7:0 Crossfeed and Apron



S11340	701	hex socket head cap screw M6x20
S22025	702	straight pin A M5 x25
M18703	703	trestle B
M18704	704	table
C30050	705	oiler M6
M18706	706	trestle A
M18707	707	locking screw
M18708	708	locking lever
S22102	709	roll pin M2 x 10
S12412	710	hex head bolt M4x12
M18711	711	handle M6x50
M18712	712	washer
M18713	713	handwheel B12 x100
M18714	714	dial seat A
M18715	715	spring piece
M18716	716	dial
M18717	717	fixing sleeve A
M18718	718	spanner nut M14x1.5
M18719	719	ball bearing 8102 ? 15x ? 28 x9
M18720	720	spacer A
M18721	721	gear
S21110	722	key 4x18
M18723	723	T-key
M18724	724	cross feed screw
M18725	725	cross feed nut
M18726	726	adjust screw
M18727	727	table gib
M18728	728	carriage gib
S18096	729	cone point set screw M6x8
S11068	730	dog point set screw M6x8
M18731	731	carriage
S11816	732	cone point set screw M8x16
S11812	733	flat point set screw M8x12
M18734	734	pin
M18735	735	locking stud
M18736	736	handle knob M6 x 20
M18737	737	handle M6x32

M18737	737	handle M6x32
S11088	739	spring
C30126	740	steel ball M 6.5
M18741	741	handle seat
M18742	742	handle BM6x50
M18743	743	eccentric wheel
S22430	744	roll pin M4x30
M18745	745	arm
M18746	746	half nut seat
M18747	747	stop pin
M18748	748	half nut
M18749	749	spacer E
M18750	750	spacer B
M18751	751	gear
M18752	752	roll pin M4 x 22
M18753	753	spacer C
M18754	754	shaft pinion
M18755	756	fixing sleeve B
M18757	757	shaft
S21058	758	key 5x8
M18759	759	gear
M18760	760	apron
S11816	761	cone point set screw M8x16
M18762	762	spacer D
M18763	763	bevel gear
S11680	764	pan phillips screw M6x8
S18110	765	washer M6
M18766	766	pull-push knob
S18200	767	washer M12
M18768	768	gear
M18769	769	shaft
M18770	770	dial seat B

Diagram 8:0 CNC Mount (Y-Axis)

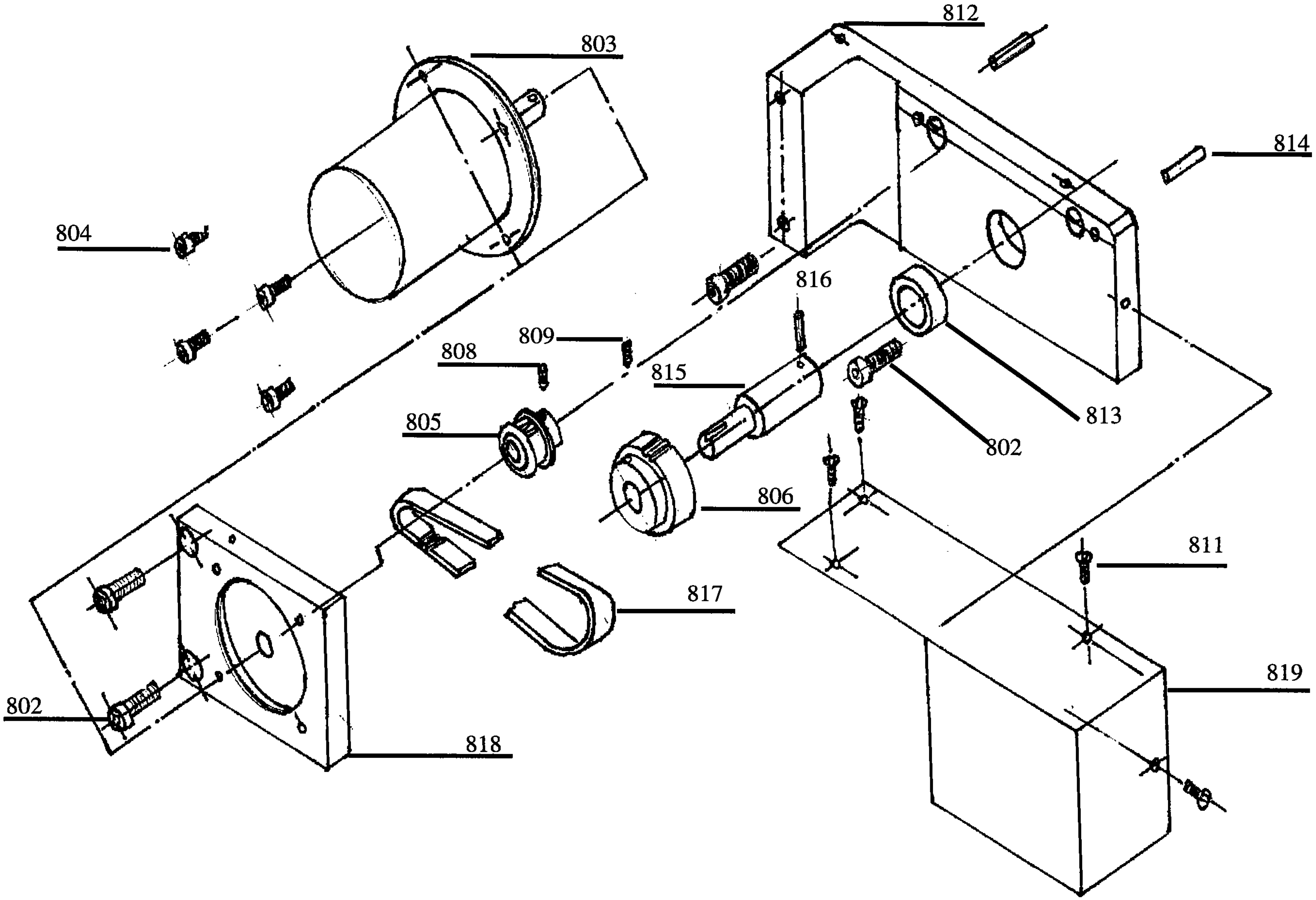
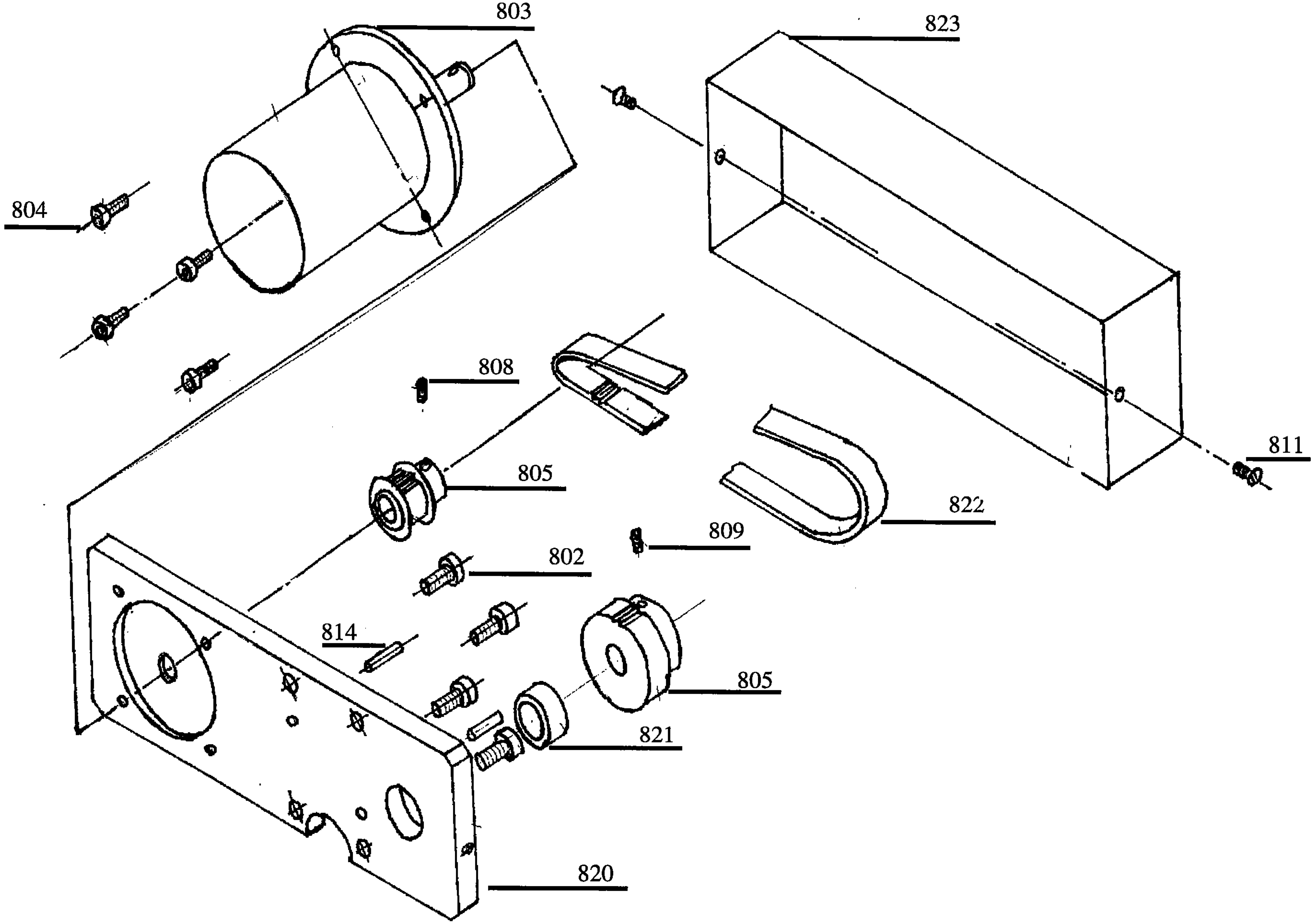
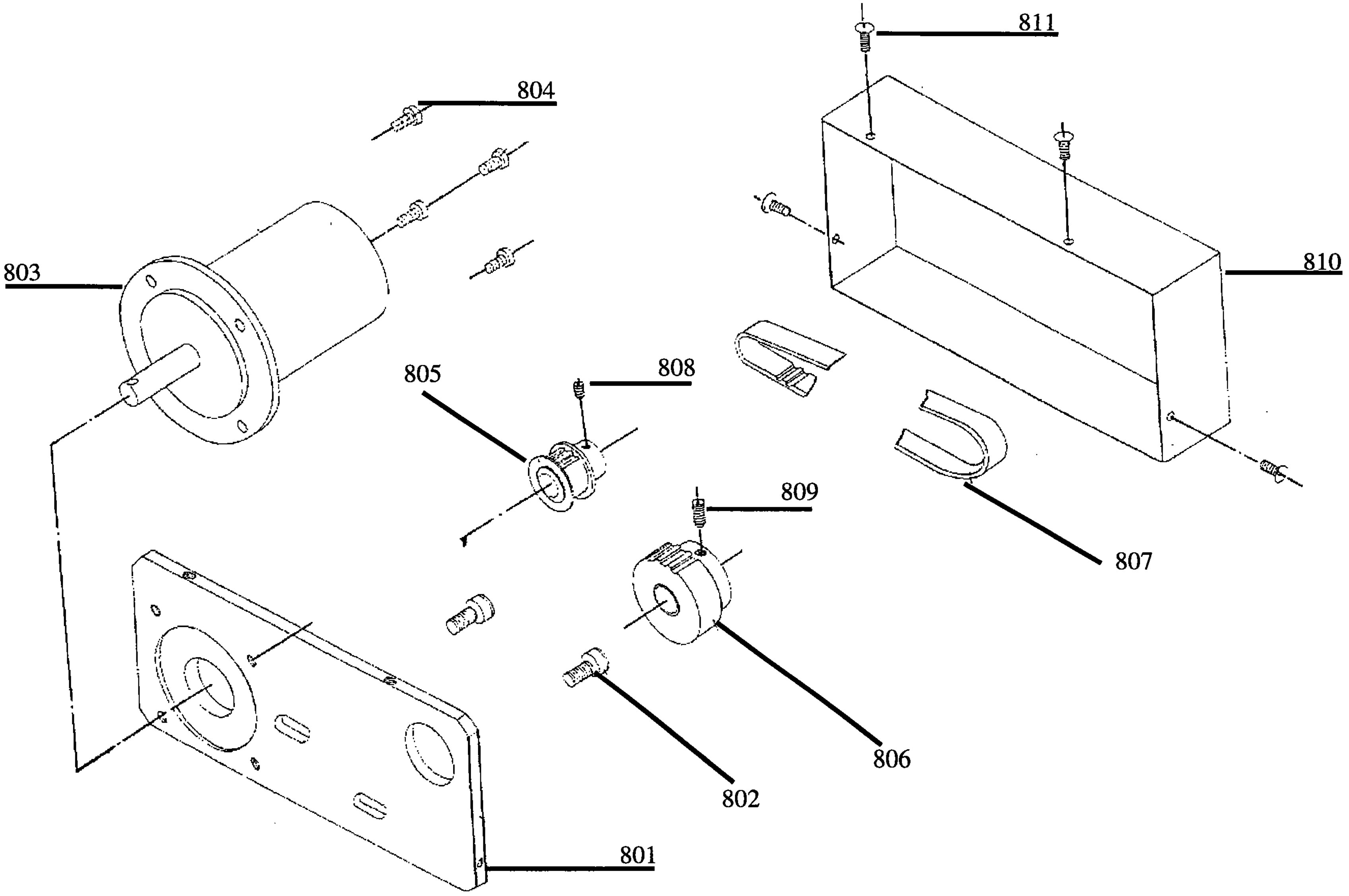


Diagram 8:1 CNC Mount (Z-Axis)

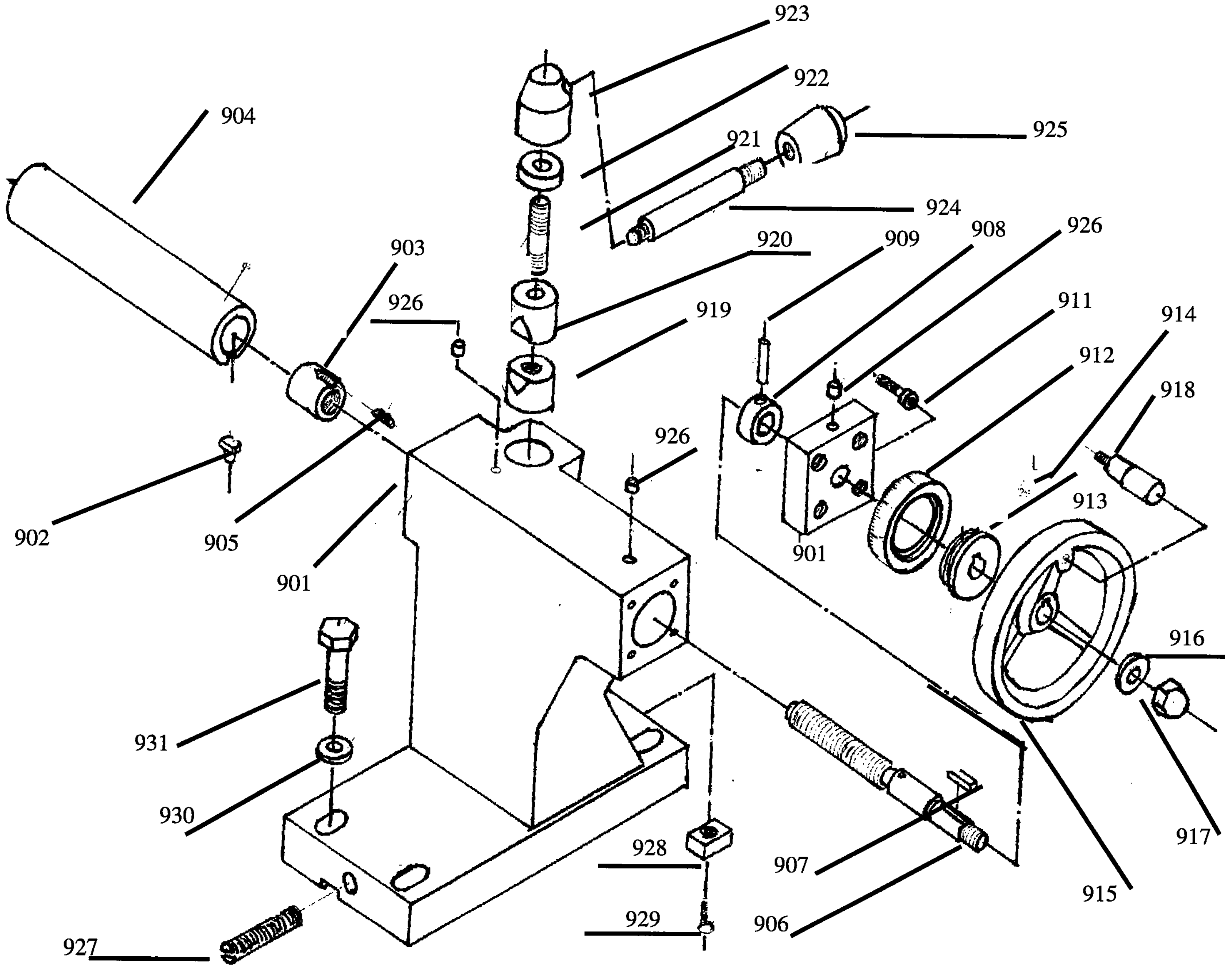


Diagarm 8:2 CNC Mount (X-axis)



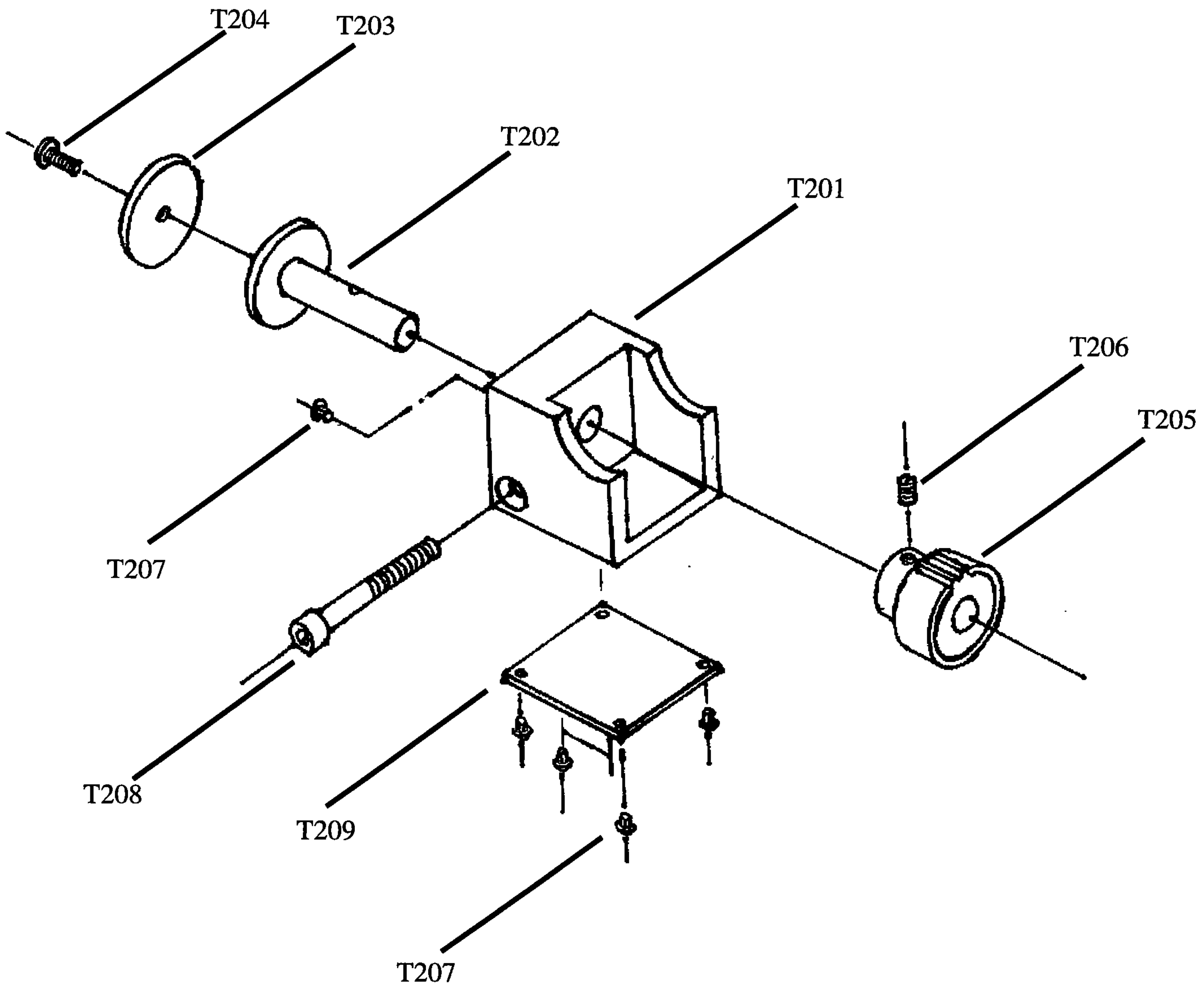
M19801	801	step motor bracket (X) (longitudinal)
S12291	802	hex socket head screw M8x20
M19803	803	step motor
S12515	804	hex socket head screw M5x15
G12001	805	small timing pulley
M19806	806	large timing pulley
M19807	807	timing belt
S11612	808	hex socket set screw M5x6
S11126	809	hex socket set screw M6x12
M19810	810	cover (X)
S11355	811	pan phillips screw M4x8
M19812	812	step motor bracket (Y) (transversal)
M19813	813	spacer
S22512	814	straight pin M5 x 12
M19815	815	shaft
S22412	816	straight pin M4 x 12
M19817	817	timing belt 100XL037
M19818	818	motor mount
M19819	819	cover (Y)
M19820	820	step motor bracket (Z) (vertical)
M19821	821	spacer
M19822	822	timing belt
M19823	823	cover (Z)

Diagram 9:0 Tailstock



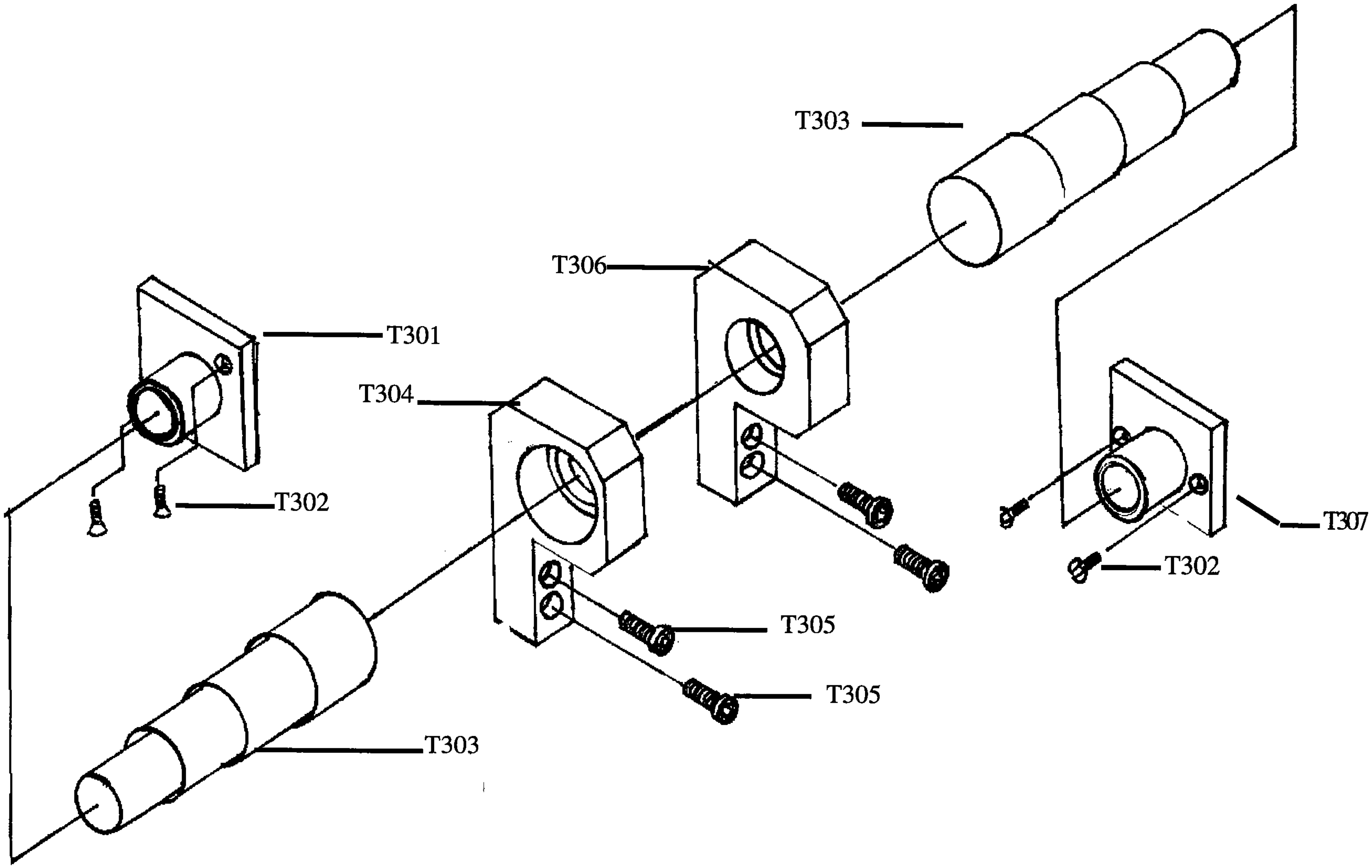
M11901	901	tailstock
M11902	902	T-key
M11903	903	tailstock nut
M11904	904	tailstock barrel
S11332	905	set screw M4x8
M11906	906	tailstock screw
S21140	907	key C4x18
M11908	908	spacer
S22502	909	straight pin D? 5 x 20
M11910	910	nut seat
S11691	911	hex socket head screw M5x20
M11912	912	dial
M11913	913	dial seat
M11914	914	spring piece
M11915	915	handwheel B12x100
S18170	916	washer M10
S18170	917	acorn nut M10
M11918	918	handle M6x50
M11919	919	locking pad (lower)
M11920	920	locking pad (upper)
M11921	921	stud AM10x40
M11922	922	locking spacer
M11923	923	handle seat
M11924	924	handle
MI1925	925	handle knob M10x32
C30050	926	oiler M6
S12105	927	set screw M10x50
M18928	928	key
S11653	929	set screw M5x12
S18170	930	washer M10

Diagram 10:0 Threading Dial



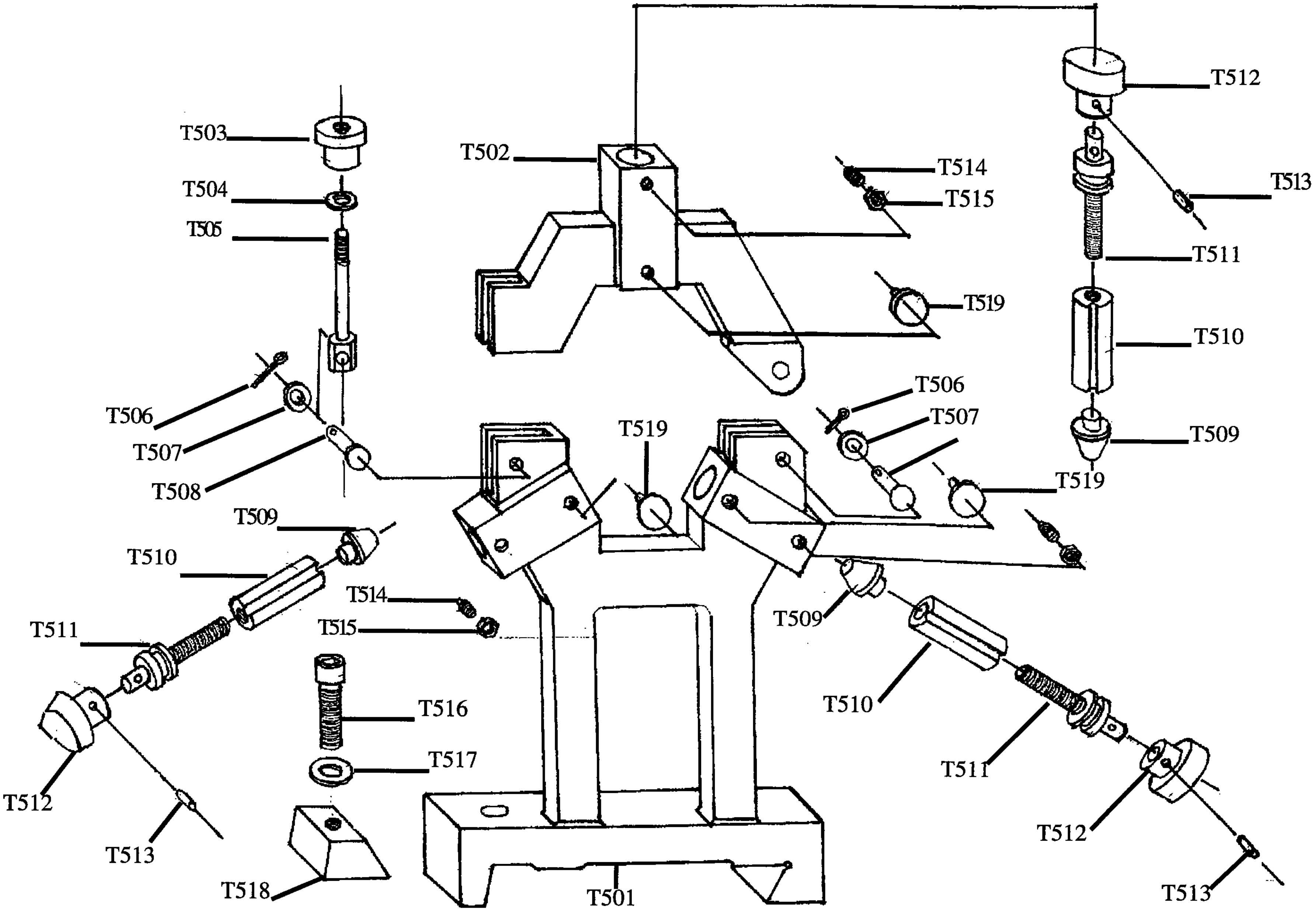
M20201	T201	threading dial seat
M20202	T202	shaft
M20203	T203	indicator plate
S11355	T204	pan phillips screw M4x12
M20205	T205	gear
S11632	T206	set screw M5x8
S22350	T207	revet M2 x 4
S11951	T208	hex socket head screw M6x12
M20209	T209	threading dial plate

Diagram 11:0 Leadscrew Cover



M21301	T301	cover mount (left)
S13353	T302	flat slotted screw M4x10
M21303	T303	leadscrew cover (left)
M21304	T304	cover seat (left)
S11951	T305	hex socket head screw M6x12
M21306	T306	cover seat (right)
M21307	T307	cover mount (right)

Diagram 12:0 Steady Rest



M22501	T501	steady rest center frame
M22502	T502	steady rest center frame head
M22503	T503	small knob
S18140	T504	washer M8
M22505	T505	locking bolt
S22162	T506	cotter pin M6x20
S18110	T507	washer M6
M22508	T508	locking pin
M22509	T509	brass head
M22510	T510	finger sleeve
M22511	T512	large knob
S22316	T513	roll pin M3 x 16
S11978	T514	slotted set screw dog point, M6x16
S18221	T516	hex socket head screw M10x35
S18170	T517	washer M10
M22518	T518	chock
M22519	T519	knurled screw

Diagram 13:0 Follow Rest

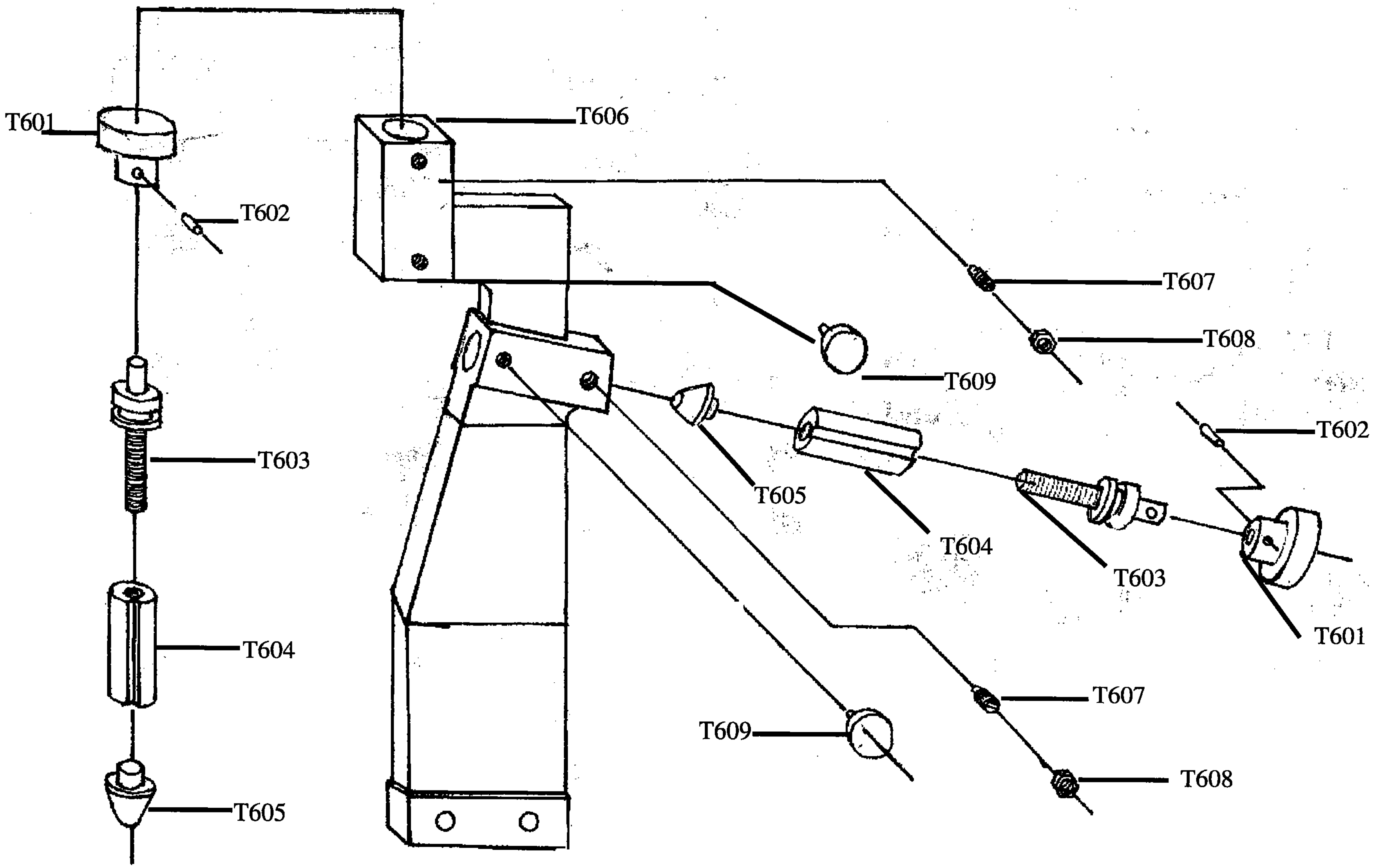
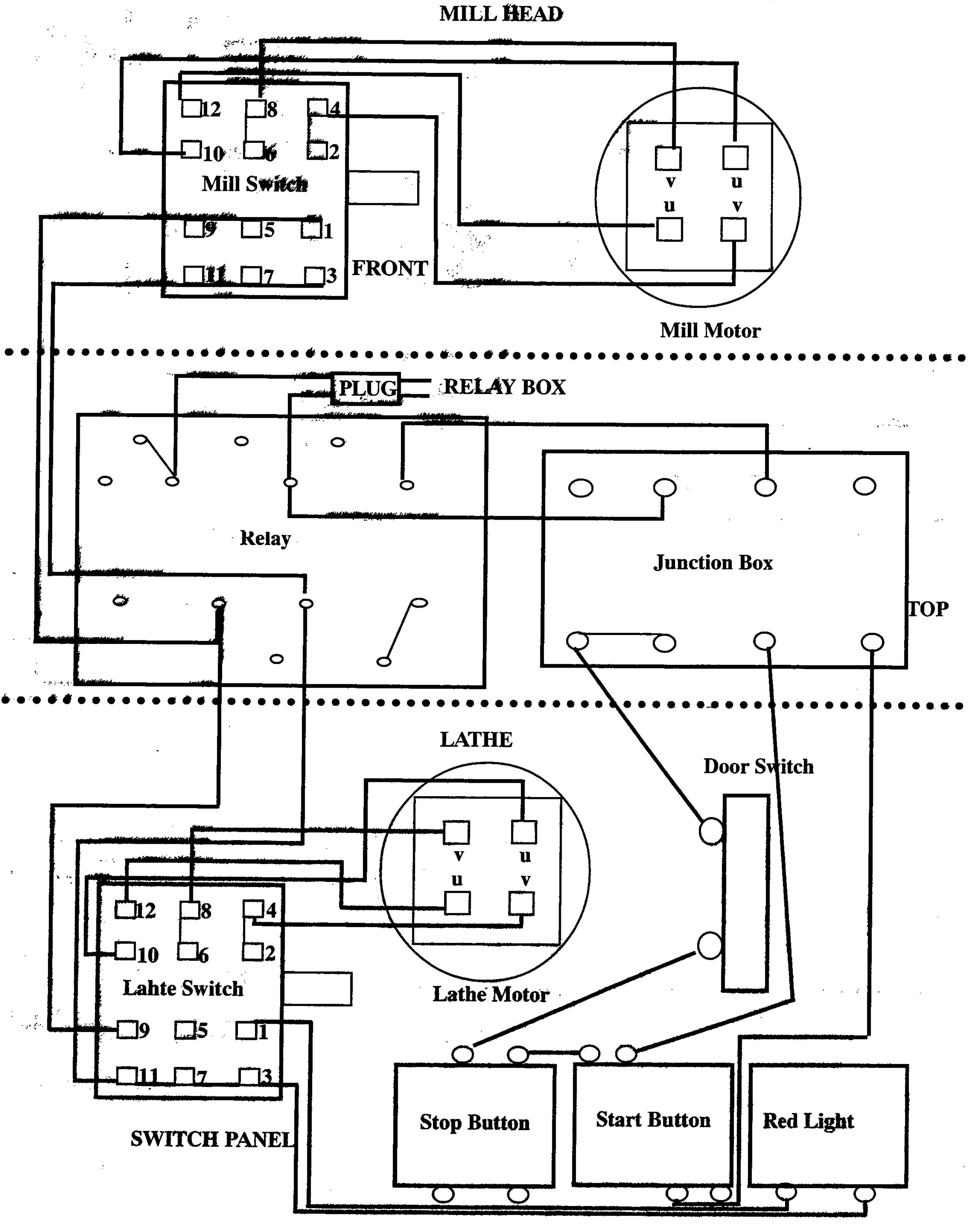
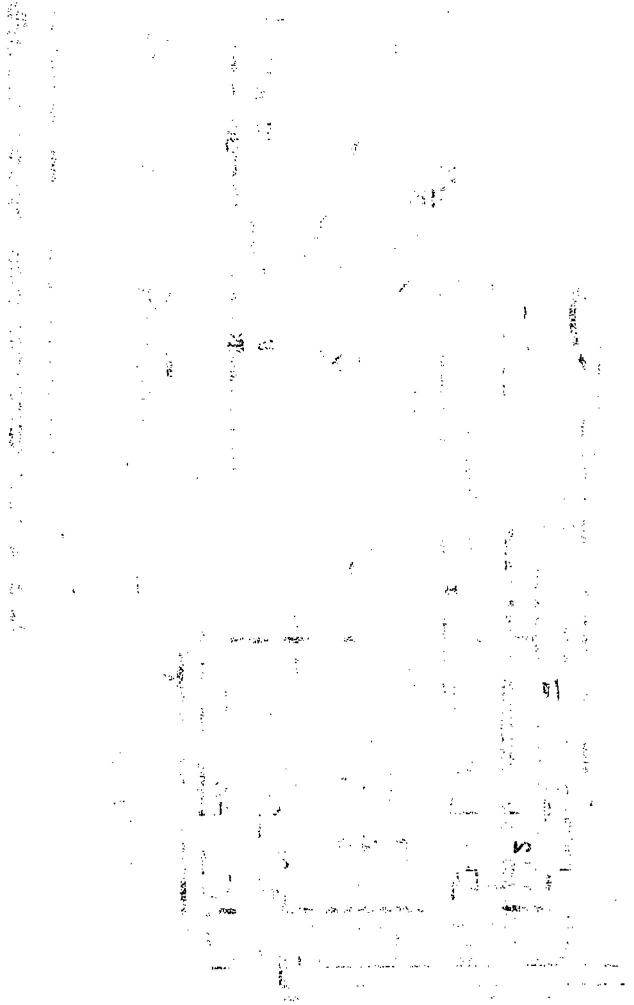


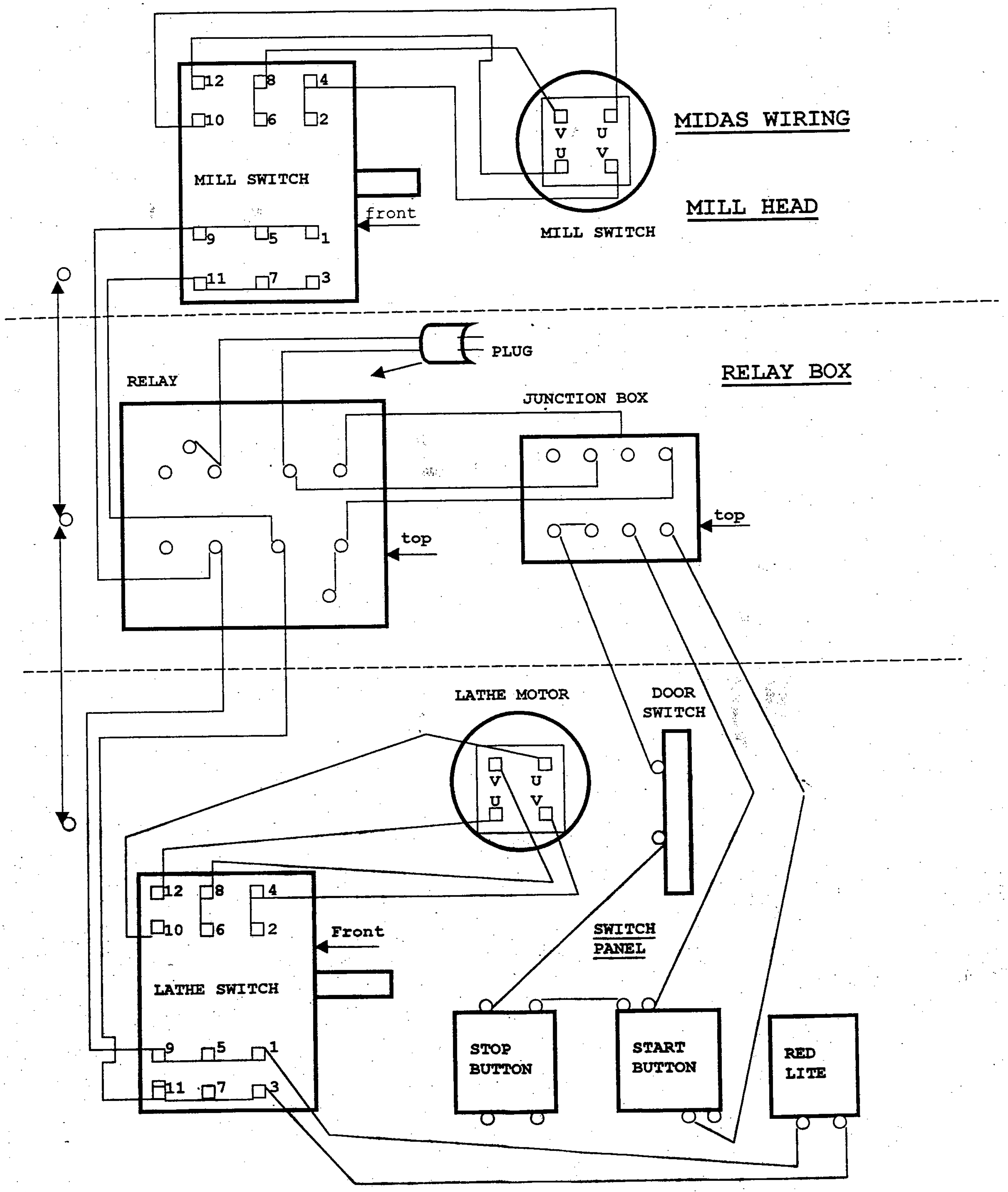
Diagram 14:0 Wiring Diagram

MIDAS 1720CNC WIRING DIAGRAM



M22512	T601	large knob
S22316	T602	roll pin M3 x16
M22511	T603	adjusting bolt
M22510	T604	finger sleeve
M22509	T605	brass head
M23606	T606	follow rest frame
S11978	T607	set screw dog pt set screw M6 x 16
S18381	T608	hex nut M6
M22519	T609	knurled screw




MIDAS WIRING
MILL HEAD
12
8
4
10
6
2
MILL SWITCH
front
9
5
1
11
7
3
V
U
U
V
MILL SWITCH
PLUG
RELAY
RELAY BOX
JUNCTION BOX
top
top
LATHE MOTOR
DOOR SWITCH
V
U
U
V
12
8
4
10
6
2
Front
LATHE SWITCH
9
5
1
11
7
3
SWITCH PANEL
STOP BUTTON
START BUTTON
RED LITE