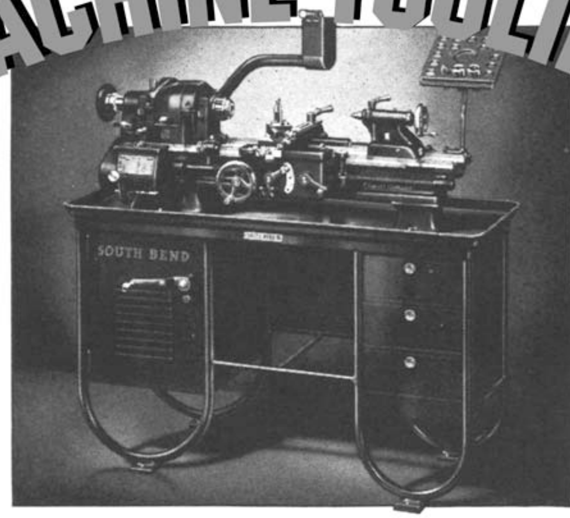
MACHINE TOOLING

WARNING!

of nearly all accidents arising from contact with machine tools it may be said it is more the want of care than the want of knowledge...



Our precious 10" swing South Bend Lathe; page 102

Machine etiquette and maintenance:

"Tools and the Craft"

For hundreds, if not thousands, of years the craftsman has had an inseparable link to their tools. We readily forget, in this disposable age of Home Depot, Menards, and Ace Hardware, that the first craftsmen had to make their own tools and learned how through a long apprenticeship. It is easily understandable that their connection to their tools was stronger than ours, they had experienced and suffered through their creation. (Even a hundred years ago it was fully expected that a machinist should be able to hand scrape a piece of iron plate to trueness with a tolerance of one ten thousandth of an inch!)

As a result they "knew" their tools; what they could do, and how to best use them. And these tools were the means to their existence, not in the simple sense that they could make things with them but in the sense that if they did not maintain and respect them they would have to make new ones or they wouldn't eat.

We need to learn our tools like that, to understand their worth and their history, and to develop a respect for them.

Good work is hard without good tools.

Understanding the "feel" of the tool enables us to use it as an extension of our hands. Like riding a bike or playing an instrument as we become proficient we no longer think about the operation of the tool but it becomes an automatic extension of our hands. While this ability takes time to develop, like becoming automatic at anything, it does not happen without understanding the tool, its limits, when it is straining, and when it is no longer efficient. This can't happen if you use the wrong tool for the job, push the tool too far, or don't understand its operation.

This all relates to "Tool Respect". Respect for the tool and what it can do, and what it can't. Using the right tool for the job results in a long life for the tool.

"Good Craftsman Respect Their Tools"

Machine tools need even more care than our typical hand and power tools.

First, in regards to safety, you cannot become too "automatic". Be aware, awake, and attentive while you are using these machines, do not let people interrupt you, do not interrupt others, and keep focused on what you are doing.

Some of the process is needfully tedious but do not let it lull you into inattention. Machine tools are designed and geared to make heavy cuts in hard materials, they will not stop or stall for your fingers, hands,or arms.

Second is the actual care of the machine.
If you use it you **must** clean it. Sanding and polishing grit and compounds, wood dust, airborne dust, and cuttings can rapidly do great damage to the machine ways, bearings, screws and gears. After each work period you are **required** to clean the machine of all loose materials, wipe the machine and ways down with a rag, and oil coat the ways and any other exposed surfaces to prevent surface rust.

In addition you **<u>must</u>** oil the machine before and after every use. This means the addition of machine oil to ALL of the oil cups and reservoirs in addition to any bearing locations.

If there is no oil obviously available request oil form the shop techs and make sure it gets done. The oiling chart for a lathe (right) indicates the general oil requirements for a machine tool. Please use common sense and oil the lathe and mill where ever you think it needs it. You cannot "over" oil the machines.

If parts or cranks move or turn "hard" check for debris and oil it. If this does not remedy the problem please ask myself or the shop techs for assistance.

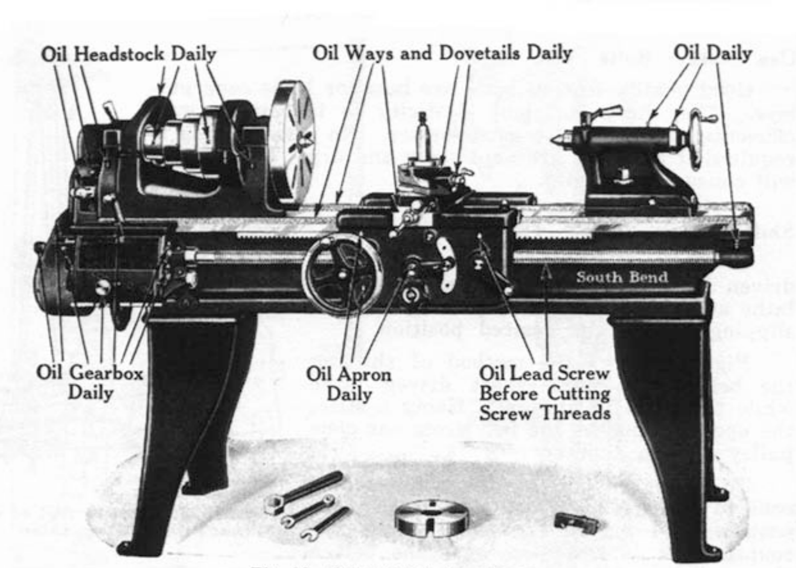


Fig. 32. Oiling Chart for a Lathe

Never force a feed wheel!

In any event **never continue to run the machine if it is making an odd noise, straining, slipping, or otherwise acting different.**

Ask for assistance.

We have available some limited tooling for both the milling machine and the metal lathe but, in general, everyone using these machines supplies their own. If you are interested in learning how to use either of these machines, or the appropriateness of them for your project, you can contact the MCAD shop staff or Professor Brad Jirka <bjirka@mcad.edu>.

There are many books available on machining and other metal processes. As a starting point my favorites, and **Highly Recommended**, are:

<u>How to Run a Lathe: the care and operation of a screw cutting lathe</u> — South Bend Lathe Works
Lindsay Publications Inc., Bradley, Il. — MCAD Bookstore
ISBN#: 1-55918-115-X

<u>The Complete Metalsmith</u>; 1982 — Tim McCreight
Davis Publications, Inc.; Worcester, Mass. — (Library)

<u>Pocket Reference</u> — Thomas J. Glover
Sequoia Publishing, Inc; Morrison, Colorado
ISBN#: 0-9622359-0-3

Resources: Tools and materials;

Discount Steel, Ambles, Midwest Steel, Vincent Brass and Aluminum.
Online at Ebay or Speedy Metals (fast shipment and good prices on small quantities)
Enco Tools (use-enco.com); MSC (mscdirect.com); J&L Industrial supply (jlindustrial.com)

METAL CUTTING ACTION

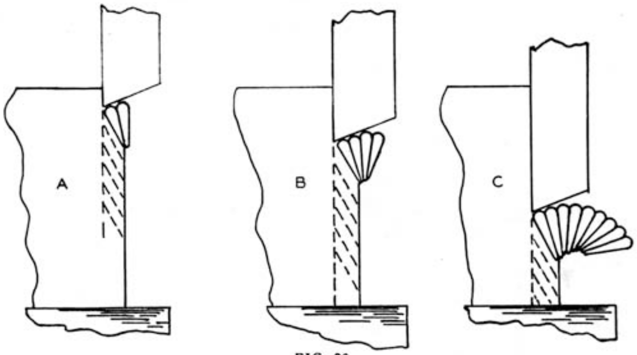


FIG. 36
Progressive steps in metal cutting. At "A" the tool is just entering the metal; at "B" the cut has progressed to a point where the triangular shape of the small sections can be seen. "C" shows the start of the curled chip. For clearness, a straight shear cut is illustrated and the size of the small sections greatly exaggerated.

The Theory of Machine cutting:

Machine cutting is similar to the cross cutting of wood with a chisel and is essentially, like most cutting tools, a variation on the chisel. In this case a metal "cold" chisel.

The finer the finish on the tool the better the finish you will get on your work. However, for heavy cutting, this is less of a concern as the tool builds up a "false edge" that does the actual cutting, or wedging off, of the material.

The materials you are cutting AND the tools will often get very hot while working even to the point of causing the stock to expand and lock into the chuck, collet, vise or mounts much like a router bit can "freeze" into a router collet. If the material you are working on is getting very hot you should consider letting it cool, using lubricants (a good idea in any case with most materials), and even loosening the chucks or mounts then retightening.

Cutters also have to have the correct shape for the job being done with proper clearances and "rakes" to "clear" the work (the same purpose of "set" in a saw blade). This information and the correct tool shapes are available in your text <u>How to Run a Lathe</u> and other machining manuals. (See chapter 4 for the basic information).

FALSE CUTTING EDGE ON THE TOOL

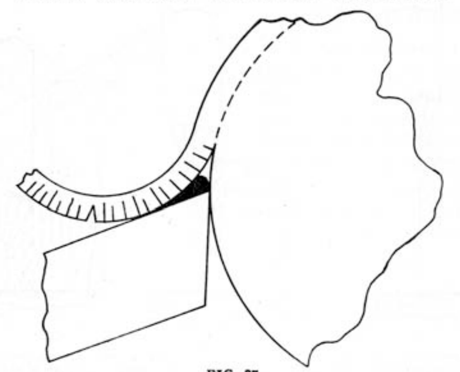


FIG. 37
The false cutting edge formed at the tip of a tool bit. The dark portion is deposited on the tool when taking heavy cuts. The wedging or cutting is done with this bit of metal, not the edge of the tool bit.

SET UP:

Proper "set up" of the machine includes mounting of the work, sharpening, mounting, and positioning of the tool (cutters), and setting of proper speeds, feeds, and cut based upon the material you are machining. This set up can often take longer than the actual making of the piece but can not be taken lightly.

While we normally mount lathe work in a chuck and mill work in a vise there are other ways to mount materials that may be more appropriate depending upon size, weight, and cutting considerations. These are introduced in the text and will be demonstrated in class but here are a few basic rules:

Lathe
- If the material extends more than twice its diameter from the chuck use a center in the tailstock to support it.
- If the material is very thin you will need to use a follower or steady rest to avoid deflection.
- If the material is held in the four jaw chuck off center you may have to add counter balance weights.
- You should set up to cut towards the headstock on most occasions.

Mill
- If the material is very tall you will have to mount it flat on the table.
- Material too large for the vise has to be firmly mounted with the hold downs.
- You must cut "against" the cutter (counter clockwise path just like a router).

Both
- Cuts should be very light until the surface is "true" (i.e. 0.005").
- You should not attempt cuts over 0.0625" (1/16") on the mill unless it "feels" good.
- You should not attempt cuts over 0.015" (15/1000") on the lathe.
- Harmonic vibration is possible on both machines and will ruin your finish. If you experience vibration change the spindle speed, feed rate, and depth of cut or all of them.

Lathe tool positioning:

Lathe tools must be positioned to "swing away" from the cut, not "into" the cut. Assuming in the images at right the tool is moving from the right to the left (towards the headstock) figure 1 shows a "jamming" cut while figure 2 shows the correct setup. Understand and practice this. Jamming of the cutter can result in damage to the machine, injury to you, and, for sure, damage to your piece.

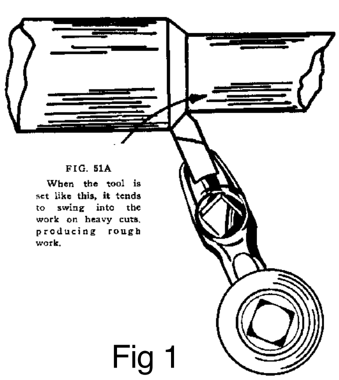


FIG. 51A
When the tool is set like this, it tends to swing into the work on heavy cuts, producing rough work.

Fig 1

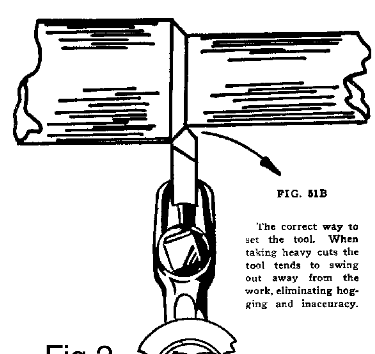


FIG. 51B
The correct way to set the tool. When taking heavy cuts the tool tends to swing out away from the work, eliminating hogging and inaccuracy.

Fig 2

Feed and carriage controls:

While a surprising amount of lathe work is done “by hand” finish cuts on straight surfaces, thread cutting, taper cutting, and facing can all be done with power feeds. These “feeds” simply run the tool down the length of the work, or across the work, and are known as the longitudinal and cross feeds.

To cut either manually or “automatically” you need to know the basic carriage controls and what they do. While explained in your book here is a quick review:

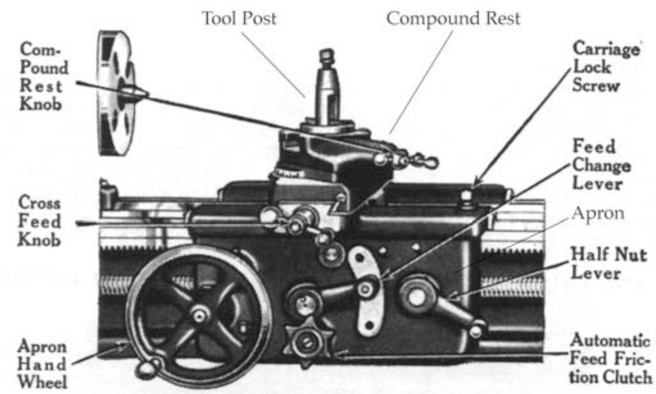


Fig. 35. Operating Parts of Lathe Carriage and Apron

The “apron” hand wheel moves the carriage left and right. The “cross feed” knob drives the tool in and out. The “compound rest knob” adjusts the cut by feeding the compound rest and tool post toward the work. (Please note that with minimal exception the compound feed is NEVER used to actually make a cutting pass).

The “half nut lever” engages the half nuts and is used only for feeds when threading. The carriage feed, cross cut feed and threading feed cannot be used at the same time (on this machine they lock each other out).

The “Automatic Feed Friction Clutch” engages the feed when turned clockwise and disengages when turned counter clockwise (as Dean says “lefty loosey”).

When you are using power feeds watch the machine at all times, be prepared to disengage the clutch (or half nuts), and leave yourself enough time to do so.

Close cuts to a shoulder, the chuck, or the tailstock should only be finished by hand. Never run the tool all the way to the finish with the automatic feeds. Running the tool or carriage into the chuck, headstock, tailstock, or material will result in a broken shear pin and an inoperable machine, at best, and can cause considerable damage to tools, holders, gears, and the lead screw. **Practice before you drive the carriage and always check for clearances at the chuck and tailstock.**

feed rates:

Feed rates are the relationship of spindle revolution to tool motion and are controlled through the quick change gear box. Feeds are indicated as “threads per inch” or decimals per revolution. The chart on the gear box gives you speeds based upon the “Tumbler lever” position; the left, center or right “Top lever” position; and the “sliding gear” position (in or out). The “back gear” does not affect feed rates only spindle speed. (A chart for spindle speeds of this machine is on page 23 of How to Run a Lathe.)

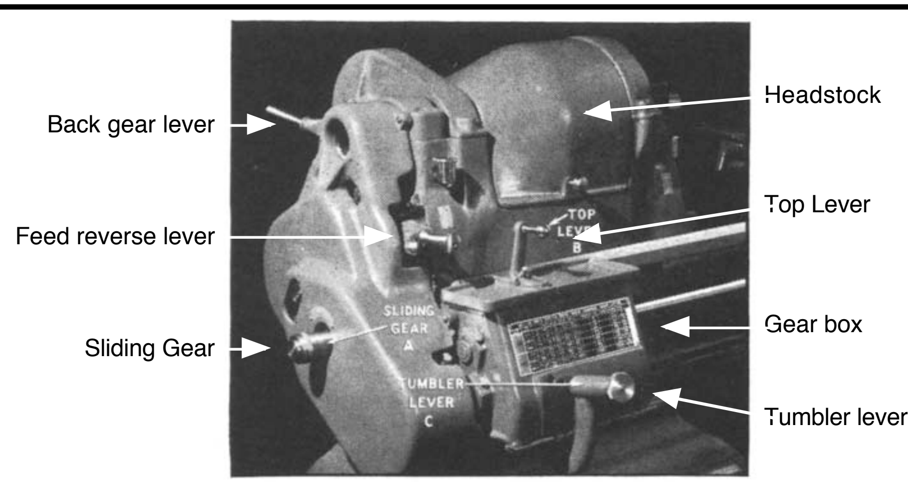


Fig. 211. Quick Change Gear Mechanism for Cutting Screw Threads

Feed, Speed, and cut:

All machining is based on the feed rate, the speed of the spindle or material (Surface Speed in feet per minute) and depth of cut. All are variable and all are dependent upon each other. The primary determinant, however, is the material being cut.

If spindle speeds (surface speeds) are lower the feed simply has to be slower (on automatic feed the lathe adjusts itself once set properly) or the cut lighter. Too fast a feed will result in “threading”. Cut (depth) is adjusted for the hardness of the material.

As a general rule the harder the material the slower the speed, the slower the feed, and the lighter the cut. Hence plastics, wood, brass and aluminum can be cut at higher speeds and feed rates than steel or bronze.

The chart at right (and the supplemental chart for materials on page 50 of How to Run a Lathe) gives much of the information needed to sharpen bits for specific materials and set up the feed and speed rates.

Surface speed in feet per minute (the 4th column from the left) is the starting point.

If we take Bessemer Screw stock we see we want a surface speed of 120 feet per minute. If our stock is 3” in diameter we we would multiply that by π (pi - 3.1416) to find the circumference:

3” x 3.1416 = 9.42”

Once we know the circumference we divide the desired surface speed by it to arrive at a RPM (revolutions per minute):

120 feet / 9.42” = 120 feet / .79 feet = 151.90 rpm

If we look at the chart on page 23, “Standard Spindle Speeds of South Bend Lathes in Revolutions per Minute”, in the row for the 10 inch lathe we see the closest options of Direct Drive belt position 3 (277 rpm) or Back Gear Drive Speed belt position 1 (127 rpm). Given that this is steel I would use the back gear option.

FIGURE 59

TOOL ANGLES AND SPEEDS FOR MACHINING STEEL

These Angles Refer to Tool Shapes on Pages 36, 37.

Description of Steel	S.A.E. No.	Machin-ability	Speed feet per minute	Side Clear-ance Angle	Front Clear-ance Angle	Back Rake Angle	Side Rake Angle
Bessemer Screw Stock	1112	100%	120	12°	8°	16½°	22°
Special Screw Stock	X1112	120	150	12°	8°	16½°	22°
High Manganese Screw Stock	X1314	95	100	12°	8°	16½°	22°
High Manganese Screw Stock	X1315	95	100	12°	8°	16½°	22°
High Manganese Screw Stock	X1335	75	100	12°	8°	16½°	18°
Open Hearth Screw Stock	1120	80	100	12°	8°	16½°	18°
Carbon Steel	1020	60	80	12°	8°	16½°	14°
Carbon Steel	X1020	70	80	12°	8°	16½°	14°
Carbon Steel	1035	62	80	12°	8°	16½°	14°
Carbon Steel	1040	61	80	12°	8°	16½°	14°
Nickel Molybdenum	4615	60	80	12°	8°	16½°	14°
Carbon Steel	1045	55	70	10°	8°	12°	14°
3½% Nickel Alloy..	2315	50	80	10°	8°	12°	14°
3½% Nickel Alloy..	2320	50	80	10°	8°	12°	14°
3½% Nickel Alloy..	2330	50	80	10°	8°	12°	14°
3½% Nickel Alloy Annealed	2335	50	70	10°	8°	12°	14°
Nickel Chromium Alloy	3115	50	70	10°	8°	12°	14°
Nickel Chromium Alloy	3120	50	70	10°	8°	12°	14°
Chrome Molybdenum	4140	50	70	10°	8°	12°	14°
Manganese Alloy ...	T1335	50	60	10°	8°	12°	14°
3½% Nickel, Annealed	2340	45	70	10°	8°	10°	12°
3½% Nickel, Annealed	2345	45	60	10°	8°	10°	12°
3½% Nickel, Annealed	2350	40	50	10°	8°	10°	12°
Nickel Chromium ..	3130	45	70	10°	8°	10°	12°
Nickel Chromium, Annealed	3135	45	60	10°	8°	10°	12°
Nickel Chromium, Annealed	3140	45	60	10°	8°	10°	12°
Chrome Vanadium, Annealed	6140	40	60	10°	8°	10°	12°
High Carbon Steel..	1095	35	50	10°	8°	8°	12°
Nickel Chromium, Annealed	3250	35	50	10°	8°	8°	12°
Chrome Vanadium, Annealed	6145	35	50	10°	8°	8°	12°

math free calculations:

A simple option to doing the math is to refer to the “Table of Cutting Speeds”. For the same material, 3” stock with a desired surface speed of 120 FPM, we can simply look up the needed RPM referencing the diameter in the left column and the surface speed across the top of the chart. Doing so we come up with 164 rpm for the spindle speed close enough to the 152 rpm determined by the math above.

FIGURE 56

TABLE OF CUTTING SPEEDS

Correct Spindle Speeds to Give Approximately the Surface Speeds Shown

Diameter of Work Inches	*Surface Speed in Feet Per Minute* 30	40	50	60	70	80	100	120	150	200	300	500
1/16..........	2072	—	—	—	—	—	—	—	—	—	—	—
1/8...........	805	1270	1270	2072	2072	2072	—	—	—	—	—	—
3/16..........	685	805	805	1270	1270	1270	2072	2072	—	—	—	—
1/4...........	418	685	805	805	805	1270	1270	2072	2072	—	—	—
3/8...........	266	418	500	685	685	805	805	1270	1270	2072	—	—
1/2...........	266	266	418	418	500	685	685	805	1270	1270	2072	—
5/8...........	164	266	266	418	418	500	685	685	805	1270	2072	—
3/4...........	164	164	266	266	418	418	500	685	805	1270	1270	2072
1.............	112	164	164	266	266	266	418	418	500	805	1270	2072
1¼...........	83	112	164	164	164	266	266	418	418	685	805	1270
1½...........	70	112	112	164	164	164	266	266	418	500	805	1270
1¾...........	70	83	112	112	164	164	164	266	266	418	685	1270
2.............	45	70	83	112	112	164	164	266	266	418	500	805
2½...........	45	70	70	83	112	112	164	164	266	266	418	805
3.............	45	45	70	70	83	112	112	164	164	266	418	685
3½...........	28	45	45	70	70	83	112	112	164	164	266	500
4.............	28	45	45	45	70	70	83	112	164	164	266	500
5.............	28	28	45	45	45	70	70	83	112	164	266	418
6.............	28	28	28	45	45	45	70	70	83	112	164	266
7.............	28	28	28	28	45	45	45	70	83	112	164	266
8.............	28	28	28	28	28	45	45	45	70	83	164	266
9.............	28	28	28	28	28	28	45	45	70	83	112	164
10............	28	28	28	28	28	28	45	45	45	70	112	164

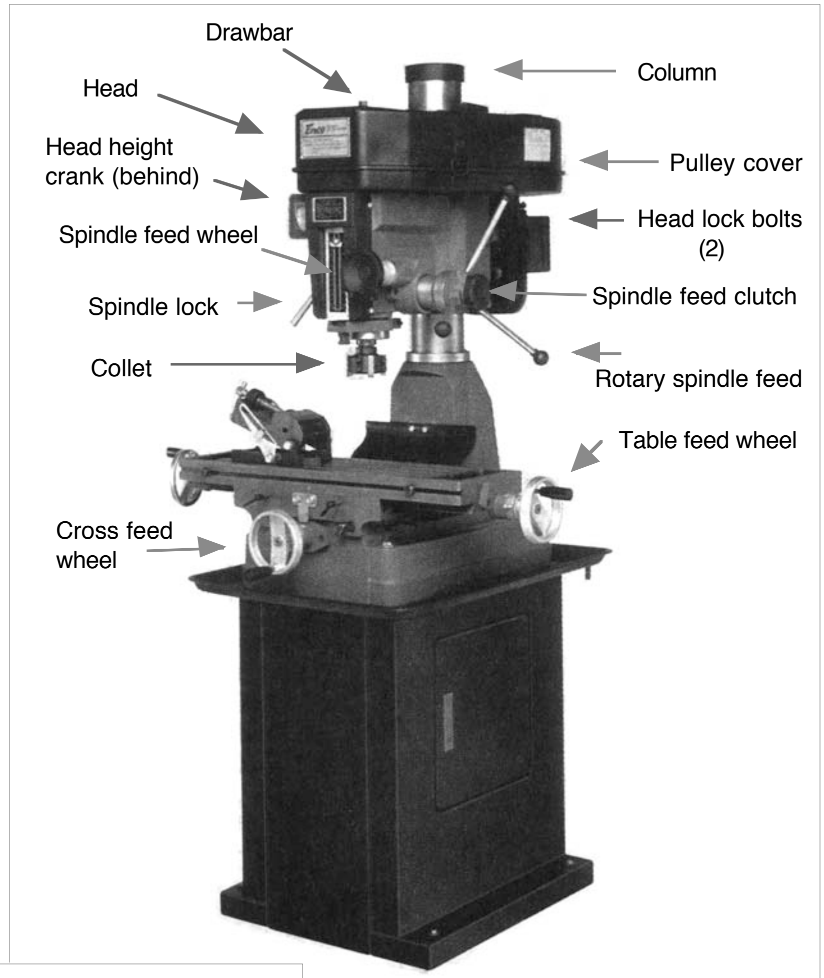
Speeds for Milling or drilling would normally be 2/3rds of the surface speed for turning. In this example (if we were actually drilling a 3” hole or using a 3” end mill) we would want a spindle speed of 100 rpm (slower is better when drilling metals). Speeds for various diameters of mill cutters can be determined by referencing the “Table of Cutting Speeds for Milling” on a later page.

The Milling Machine:

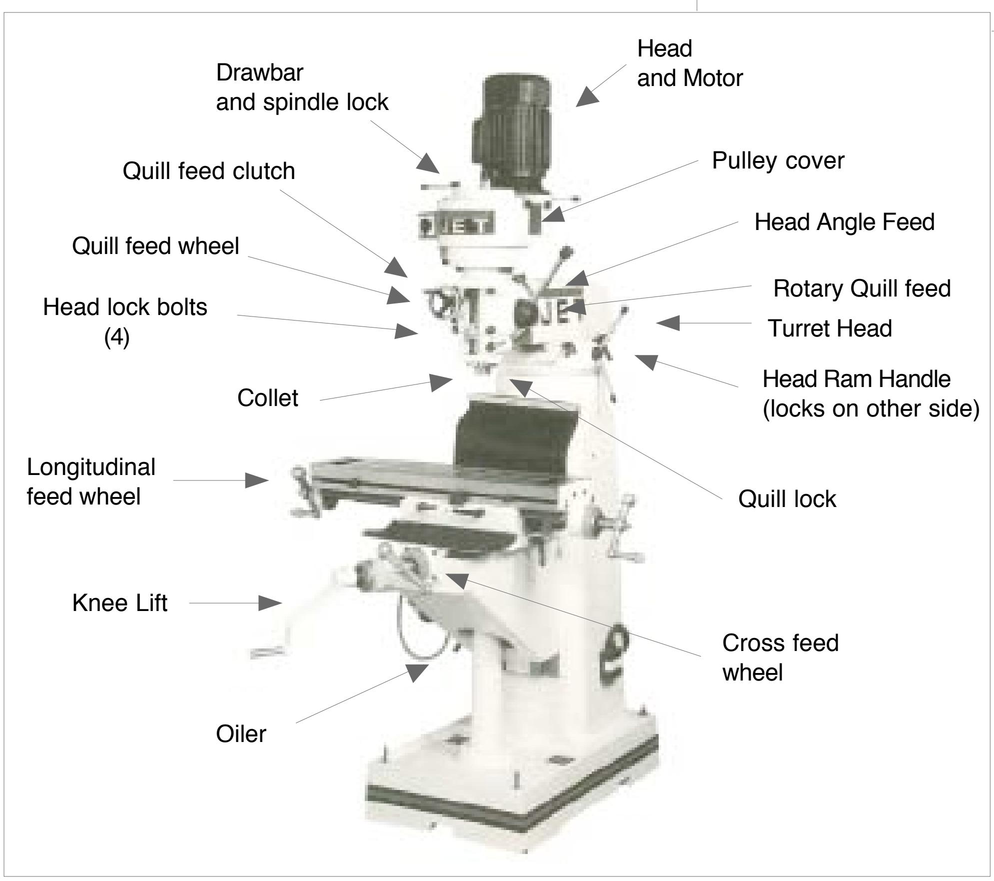
The milling machine is much the same for metal as a router is for wood. While there are not a lot of similar profiles to router bits it is used for squaring, notching, cutting channels, surfacing, boring, rounding and chamfering. Anything that cannot be accomplished on the lathe can probably be done on a mill short of compound curved surfaces.

We have two machines. The Enco is actually classified as a “mill/drill” because the “Z” (vertical) axis is adjusted by moving the machine head or quill up and down. The Jet Mill is a true “knee” mill lifting the the work in the “Z” axis up to the cutter. While the Jet Knee mill still has an adjustable “quill” it is rarely used to position cutters and always locked (it can be used for drilling and boring operations).

Similar to the carriage controls of the lathe the mill has a table feed (longitudinal: x) and cross feed (in and out: y) while the cutting depth is controlled by the hand wheel spindle feed on the Mill/Drill and the Knee on the Jet mill (up and down: z). The head lift on the mill/drill is only used for rough adjustment bringing the cutters closer to the work while the feed wheel is use for cutter position. The rotary spindle feed on the mill/drill is only used for drilling.



Standard Mill/Drill



Jet Knee Mill

Speed relative to cutter diameter and material are the only “set” values for a manual “tool room” mill. Speed is set by adjusting the step pulley drive under the top cover. Cut depth is normally limited to ten to twenty thousandths (0.010” to 0.020”). Finish cuts, as with any machine tool, are normally two to three thousandths (0.002” to 0.003”).

Feed rate is then totally in the operators “feel” of the cut. It should be readily clearing chips and “free” cutting with no bogging down or stiffness in the feeds. The smaller in diameter of the bit, the deeper the cut, and the harder the material to cut, the slower the feed needs to be.

Milling cutters should **ONLY** be mounted with collets, never in the drill chuck. The collet is inserted into the bottom of the spindle and held in place with the drawbar which is also used to tighten the collet around the milling cutter

MAINTENANCE AND LUBRICATION: All Machines

Anyone not following these maintenance instructions WILL loose access to the machines!

CLEAN!

Anyone who uses the Machine Tools MUST clean all swarf from the machines and accessories AND wipe them down with a clean rag and WD-40. This includes ALL machined and painted surfaces. Cover with supplied tarps.

Close and lock all machine cabinets.

LUBRICATION! CAUTION!

Do not operate the mill before lubricating the machine fully. Failure to comply may cause damage to the machine.

Reference Fig. 3 for parts of the mill to lubricate:

A. Spindle Bearings - fill oil cup once daily with Mobil Light Spindle Oil.
B. Oil Pump - Pump oil with release handle once for every hour of operation. Way surfaces and leadscrews are lubricated in this manner. Fill reservoir as needed by removing cap on top of tank and filling with Mobil Vactra Oil NO.2.
C. Knee Leadscrew - lubricate with Mobilith@AW2 once a week.

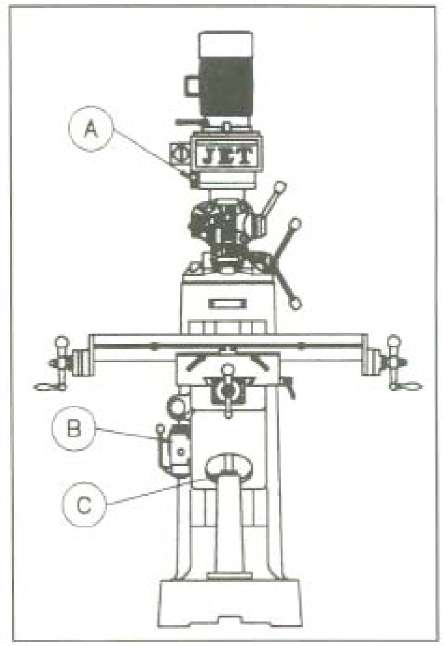


Fig. 3

JET Knee Mill Controls:

A. *Belt Cover Lock Knobs* (A, Fig. 4) located on the right side of the head. Loosen and remove belt cover to change belt position on the pulleys.
B. *Spindle Brake* (B, Fig. 4) located on left side of the head. Move in either direction to stop spindle once power has been turned off.
C. *Quill Feed Handle* (C, Fig. 4) located on the right side of the head. Rotate counterclockwise to lower spindle. Return spring will retract the spindle automatically once the handle is released.
D. *Quill Lock* (0, Fig. 5) located on the right side of the head. Rotate the handle clockwise to lock the quill in a desired position. Rotate the handle counterclockwise to release.
E. *Micrometer Adjusting Nut* (E, Fig. 4) located on the front of the head. Use for setting specific spindle depth.
F. *Manual Fine Feed* (F, Fig. 4) located on the left side of the head. Must engage fine feed (I, Fig. 5) for handwheel to function.
G. *Quill Stop* (G, Fig. 4) located on the front of the head. Used in conjunction with micrometer adjusting nut for predetermined depth.
H. *Reversing Switch* (H, Fig. 5) located on the left side of the head. Switches rotation of spindle and low and high-speed option.
I. *Fine Feed Engagement* (I, Fig. 5) located on the left side of the head. Turn clockwise until tight. This engages the manual fine feed.
J. *Draw Bar* (J, Fig. 5) located on the top of the head. This is used to tighten a R-8 collet or R-8 tool into the quill. Tighten draw bar enough to hold tool securely during milling operations.

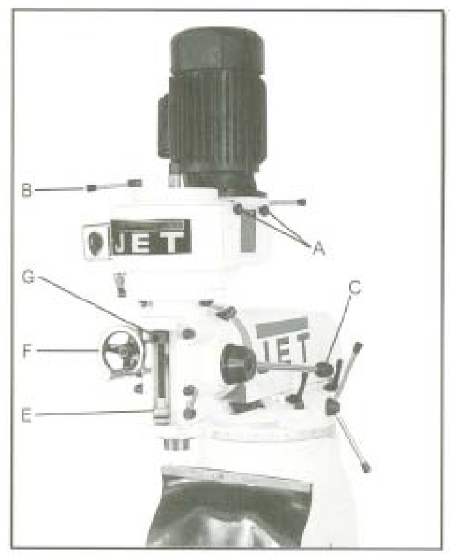


Fig. 4

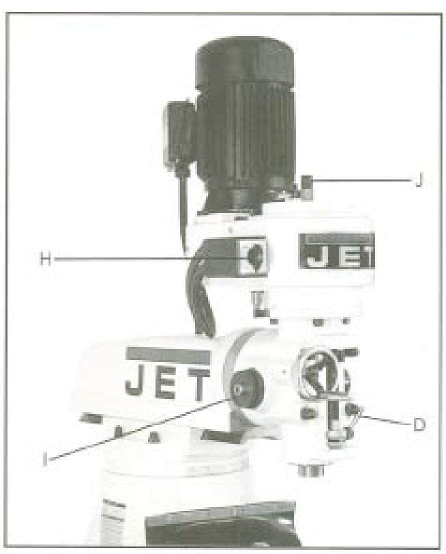


Fig. 5

<u>JET Knee Mill Controls:</u> continued

K. *Longitudinal Movement* (J, Fig. 6) handles located on opposite ends of the table. This controls the X axis.
L. *Cross Movement* (K, Fig. 6) handle located directly in front of the machine. This controls the Y axis.
M. *Knee Handle* (L, Fig. 6) this raises the table up and down.

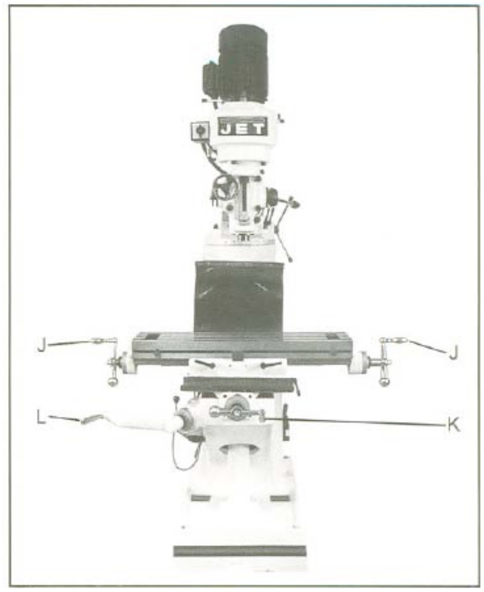


Fig. 6

Position of Ram:

CAUTION!

Ram must be locked securely after setting!
The Ram can be moved to extend the cutter further over the work surface.
To move the Ram loosen the two handles (A, Fig. 8; other side on our machine), and turn handle (B, Fig. 8) to drive the ram to the desired position.
Note: It is recommended while doing heavy milling work the head should be left as close to face of turret as possible. Maximum rigidity is then obtained.

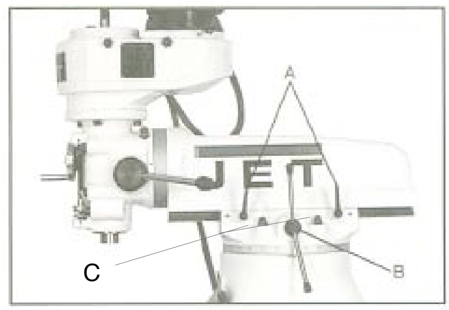


Fig. 8

Position of Turret Head:

CAUTION!

Turret must be locked securely after setting!
The Turret head can be rotated to facilitate access to projects.
To rotate loosen turret lock nuts (C, Fig. 8), rotate to position, then retighten the lock nuts.

Changing Speeds:

1. Unscrew three screws (A, Fig. 4) and remove belt cover.
2. Loosen hex nut (A, Fig. 7).
3. Take the tension off the belt by moving handle (B, Fig. 7).
4. Use the RPM chart, located on the back of the belt cover, to place the belt in the desired position.
5. Place tension on the belt by moving handle.
6. Tighten hex nut.
7. Always replace the belt cover!

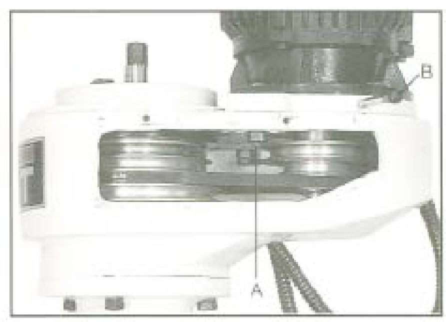


Fig. 7

FIG. 231

TABLE OF CUTTING SPEEDS FOR MILLING

Actual Speeds Obtainable on the Atlas Lathe to Give Approximately the Surface Speeds Shown

Size of Milling Cutter	*Surface Speed in Feet Per Minute* 30	40	50	60	70	80	100	120	150	200	300
1/4" End Mill...	418	685	805	805	805	1270	1270	2072	2072	——	——
5/16" End Mill...	418	500	685	685	805	805	1270	1270	2072	——	——
3/8" End Mill...	266	418	500	685	685	805	805	1270	1270	2072	——
7/16" End Mill...	266	266	418	500	500	685	805	805	1270	2072	——
1/2" End Mill...	164	266	418	418	500	685	685	805	1270	1270	2072
5/8" End Mill...	164	266	266	418	418	500	685	685	805	1270	2072
Woodruff Cutters											
1/2" x 1/8"......	164	266	418	418	500	685	685	805	1270	1270	2072
3/4" x 3/16".....	164	164	266	266	418	418	500	685	805	1270	1270
1" x 1/4"........	112	164	164	266	266	266	418	418	500	805	1270
1-1/8" x 5/16"....	112	112	164	164	266	266	266	418	500	685	805
1-1/4" x 3/8".....	83	112	164	164	164	266	266	418	418	685	805
Angular Cutters											
1-1/4" x 7/16"....	83	112	164	164	164	266	266	418	418	685	805
1-5/8" x 9/16"....	70	83	112	112	164	164	266	266	266	418	685

Table of Surface Cutting Speeds for Various Materials

Must be matched to spindle speeds for tool diameter (see charts)

Material	Surface Speed (feet per Minute)	Cuts: Roughing/Finish	Material specific note	Lubricant
Cast Iron	50 FPM	0.062 to 0.125/0.15	First cut heavy (.062+) to cut beneath scale	Iron specific or none
Stainless	40 FPM	0.062/ 0.10;	Make one finish pass to avoid hardening	Standard; WD-40
Copper	120 FPM	0.030 /0.010	0.003 finish cut	WD-40 or none
Brass and Copper Alloys: Turning requires back rack on tool bits				
Free Cutting	300 FPM	0.062 to 0.125	Finish fine at .010 to .003	WD-40 or none
Yellow Brass	200 FPM	0.062 to 0.125	Finish fine at .010 to .003	WD-40 or none
Silicone and Phosphor Bronze (difficult to machine; cobalt bits are good)				
Commercial	80	0.015 to 0.125	Finish fine at .010	WD-40 or none
Cast Bronze	50 FPM	0.030	Finish fine at .010	WD-40 or none
Aluminum Alloys: 200 to 800 FPM; Roughing cuts up to 0.25; finish .003				WD-40
6061	600 to 800 FPM		Best for Machining; most extruded material and rod;	
52xx	300 to 500 FPM		Good for machining; most plate	
2xxx	200 to 300 FPM		Pure aluminum, difficult to machine	
Cast	200 FPM		Very pure Aluminum, difficult to machine	
Plastics:				
Heat cast (i.e. Bakelite)	100 to 120 FPM		Cooling necessary	None or Paraffin
Cast (i.e. Acrylic rod)	up to 200 FPM		Cooling necessary	None or Paraffin
Drilling Acrylic	Wood drill speeds		Hole will shrink after cooling	Paraffin

NOTE: All materials expand as they heat up during cutting. The tightness of the material against the tailstock of a lathe must be checked periodically. This is especially important with Stainless Steel and Aluminum. When machining these materials final measurements should never be taken before the material has cooled to the touch.