T H E S I S

on

UNIT INSTRUCTION SHEETS AND A COURSE
OF STUDY BASED UPON AN ANALYSIS OF
THE MACHINISTS' TRADE

Submitted to
the

OREGON STATE AGRICULTURAL COLLEGE

In partial fulfillment of
the requirements for the
Degree of

MASTER OF SCIENCE

by

Oscar A. Trygg
July 25, 1932

APPROVED:

[redacted]

Professor of Industrial Education.

[redacted]

Chairman of Committee on Graduate Study.

ACKNOWLEDGMENT

The author wishes to express his sincere appreciation to Professor George B. Cox for his kindly and helpful assistance in the preparation of this thesis.

TABLE OF CONTENTS

SECTION I.

INTRODUCTION

A COURSE OF STUDY FOR MACHINE SHOP WORK
IN A
TECHNICAL HIGH SCHOOL

The teaching of a trade always involves the solution of many problems concerned with subject-matter and equipment. Before a satisfactory beginning can be made, the instructor must decide:

1. What to teach, and
2. How to teach it.

The most logical way to decide on what to teach is by means of a trade analysis, listing all of the things a person must know and be able to do in order to become proficient in the trade.

It is impossible to teach all the things the student should know, or to develop all the skills that he should have, in the time alloted to the average shop course. It is, therefore, necessary for the instructor to pick from the total content those things that he deems most necessary for the student to acquire.

The things selected would vary somewhat, depending upon the exact purpose of the course. If the purpose is exploration and guidance, the content and emphasis would not be the same as for trade training purposes. If the purpose is exploration and guidance, the sampling would be as wide as possible in order that the student may determine his interest in, and aptitude for, the trade in as short a time as possible.

If the purpose is trade training, more emphasis must be placed on developing a degree of skill in the most commonly used and fundamental operations. The organization and presentation should be such that the student will also develop the ability to secure further information for himself when on the job.

The accompanying chart lists all the basic machines used in machine shop work. Under each machine is listed all the operations usually performed on that machine. The items of information that the machinist should acquire are listed under appropriate headings. When a course is to be organized, the instructor selects from this total content the operations and items of information that will best meet the aims of the course.

The next step is to select a list of projects or jobs that can be used as a medium by which to teach the desired operations and items of information. These are placed at the bottom of the chart, as nearly as possible in the order of difficulty of the operations involved.

The next step is to indicate in the blank spaces provided, those operations and items of information that are involved in the making of each project. The order in which the operations should be performed is indicated by the sequence of numbers listed in these spaces.

After the content of the course has been determined, the next step is to determine the method of teaching. The large classes found in modern shops, and the varying rates at which students work, make the old methods of class demonstrations and individual help inadequate. Individual instruction sheets have been developed in order to improve the instruction under these conditions.

An instruction sheet is an attempt to put down in writing and by means of illustrations the information necessary to do a job or perform an operation.

It is not intended that the instruction sheets should displace the instructor. Demonstrations will still be necessary, but the instruction sheets will supplement the demonstrations and will be a great help to those students who may be either ahead of or behind the average of the class. They will also assist in cases of absence from classes. Several kinds of instruction sheets have recently been developed. Professor R. W. Selvidge in his book, "Individual Instruction Sheets", classifies them as follows:

<u>The Instruction Sheet</u>.

(1) Operation Sheets. Those instruction sheets that tell how to perform the manipulative processes or operations we wish to teach.

(2) Information Sheets. Instruction sheets that deal with items of information or simple statements of fact.

(3) Assignment Sheets. Instruction sheets, often composed largely of questions, designed to direct observation, reading, and drill.

b. Sheets dealing with jobs as units.

(1) Job Sheets. Instruction sheets that tell how to do complete jobs which may involve a number of operations or units of instruction, are called job sheets. They are especially designed to secure production. They also may be used on small unrelated jobs requiring little skill, such as home mechanics.

Operation sheets and information sheets seem best suited to machine shop work because one fairly complete set can be used for a large variety of jobs.

The method of procedure attempted in this course is to have the instructor demonstrate a few of the simpler operations at the beginning of the course. His object should be to get each member of the class started on some simple project as soon as possible. This can be done most rapidly by demonstrating bench work and lathe work first, because these tools and machines are usually the most numerous. He can then demonstrate new operations as soon as he thinks the mythical "class average" is ready for them. The instruction sheets will be a great help in taking care of those pupils who are ahead of the average and also those who are behind.

At first an operation schedule of the job assigned should be given to each student. This can be copied from the chart. It should contain all the operations involved in the job. They should be listed in the proper sequence. Reference to the instruction sheets explaining the new operations and items of information should be given.

Later, after the student has become familiar with the tools and machines to be used on a given job, he should be required to write his own operation schedule in order that he may gain experience in planning his work. His schedule should be checked by the instructor in order to avoid discouraging mistakes. Such procedure will bring the greatest flexibility to the teaching program by way of assisting individual progress to keep pace with individual ability. It will also bring out the maximum of initiative and planning ability necessary to the continued success of those who care to advance beyond the stage of unthinkingly following plans of procedure prepared by others.

SECTION II.

GRAPHIC ANALYSIS OF MACHINIST'S TRADE

SECTION III.

INSTRUCTION SHEETS FOR TEACHING THE TRADE CONTENT

INSTRUCTION SHEETS FOR TEACHING THE TRADE CONTENT.

It is obvious that a complete set of instruction sheets, covering all the operations and items of information listed in the chart, could not be written in the time allotted to this thesis. It was possible, however, to select a few operations or items of information from each of the more important blocks and to write instruction sheets covering them. The operations selected in each block have been presented in the sequence in which they would ordinarily be taught. This list is complete enough to indicate how a complete set can be worked out.

An operation schedule of one job is included as a sample. Other operation schedules should be gotten out by the instructor after his list of jobs has been selected. These operation schedules will vary somewhat for each shop, depending upon the project selected, the equipment available, and the ability and experience of the students in the class. These operation schedules will be most useful in the presentation of elementary projects or jobs. The schedule of operations for the more advanced jobs should be made by the student, as an outgrowth of the broadening horizon of experience in the work. Such procedure will insure his having thought out the processes to be used and will be a decided aid in the execution of the work assigned.

Name____________________________ No.___

Course________________Date________________Appd.____________

OPERATION SCHEDULE

for

SETTING HAMMER

1. Lay out pattern on one side of stock.

2. Plane front and back.

3. Plane angular portion.

4. Plane face.

5. Plane sides.

6. Lay out holes to be drilled for handle.

7. Drill holes.

8. File hole for handle to the desired shape.

9. Draw file all surfaces.

10. File corners just enough to take off sharp edges.

11. Polish.

12. Harden.

13. Temper.

REFERENCES:

Operation Sheet MS-10.

Information Sheets MS-8, MS-9.

Operation MS-1

LOCATING CENTERS FOR TURNING IN THE ENGINE LATHE

INFORMATION:

Many jobs that are to be machined in the lathe are held between lathe centers. Holes must be drilled and countersunk on the ends of the stock to form bearing surfaces for the lathe centers, if this method is to be used.

If these holes are not accurately centered, the outside of the stock will rotate eccentrically, causing the cutting tool to take a thin chip on one side and a thicker chip on the other. This condition is apt to cause the center holes to wear out of round.

PROCEDURE:

1. Remove rough spots on ends of stock. Use a file for steel, or a coarse grinding wheel for cast iron stock.

2. Rub chalk on both ends so that lines drawn on them may be more easily seen.

3. Grip stock in a vise in a vertical position, with one end projecting above vise jaws.

4. Set hermaphrodite caliper to about the radius of the stock and scribe four arcs as indicated in Fig. 1.

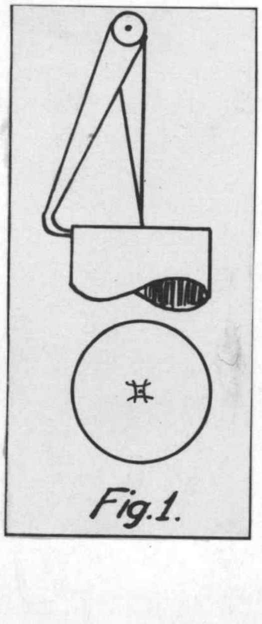


Fig.1.

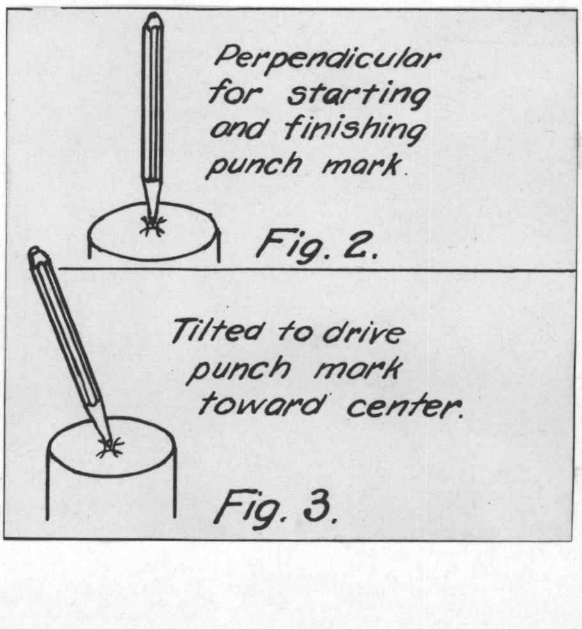


Fig. 2.

Fig. 3.

Operation MS-1

5. Set a well-sharpened center punch as near as possible to the center of the "square" made by these arcs, and strike it lightly with a hammer, Fig. 2.

Note: If punch "pulls" to one side, tilt it so as to draw the punch mark toward the center, Fig. 3.

6. With heavier blows enlarge the mark enough to support the stock between the points of the lathe centers when turned by hand.

7. Repeat steps 3, 4, 5, and 6, on the other end of the stock.

8. Place the piece between centers in the lathe and revolve by hand, at the same time holding a piece of chalk lightly against one end as indicated in Fig. 4. Repeat on other end.

Note: This test will require well-pointed lathe centers.

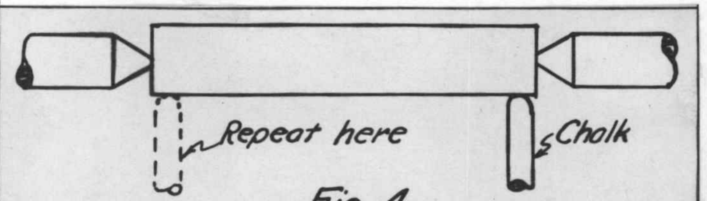


Fig. 4.

9. If the stock does not run true, the "high spot" will be indicated by a chalk mark.

10. If not <u>reasonably</u> true, place stock in vise again and draw punch mark toward "high spot" by the amount estimated as necessary to center it.

Note: If center must be moved as much as 1/16" it will be better to place the center punch in the new position and begin anew. In some cases it may even be necessary to remove the old mark by filing or grinding.

11. Repeat until stock runs reasonably true at both ends.

12. The stock is now ready for the drilling of center holes. See Operation MS-2.

QUESTIONS:

1. What are some other methods of centering stock? (Ref. #1, p. 84).

2. How would you center a bolt, with its head forged off center, so that it would run true? (Ref. #1, p. 86).

3. Does it make any difference if the hermaphrodite caliper is set a little more or a little less than the radius?

REFERENCES:

1. Burghardt, H. D. - Machine Tool Operation, Vol. I.

2. Jones, H. A. - Machine Shop Practice, Book I.

Operation MS-2

DRILLING AND COUNTERSINKING CENTER HOLES IN THE ENGINE LATHE

INFORMATION:

Drilling and countersinking for lathe centers is usually done in one operation by using a combination drill and countersink, Fig. 1. The countersink has an included angle of 60°, the same as the lathe centers. Combination drill and countersink kits come in various sizes. The size required is determined by the size of the stock to be turned and the amount of metal to be removed. The following table will serve as a guide to the beginner.

Diameter of work	Large Diameter of Countersunk Hole	Diameter of Drill	Diameter of Body
3/16" to 5/16"	1/8"	1/16"	13/64"
3/8" to 1"	3/16"	3/32"	3/10"
1-1/4"to 2"	1/4"	1/8"	3/10"
2-1/4"to 4"	5/16"	5/32"	7/16"

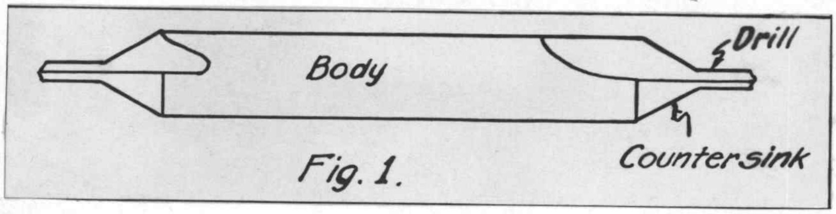


Fig. 1.

If a combination drill and countersink is not available, a common drill of the proper size and a 60° countersink can be used as separate tools.

PROCEDURE:

1. Remove live center. Clean out hole with a rag. Insert a drill chuck having a shank of proper size and taper, and "slap" into place.

2. Fasten the combination drill and countersink firmly in the chuck leaving about one-third of its length projecting.

3. Check to see that the tail stock is in line. This is indicated by marks near the base. If it is out of line have the foreman or instructor

show you how to line it up

4. Adjust the work between the dead center and the combination drill and countersink. This is done by first loosening the tail-stock clamping bolt, (see wall chart of lathe), and sliding tail-stock to approximately the right position. Then fasten the tail-stock and make final adjustment by turning the tail-stock hand wheel until the work is held loosely in place between the dead center and the drill, one centerpunch mark on each.

5. Hold the work with the left hand, allowing the tool rest to support the wrist. Start the lathe (fastest speed) and gradually feed by turning tail-stock hand wheel slowly until center is reamed to the correct depth, Fig. 2. Hold the work firmly against the dead center and draw it back by turning tail-stock hand wheel.

Note: If the work is not held firmly against tail-stock center when drawing it back, the drill point is very apt to wedge and break.

6. Reverse ends and repeat, resting center just drilled upon the dead center while the second end is drilled and countersunk.

Suggestion: The drill press may also be used for drilling and reaming center holes. The lower end of the stock should then be placed on a cone center rather than on the drill press table.

Caution: Do not wear loose clothing that might catch in revolving parts of lathe. Sleeves should be rolled up and neckties tucked in. Serious injury might result from carelessness in this respect.

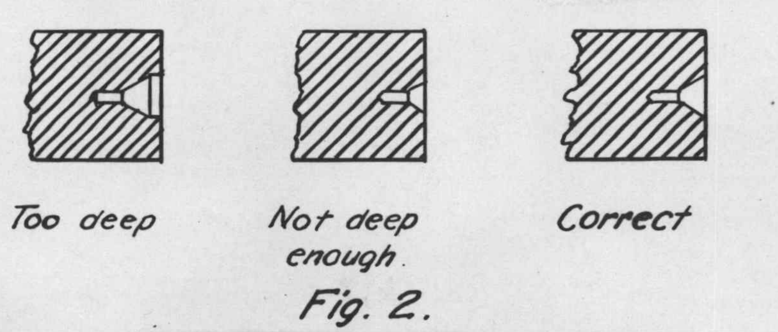


Fig. 2.

QUESTIONS:

1. Why is it more efficient to use a combination drill and countersink than to use a drill and a countersink as separate tools?

2. Why does a drill break so easily?

3. What would happen if one attempted to drill out the broken point of a drill without first annealing it?

4. What will happen if the center hole is drilled too deep?

REFERENCES:

1. Burghardt, H. D. - Machine Tool Operation, p. 271.

Operation MS-3

FACING AND SQUARING CYLINDER ENDS IN THE ENGINE LATHE

INFORMATION:

After centers have been located and drilled on a piece of stock that is to be machined in the lathe, the next operation is usually to face and square the ends.

Stock, as it comes from the foundry, mill, or saw, is rough and uneven on the ends. If it were allowed to rotate on centers in this condition for any appreciable length of time it would cause unequal wear on both the dead center and the center hole.

PROCEDURE:

1. Fasten the dog on the stock near one end, placing it so that the tail of the dog will enter the slot in the faceplate without binding. Fig. 1.

Note: Dogs of various sizes may be obtained from the tool room.

2. Check to see that the centers are in allignment. Approximate Method: Unclamp tail-stock (see wall chart) and move upper part by screws "C" and "C' " (not shown) until zero lines at "D" coincide.

Fig. 1. Fig. 2. Fig. 3.

3. Put a few drops of white lead in the center hole that is to rotate on the dead center.

4. Place the work between centers, and bring the dead center up until there is no "shake" of the work between centers; neither should it be too tight

because the work must be free to turn between the centers.

Note: A good test is to rotate the work slowly, pulling the belt by hand. If the dog will drop from one side of the faceplate slot to the other, after it passes the top center, the work is not too tight.

5. Tighten the tail-stock spindle locking lever (see wall chart) to hold stock securely.

6. Set a right-hand side tool in a right-hand tool holder. Place the tool well back to prevent springing, Figs. 2 and 3.

7. Set the tool holder in the tool post so that the point of the cutting tool will strike the end of the work first, making an angle of about 5°. Fig. 4. The height of the point should be about even with the center of the work. Fig. 5.

8. Set the driving mechanism to secure the correct spindle speed. (See Information Sheet MS-5).

9. Start the lathe. With the right-hand on cross feed handle and the left-hand on the longitudinal feed hand wheel, bring the tool point up close to the center of the work. Fig. 4.

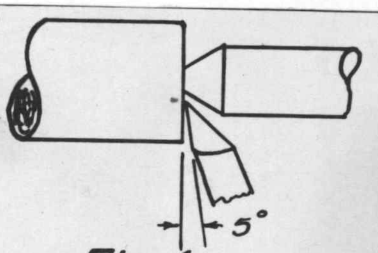


Fig. 4.

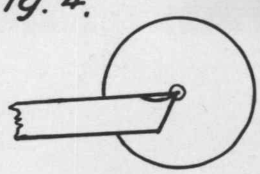


Fig. 5.

10. Move the tool point into the work with the longitudinal feed until a light chip is taken around the center.

11. Hold the longitudinal feed stationary (on large stock, lock carriage with lock screw) and move tool slowly outward with hand screw.

12. Finish with a light cut and slow feed to get a smooth surface.

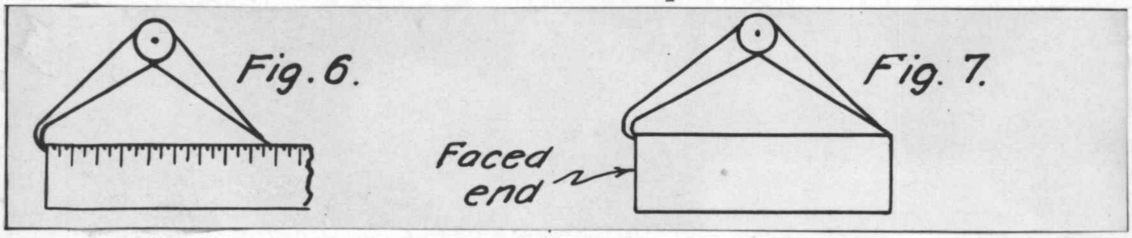
13. To remove small fin around center hole, return the side tool to the surface just faced and move it down against the fin. Ease off the dead center

about 1/32" and clamp the tail-stock spindle. Start the lathe. The work will have a tendency to ride on the tool and clean off the fin.

14. Remove the stock from the lathe and remove the dog from the stock.

15. Set the hermaphrodite caliper to the length required and mark the finished length on the stock. Figs. 6 and 7.

Note: Two methods are used to make it easier to see the marks: (a) if the work is rough or scaled, rub on chalk; (b) if the work is smooth, such as cold rolled steel, use a copper sulphate solution. For long pieces use a steel scale and a scriber instead of the calipers.



16. Place the dog on the faced end and replace the work in the lathe.

17. Square the second end to length, working to the line established in step 15, by repeating steps 9 to 13.

QUESTIONS:

1. What is another method of facing?

   Jones, H. A. - Machine Shop Practice, p. 56.

2. Why is it important to have the lathe centers in line?

   Burghardt, H. D. - Machine Tool Operation, p. 93.

3. Why must one be careful not to move the longitudinal feed when facing?

4. What other method can be used to permit facing the end entirely up to the center hole without backing off the dead center?

   Jones, H. A. - Machine Shop Practice, p. 56.

Operation MS-4

TO TURN A STRAIGHT CYLINDER IN THE ENGINE LATHE

INFORMATION:

After the ends have been faced, the next operation is usually to machine the curved surface. This is called straight turning if the diameter remains the same throughout the length of the cut. It is called taper turning if the diameter is greater at one end than at the other. Straight turning is one of the basic operations of most frequent occurance in engine lathe work.

PROCEDURE:

1. Mount the work between centers as for facing ends (Steps 1 to 5, inclusive, Operation Sheet MS-3.)

2. Set a round-nosed turning tool in a tool holder. Set well back to prevent springing.

3. Set the tool holder in the tool post and move the whole tool post assembly close to the left end of the "T" slot in the compound rest. This allows the cutting tool to move closer to the dog without having the dog strike the compound rest, Fig. 1.

4. Slide the tool holder back and set the point of the tool even with, or slightly above the center of the stock and with the cutting edge at an angle of about 20° to the face of the work when seen from above. Fig. 1.

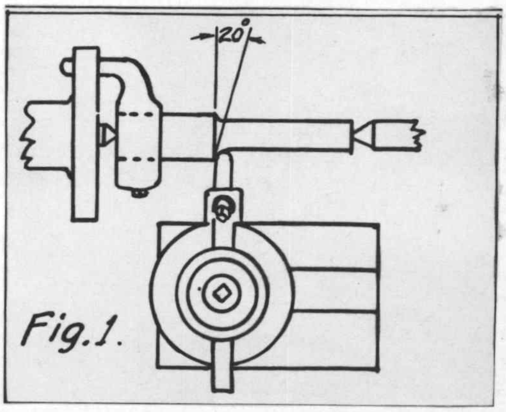


Fig.1.

Note: The height of the tool may be varied by moving the "rocker" in the tool post under the tool holder. Tighten tool post assembly.

5. Check allignment of lathe centers as follows: (An approximate method was given on Operation Sheet MS-3. A more accurate method follows.)

<u>Accurate Method</u>

a. Run the cutting tool up as close to the dog as possible without having the dog strike the tool rest, and take a short cut, removing just enough material to get under the scale.

Note: Read Information Sheet MS-5 to determine correct speed and feed.

b. Without moving the cross feed, back off the dead center to release the stock and run the carriage back to the dead center.

c. Replace the work on the dead center and take a short cut. Fig. 2.

d. Caliper both cuts. Figs. 2 and 3. Work carefully!

e. If they are not alike, move the tail-stock to correct the difference. If the end at the dead center is larger, move the tail-stock toward you. If it is smaller, move the tail-stock away from you.

f. Take another small cut at each end and caliper again.

g. Repeat these operations until both ends are the same diameter.

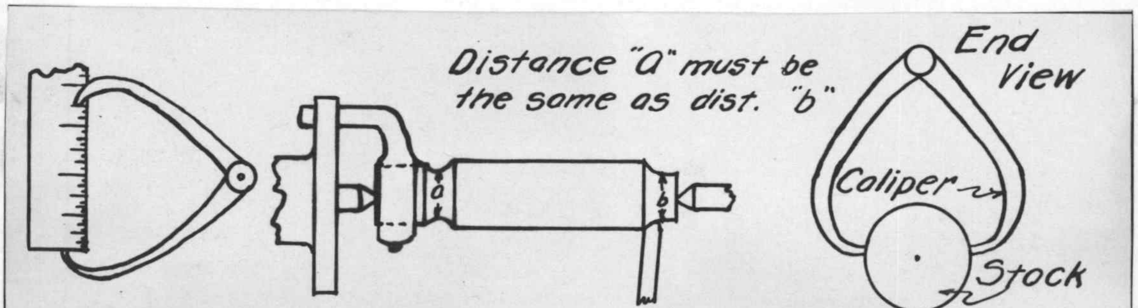


Fig. 1. Fig. 2. Fig. 3.

6. Set the caliper to 1/32" over the required diameter, Fig. 4, and by short trial cuts (about 1/8 of an inch long), bring the end of the work down to this diameter.

7. Have the foreman or instructor show you how to use the longitudinal power feed.

8. Take the roughing cut, as near to the dog as possible without having the dog strike the tool rest. Use the power feed and move tool from dead center toward dog.

9. Without disturbing the cross feed, remove stock from centers, return carriage to dead center, place the dog on opposite end, and replace stock with ends in reversed position.

10. Again engage power feed to move carriage toward live center. If the cross feed has not been moved, the two cuts should join smoothly at the center and leave a single diameter.

11. Run the cross feed back and with a few strokes of the oil stone "touch up" the cutting edge of the tool.

12. Readjust the cross feed until the tool just touches the rough turned surface of the work. Then run the tool off the work toward the dead center, moving the longitudinal feed only.

13. Measure the work with a micrometer to determine how much it is oversize, in thousandths of an inch.

14. Move the cross feed toward the dead center by one-half the number of thousandths that the stock is oversize.

Note: The graduated bushing on the cross feed reads in thousandths of an inch.

15. Take a short cut with hand feed, (about 1/8") and "mike" again. If the adjustments have been correctly made, the stock should be of the desired diameter. If correct, proceed to step 16.

If not correct, make slight adjustments of cross feed until the correct diameter is obtained.

Note: "Back lash" or lost motion is always found between any freely moving screw and nut. To get rid of lost motion, the screw should be moved back more than the desired setting and the lost motion taken up by returning the screw toward the stock to give the proper setting.

16. Decrease the speed of the longitudinal power feed, as explained by the instructor at step 8, and continue the cut.

17. Back off the dead center, remove the stock, and move the carriage back to the dead center without disturbing the cross feed adjustment.

18. Reverse ends of work. Place a thin piece of brass or copper under screw of dog to prevent it from marring the finished surface.

19. Replace the stock between centers and take finishing cut on second end.

20. Examine the point where the finishing cuts have joined. Any noticeable break will indicate careless work, or that the live center is not running true.

QUESTIONS:

1. What happens if the live center is not true? The dead center?

2. What would happen to the cutting tool if the point did not go below the scale all the way around when taking a roughing cut on cast iron?

3. Why be careful not to get the centers too tight?

4. When reducing the stock to exact size, why move the cross feed only <u>one-half</u> the number of thousandths that the diameter is to be reduced?

REFERENCES:

1. Burghardt, H. D. - <u>Machine Tool Operation</u> - pp. 91; 83; 99.

Information MS-5

SPEED, FEED, AND DEPTH OF CUT FOR LATHE WORK

GENERAL INFORMATION:

Many things should be considered in determining the correct cutting speed for a given job. Some of the more important are: hardness of the metal to be turned; the kind of steel of which the turning tool is made; the shape of the tool, and its heat treatment; the feed and depth of cut; the cooling medium used, if any; the power of the machine; and its design and condition. These various factors are scientifically determined only in production work involving many duplicate parts. For ordinary work, the machinist is guided by a table of average speeds, feeds, and depth of cut. Variation from this table will depend on his judgment as to modifying factors on a given job.

The following table gives the speed, feed, and depth of cut on cast iron and steel as recommended by Mr. F. W. Taylor, who made extensive experiments using high speed cutting tools. For carbon steel cutters these speeds must be reduced by one-half.

Depth of cut in inches	Feed in inches	Speed in feet per minute soft steel	medium steel	hard steel
1/16	1/64	548	274	125
	1/32	358	179	81
	1/16	235	117	53
3/32	1/64	467	234	106
	1/32	306	153	69
	1/16	200	100	45
	3/32	156	78	35
1/8	1/64	417	209	95
	1/32	273	136	62
	1/16	179	89	40
	3/32	140	70	31
3/16	1/64	362	181	82
	1/32	236	118	54
	1/16	155	77	35
1/4	1/64	328	164	74
	1/32	215	107	49
3/8	1/64	286	143	65

Information MS-5

Depth of cut in inches	Feed in inches	Speed in feet per minute		
		Soft cast iron	Medium cast iron	Hard cast iron
3/32	1/32	160	80	46
	1/16	110	55	32
	1/8	75	38	22
1/8	1/32	148	74	43
	1/16	104	52	32
	1/8	69	35	20
3/16	1/64	183	91	68
	1/32	135	67	39
	1/16	94	47	27
	1/8	64	32	19
1/4	1/64	171	85	50
	1/32	126	63	37
	1/16	88	44	25
	3/32	70	35	20
3/8	1/64	156	78	45
	1/32	116	58	34
	1/16	80	60	23

The following table gives the speeds recommended for some other metals by the South Bend Lathe Works. The speeds are based on an average turning feed.

		Heavy cut	Finishing cut	Threads
Brass	F.P.M.	150	200	55
Aluminum	"	200	300	50
Bronze	"	90	100	25

Ordinarily metal can be removed more quickly by using a coarse feed and a slower speed than by using a fine feed and a greater speed. Sometimes the cut must be comparatively light, either because the work is too frail and springy to withstand the strain of a heavy cut, or because the lathe does not have sufficient power.

The finishing cut should not be over 1/64" deep and the feed should be about 90 revolutions to each 1" of tool trave.

Cast iron and brass are machined dry. Lard oil, soda water, or a commercial cutting compound is used on steel. Kerosene or turpentine is used on aluminum.

TO SET LATHE TO A GIVEN SPEED

Eight or ten speeds are possible on most engine lathes. These speeds are all determined by the speed of the main drive shaft or by the speed of the motor on a lathe having an individual motor.

The spindle speeds for a lathe can be read directly by a speed indicator. These should be put on a chart for future reference.

Speed tables are given in F.P.M., or the number of feet per minute that the cutting tool passes across the surface of the work. This can be read directly by holding the wheel of a cut-meter against the revolving surface of the work, or it can be figured from the spindle speed and diameter of the work by using the following formula:

$$\frac{D \times 3.1416 \times R.P.M.}{12} = F.P.M.$$

D = diameter of stock.
R.P.M.= revolutions per minute.
F.P.M.= feet per minute
12 = constant which changes inches to feet.

The average lathe has only eight or ten speeds. When a desired speed is not exactly obtainable, the lathe should be set to the speed nearest the one desired.

Note: The beginner should run the lathe at considerably slower speeds until he has become familiar with the lathe and feels confident that he knows how to control it.

TO SET THE LATHE TO THE DESIRED FEED

Most modern lathes have a quick change gear box at the left end of the feed screw. This has an index plate which shows where to set the two levers to secure the desired feed. If the desired feed is not on the index plate, the one nearest to it should be used.

QUESTIONS:

1. If a cut-meter is not available, what other methods can be used to determine the surface speed directly?

   Smith, R. H. - Advanced Machine Work, p. 146.

2. What two methods are commonly used to secure the various spindle speeds on the engine lathes?

   Smith, R. H. - Advanced Machine Work, pp. 101-102.

3. How are the various feeds obtained on a lathe that does not have a quick change gear box?

   Smith, R. H. - Advanced Machine Work, p. 104.

THE MICROMETER CALIPER

INFORMATION:

The micrometer principle consists of a combination of an accurate screw and a graduated head by which fine measurements may be taken. It is applied to inside and outside micrometer calipers. It is also used on feed screws of lathes, planers, milling and grinding machines, and many other places where accurate measurements are needed.

The following explanations will refer specifically to the outside micrometer caliper because it is the most used by machinists. A person who understands this description should have no difficulty when reading other measuring devices based on the same principle.

Outside micrometer calipers come in various sizes. The different sizes are designated by the largest piece of work they will measure; i.e., a 1" micrometer has a range from 0 to 1", a 2" caliper has a range from 1" to 2", and so on up to 12". The 12" size is the largest usually listed by the manufacturer, although larger sizes can be obtained. The range of the screws in each case is usually 1".

Fig. 1 shows a sectional view of the inside construction of the head of the micrometer. The spindle "A" and thimble "B" are securely fastened together. There are forty threads to the inch on spindle "A". Therefore, it will have to make 40 revolutions in order to move lengthwise one inch. Turning it one revolution will move it 1/40" or .025".

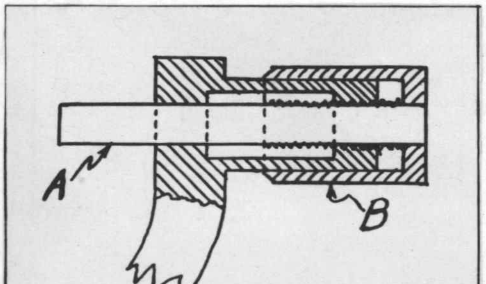


Fig. 1.

The graduations on sleeve "C", Fig. 2, are .025" apart. Turning the thimble one revolution will move the screw lengthwise one space along the graduated sleeve.

The thimble "B", Fig. 2, has 25 graduations. Therefore, if the thimble is rotated an amount equal to the distance between two of these graduations, it will move the screw lengthwise a distance equal to 1/25 of a revolution, or 1/25 of 1/40 of an inch.

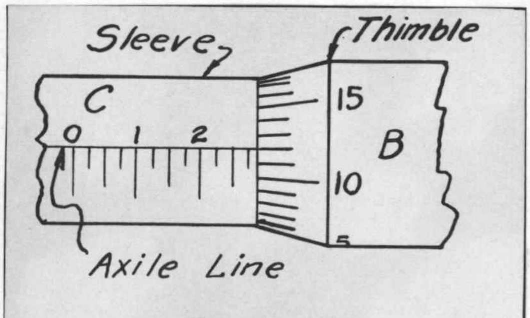


Fig. 2.

But 1/25 of 1/40 = 1/1000 inch. Every fourth graduation on sleeve "C" is numbered. The first numbered graduation represents .100", the second .200", and so on. To read a micrometer set as in Fig. 2, two is the largest visible figure on the sleeve indicating that the reading is greater than .200" but less than .300". Three additional spaces are visible on the sleeve. Each is equal to .025". Therefore, add 3 x .025" or .075" to .200" making a total of .275". The figure on the thimble opposite the axile line on the sleeve is 12. Since each of these spaces is equal to .001", .012" must be added to the .275" making a total of .287".

Some micrometers have a veriner scale on the sleeve in addition to the regular graduations, so that measurements to 1/10,000 part of an inch can be taken.

To read a micrometer of this type, first determine the number of the thousandths, as with the ordinary micrometer and then find a line on the veriner scale that exactly coincides with one on the thimble, Fig. 3. The number of this line represents the number of ten thousandths to be added to the number of thousandths obtained by reading in the ordinary way.

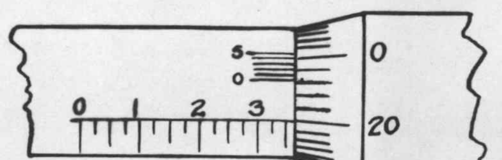


Fig. 3.

The reading in Fig. 3 would be .370" .0004" or .3704".

To measure with the micrometer, place the jaws over the object to be measured. Rotate the thimble with thumb and finger until a light, but distinct, contact is obtained. Some micrometers have a friction slip nut at the end of the thimble. On these, the thimble is turned up with the friction nut until the nut slips, at which point the setting should be correct.

When making a measurement, be sure that the micrometer is held squarely across the diameter of the work. Be sure that the object to be measured is free from chips and dirt and the contact points on the micrometer are clean.

Sometimes a micrometer is sprung so that the zero lines do not coincide when the anvil and screw are clean and in contact. To correct this error, loosen screw "A", Fig. 4, and adjust anvil "B" by turning screw "C" until the desired setting is obtained. Then tighten screw "A".

PROBLEM:

1. Secure several pieces of stock from the instructor.

2. Measure them carefully.

3. Write down your readings.

4. Have the instructor check your readings.

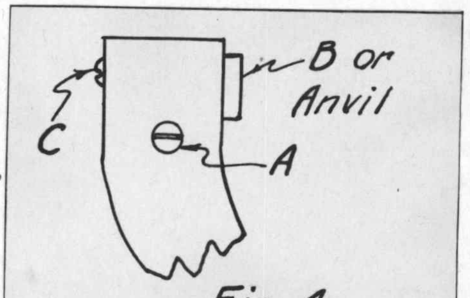


Fig. 4

QUESTIONS:

1. How can you measure a hole in a piece of stock accurate to .001"?

   Smith, R. H. - Advanced Machine Work, p. 441.

2. What other instrument is used in the machine shop for taking accurate measurements?

   Smith, R. H. - Advanced Machine Work, p. 211.

Operation MS-7

KNURLING IN THE LATHE

INFORMATION:

The curved surface of an object that is to be turned by hand is usually checked to form a better grip. The process of checking is called knurling. Some familiar examples of objects treated in this way are nuts, bolt heads, and handles of various kinds.

The knurling is done by two hardened steel wheels, mounted in a head and with milled edges that are pressed against the work as it is rotated in the lathe.

PROCEDURE:

1. Check to see that the lathe centers are approximately in line. (See Operation Sheet MS-4).

2. Place the piece to be knurled between centers in the same manner as for straight turning. Place the dog so as not to interfere with the knurling tool.

3. Select the desired knurling tool.

Note: Knurling tools may be secured in fine, medium, and coarse pitches. The medium is most used.

4. Set the knurling tool in the tool post so that the faces of the rollers are parallel with the surface to be knurled, and the center of the knurling tool is slightly below the center of the work to be knurled. Fasten securely.

5. Oil the surface to be knurled.

6. Run the rollers up so that about half of their width extends past the end of the work.

7. Start the lathe (medium speed).

8. Press the rollers into the work. Press hard at first and move carriage slightly to left and right. This will prevent the rollers from cutting double and spoiling the work.

9. Stop the lathe and inspect the work. Make sure that the rollers track.

10. Reduce the pressure slightly and start the lathe.

11. Throw in long feed (fast speed). Travel back and forth over surface until diamonds come nearly to a point.

QUESTIONS:

1. What is another method of knurling narrow thumb nuts?

   Smith, R. H. - Advanced Machine Work, p. 635.

2. What is another method of knurling a piece of small diameter, projecting from a chuck where pressure would throw it out of line?

   Burghardt, H. D. - Machine Tool Operation, p. 118.

Information MS-8

FEED, SPEED, AND DEPTH OF CUT FOR SHAPER WORK

INFORMATION:

Whenever a considerable amount of material is to be removed, the shaper should be made to "work"; that is, the speed, depth of cut, and feed should be proportioned to remove as big a chip as the shaper will drive, provided the nature of the work, the strength of the tool, and the way it is held will permit.

It is impossible to give a definite rule as to the proportion of feed to depth of cut, but the following general directions should be of help to the beginner.

A coarse feed and a light chip is not as good as a finer feed and a deeper chip, for the following reasons: a wide chip does not curl so easily and it takes more power. The tear in the metal is greater, thus producing a rougher surface.

A good rule to follow is to give as much feed as possible without producing a rough or torn surface, and then all the chip the machine and tool will stand, provided that amount of metal is to be removed.

The angle that the cutting edge of the tool makes with the surface of the work has considerable influence on the thickness of the chip.

Fig. 1 illustrates three chips taken with the same feed and the same depth of cut.

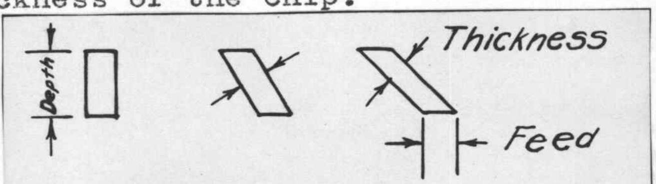


Fig 1

It has been found by experience that a tool with a cutting edge at about 20° from the perpendicular, and with a well-rounded nose, gives the best results for roughing either cast iron or steel.

The following table gives the average cutting speeds and feeds for the most commonly used metals.

Information MS-8

Table of Speeds and Feeds

From Burghardt - <u>Machine Tool Operation</u>
(Roughing Cuts)

Cutting Tool	Cast Iron		Machine Steel		Carbon Steel		Brass	
	Speed	Feed	Speed	Feed	Speed	Feed	Speed	Feed
High-speed steel	60	1/12	80	1/16	50	1/20	160	1/20
Carbon steel	30	1/16	40	1/20	25	1/25	100	1/20

The finishing cut is taken with a square-nose tool. A coarse feed and a very light chip (about 1/128") is best. The speed should be about the same as for the roughing cut.

QUESTIONS:

1. What determines the depth of chip that can be taken?

2. Why is a square-nosed tool used in finishing?

3. Why does a high speed destroy the cutting edge of the tool?

REFERENCES:

1. Machinery's Handbook, p. 877.

2. Smith, R. H. - Advanced Machine Work, p. 905.

SETTING THE SHAPER TO A GIVEN CUTTING SPEED

INFORMATION:

At a given speed of the driving gear, the shaper will make a constant number of strokes per minute whether they are long or short.

Therefore, to allow for the different lengths of stroke and also for the different speeds required on different metals and kinds of cuts, the shaper is provided with several speeds.

In a shaper with a cone pulley drive, the speed changes are obtained by shifting the belt to the various steps on the cone pulley.

In a shaper having an individual motor or a single pulley drive, the various speeds are obtained by shifting gears in a speed box. The smaller shapers of this type (14" or under) are usually provided with four speeds. In the larger shapers the number of speeds is doubled by using back gears.

The following table tells how to set the levers for the various number of strokes per minute obtainable on the large shapers built by the American Tool Works Company.

Speeds	Lower Lever (31)	Upper Lever (30)	Strokes per Minute
First	Left	Left	8
Second	Right	Left	11
Third	Left	Right	15
Fourth	Right	Right	23
Fifth	Left	Left	34
Sixth	Right	Left	49
Seventh	Left	Right	71
Eighth	Right	Right	105

Shapers vary so much in construction that it is impossible to give a concrete rule that would apply to all makes. Directions for setting a given make of shaper can usually be obtained from the handbook accompanying the shaper. If such a book is not available, the number of strokes for the various settings can be determined by actual count and a table made for future reference.

If the length of the stroke and the correct cutting speed for a given job is known, the calculations required to get the correct number of strokes per minute can be reduced to the following formula. The setting of the speed box for the American Shapers can then be obtained from the table previously given.

$$\text{Formula:} \quad N = \frac{C.S. \times 7}{L}$$

C.S. = cutting speed
L = length of stroke
N = number of strokes per minute

Example: Length of stroke 10"
Required cutting speed 40' per minute.

$$\text{Solution:} \quad N = \frac{40 \times 7}{10} = 28 \text{ strokes}$$

The setting nearest to 28 on the table is 23 strokes.

PROBLEMS:

1. Length of stroke 8"
   Required cutting speed 36'
   Find number of strokes.

   What will be the setting of levers to give a cutting speed nearest to this value?

2. Length of stroke 4"
   Required cutting speed 30'
   Find number of strokes.

3. Length of stroke 6"
   Required cutting speed 38'
   Find number of strokes.

Operation MS-10

TO PLANE RECTANGULAR WORK IN THE SHAPER

INFORMATION:

See Information Sheet on Planing Job.

The shaper is especially well adapted to small work which may be held in a vise bolted to a work table. The tool head is so constructed as to permit of horizontal, vertical, or angular cuts being taken. It is well suited for making small irregular shapes of various kinds. If a large number of duplicate parts are to be made, it is usually cheaper to have a special milling cutter made and have the work done on a milling machine.

Shapers are classified as to size (14", 16", 20", etc.) by the maximum length of the cut that may be taken. A standard shaper of a given size will hold and plane a cube of that size.

PROCEDURE:

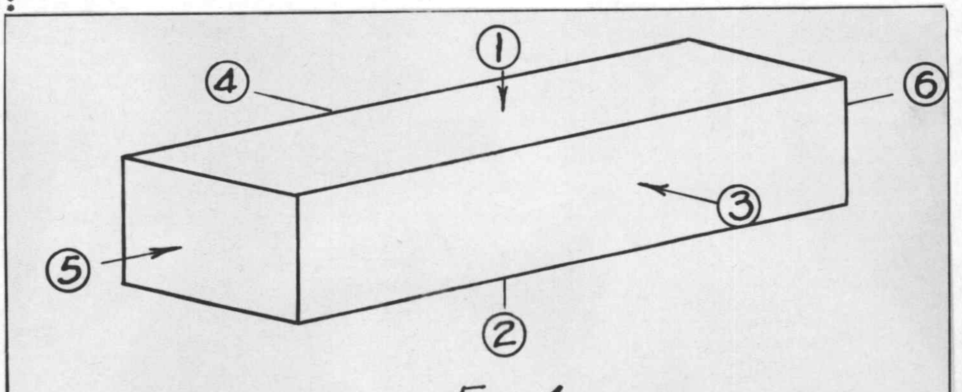


Fig. 1.

1. Clean and oil vise and shaper. Make sure that all metal chips are brushed away around the vise.

2. Set the vise so that the edges of the jaws run parallel with direction of the stroke, by loosening lock bolts on swivel base and adjusting according to index on swivel base.

3. Set work in vise with best surface, 1 on Fig. 1, against solid jaw. Slip pieces of cardboard or thin metal between work and vise jaws to prevent work from injuring the jaws. If the work does not project above the jaws, slip parallel bars of

suitable thickness under the work. The work should project up far enough to prevent the cutting tool from striking the vise. Tighten the vise jaws.

4. File off the corner at end of stroke to a depth equal to, or slightly more than depth of cut, to prevent corner from breaking off while machining. File at an angle of about 20° with the surface being planed.

Note: The following letters, and numbers in parentheses, refer to wall chart on the shaper.

5. Set the head (D') in vertical position by loosening clamping bolt (14), and adjusting by use of graduated swivel head plate.

6. Insert tool in tool holder. "Catch short" to prevent springing. Insert tool holder in tool post. Set well back.

7. Turn the down feed (11) as far back as possible.

8. Loosen clamping screws (19) and raise table with screw (18). Use removable crank handle. Work should be about 1/4" from the cutting tool. Tighten clamping screws (19).

9. Set the length of stroke to about 3/4" longer than length of stock to be machined. This is done by loosening knob (5) and turning shaft (4) by means of removable handle. The length is determined by means of index plate and pointer (3). After the correct length is obtained, lock by turning knob (5).

10. Set the position of the ram so that the extreme end of the cutting path of the tool extends about 1/2" past the work on the starting side and about 1/4" past the work on the finish side. To do this, loosen clamp lever (1). Next bring the ram to its extreme forward position by turning the driving pulley or hand wheel (32) by hand. Then by turning handle (2) adjust the cutting tool to about 1/4" past the end of the stroke. Lock lever (1).

11. Set the table over so that cutting tool projects about 1/8" over right side of table by putting crank handle on feed screw (43).

Operation MS-10

12. Adjust the cutting tool for depth of cut desired by hand feed screw (11). See Information Sheet MS-8.

13. Set the automatic cross feed mechanism to the desired feed by adjusting knob (39). See Information Sheet MS-8).

14. Determine the speed best suited to this job. See Information Sheet MS-8.

15. Set the shaper to the desired speed. See Information Sheet MS-9.

16. Have the instructor or foreman check the set-up to make sure everything is O.K.

Note: Be sure you know how to stop the shaper quickly in case of accident.

17. Start the shaper by pushing the friction clutch engaging lever (29) to the right, and rough plane the first side. See Fig. 1, side 1.

18. Stop the shaper by moving the friction clutch lever to the left.

19. After side 1 is rough planed, loosen the vise jaw and turn side 2 up. Tighten vise slightly and tap (not pound), on the work with a lead hammer to make sure that it rests snugly on vise bed or parallel bars. Tighten vise. File bevel on end as before.

20. Adjust the cutting tool to give the proper thickness of stock. Allow about 1/64" for the finishing cuts. Bevel end as before.

21. Rough plane side 2 in same manner as 1.

22. Loosen vise, remove cardboard or metal plates, and set side 1 against solid jaw of vise. Tighten vise.

23. Rough plane side 3.

24. Loosen vise and turn side 4 up. Tighten jaws slightly and tap work with a lead hammer as in step 19. Tighten jaws.

25. Adjust the cutting tool to leave the proper width of stock. Allow 1/64" for the finishing cuts. Bevel ends.

26. Rough plane side 4.

Note: There are two methods of planing the ends; one method if the stock is short, and another method if the stock is long.

27. If the stock is short, set it on end as indicated in Fig. 2. Hold the square down firmly on the parallel blocks or bed of vise, and the piece of work firmly against the blade of the square. Tighten the vise lightly. Check the setting, tapping the work lightly one way or the other as necessary to square it, and tighten vise.

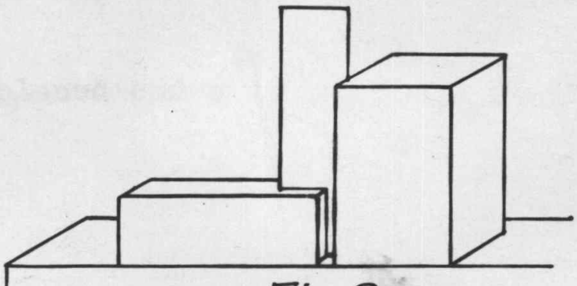


Fig. 2.

28. Rough plane end 5.

29. Loosen vise and set planed end on bed of vise. Set the tool to the proper height and plane end 6.

30. If the stock is too long to stand on end, clamp it in the vise so that one end projects a short distance to the right. Set a left-hand, down-cutting tool as indicated in Fig. 3. Set the top of the apron at an angle of about 30° in a direction away from the surface of the cut to be taken.

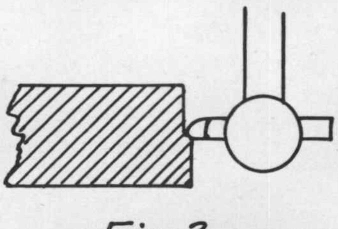


Fig. 3.

31. Engage the automatic down feed by paul knob (46). See wall chart. Regulate feed by feed regulating knob (47).

32. Adjust the table and tool head to take desired cut.

33. Take roughing cut.

34. Reverse ends. Measure for length, allowing for

finish cuts, and complete the roughing cut.

35. To finish the block, repeat finishing cuts in same order as for roughing. No cardboards or metal strips should be used. The vise jaws and work must be clean and the work carefully fitted. The depth of cut should be about 1/128". A special finishing tool must be secured for this purpose.

QUESTIONS:

1. Why are cardboard or metal strips placed between vise jaws and rough surfaces?

2. Why is the clapper box made so that it can be swiveled at the head?

3. How is a comparatively narrow, irregular surface planed?

4. How can you test the solid vise jaw?

5. What would happen if you did not cut below the scale on cast iron?

REFERENCE:

1. Burghardt, H. D. - Machine Tool Operation, pp. 65-115.

Operation MS-11

CUTTING INSIDE THREADS BY HAND

INFORMATION:

Inside threads are usually cut in a machine of some sort. It is, however, sometimes necessary to cut new threads or repair old ones by hand. A machine may not be available or the hole may be so located that it is difficult to cut the threads with a machine.

PROCEDURE:

1. Select a drill of proper size and drill the hole for the tap to follow. The following table gives drill sizes for U. S. standard threads. These are the most commonly used. For other thread forms see Machinery's Handbook, p. 935.

Tap Drill Sizes for U.S. Standard Threads

Thread Diameter	Number of Threads	Diameter of Tap Drill
1/4"	20	3/16"
5/16	18	15/64
3/8	16	19/64
7/16	14	11/32
1/2	12	25/64
9/16	12	29/64
5/8	11	1/2
11/16	11	9/16
3/4	10	39/64
13/16	10	43/64
7/8	9	23/32
15/16	9	25/32
1	8	53/64
1-1/16	8	57/64
1-1/8	7	15/16
1-3/16	7	1
1-1/4	7	1-1/16

2. Select a taper tap of desired size and form of thread, and fasten it to the adjustable tap wrench.

3. Turn the tap into a bushing of corresponding size and form of thread. It should be turned through

until the taper part projects as shown in Fig. 1.

Note: Experienced workmen never use a bushing, but it will help the beginner to start the tap and keep it straight. In case a taped bushing is not available, use a finished machine nut of the proper size.

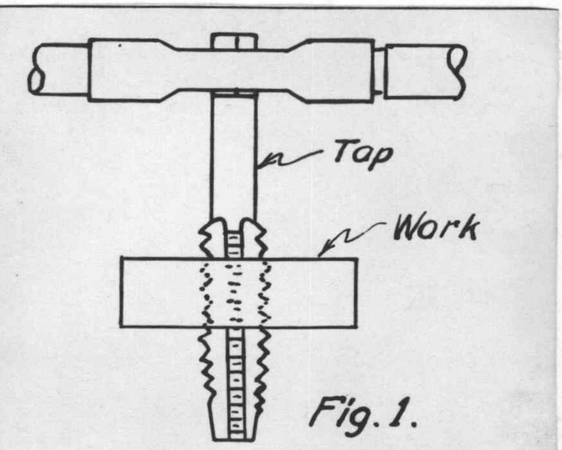


Fig. 1.

4. If the work is wrought iron or steel, place a few drops of cutting oil in the hole to be threaded. Cast iron is usually tapped dry. Sometimes a few drops of oil is placed on the threads already cut to prevent rubbing. On copper, use a mixture of turpentine and lard oil, and on aluminum use a mixture of kerosene and lard oil.

5. Set the tap in the hole to be threaded and turn the bushing down until it rests evenly on the work, touching on all sides.

6. Hold the bushing firmly against the work with one hand and turn the tap with the other. At the same time exert a downward pressure to prevent the tap from slipping and reaming out the hole. When the tap is well started it will feed itself.

7. Turn the tap backward occasionally about one-half a turn. This will break the chip and prevent binding.

   In tapping soft metal, such as copper or babbit, it is necessary to remove the tap occasionally and to clean out the chip.

Note: Care must be taken not to break the tap. It is very hard and brittle, and the smaller sizes break very easily. If the tap breaks, do not try to drill it out. See Operation Sheet MS-13.

8. If the hole goes all the way through the stock, the taper tap is usually the only one needed. If the hole is "blind"; that is, if it does not go all the way through, the taper tap is followed by the plug tap, and the plug tap is followed by the bottoming tap. The taper tap is used first because it starts more easily. The plug and bottoming taps are used to cut threads to the bottom of blind holes.

QUESTIONS:

1. What is apt to be the result if the hole to be tapped is too small?

2. What would happen if you attempted to tap a hole in a piece of work that had been hardened?

3. How deep is it necessary to tap a hole in order to give the threads sufficient strength to prevent stripping?

REFERENCE:

Information Sheet MS-10.

TAPS AND TAPPING

INFORMATION:

Taps are tools used to cut inside threads. They may be used either by hand or in a machine. The drill press with a tapping attachment, and a tapping machine, are two machines commonly used for this purpose.

Taps vary in size and form to correspond with standard threads of various sizes and forms. The size of the tap and form of thread is stamped on the shank.

Taps for a given size thread usually come in sets of three. They are termed taper, plug, and bottoming taps, Fig. 1. The taper tap is chamfered back from the end at least six threads. The plug tap is chamfered back about three or four threads. The bottoming tap is chamfered on the end teeth only.

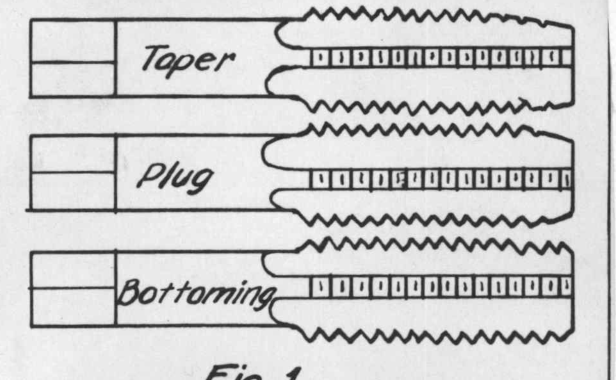


Fig. 1.

The purpose of the chamfering is to make it easier to start the tap. It is almost impossible to start a tap with full sized teeth at the end.

If the hole goes through the work to be tapped, and if the full teeth on the taper tap project through when it is turned down, the other two taps need not be used.

If the hole does not go through the work to be tapped the taper tap is used first to start the threads. It is followed by the plug and bottoming taps. They cut the threads near the bottom of the hole that can not be reached by the taper tap.

Tap Drill Sizes

The diameter of the hole to be drilled for the threads in a nut or any inside threaded piece is theoretically the root diameter, Fig. 1, of the corresponding screw size. In actual practice it is not practical to tap a full thread, as it takes too much power and causes a great increase in the percentage of broken taps. An ordinary nut drilled so that it has only half of a full depth of thread will break the bolt before it will strip the threads. A two-thirds depth of thread will give a margin of safety of about 2 to 1. A two-thirds depth of thread requires only one-third the power for tapping that is required to tap a full thread.

The thickness of the nut or threaded part is a factor in determining the size of tap the drill. In general, if the screw enters more than one and one-half times its diameter, one-half of the full thread is sufficient.

Hard, crystalline metals such as cast steel require a larger factor of safety than do soft, tough metals such as Norway iron. The Machinery's Handbook gives the following formula for determining tap drill size for U. S. Standard and V threads.

$$D = T - 0.75 \times 2d$$

D = drill diameter
T = diameter of tap or thread
d = depth of thread

The values of the factor "2d" for various numbers of threads can be obtained from taps on screw thread systems or it can be taken on the tap by using a thread micrometer. See Machinery's Handbook, p. 1181.

Machinery Handbooks and many texts on machine shop work have tables giving the tap drill sizes for various shapes and forms of threads.

QUESTIONS:

1. What is apt to happen if the hole drilled for tapping is too small?

2. What must be watched out for in refinishing old threaded holes?

3. What is meant by "full thread depth"? "Half thread depth"?

Operation MS-13

TO REMOVE A BROKEN TAP OR DRILL FROM THE WORK

INFORMATION:

Taps and drills are very hard and brittle. The smaller sizes break very easily. Many beginners try to drill out the broken parts. This always results in either another broken drill or a badly damaged cutting edge.

There are several ways of removing these broken parts. The method selected depends on where it is broken and how hard it is to loosen. Most drills and taps are broken near the surface of the work.

One of the following three methods will usually be effective:

A. <u>When the broken part projects above the work</u>.

1. Grasp the projecting part securely with a wrench or pliers.

2. Jar it loose by carefully working the wrench or pliers back and forth.

Note: A few drops of nitric acid diluted with water in the ratio of one acid to five water, will often loosen the tap or drill so that it can be removed more easily. After the broken part is removed, wash out the hole to prevent further action of the acid.

3. Turn the broken part backwards until it is removed.

B. <u>When broken part is flush with, or slightly below the surface</u>.

1. If the work is small, fasten it in a vise so that the broken part is easy to get at.

2. Secure two blunt cape chisels or other suitable pieces of metal.

3. Get somebody to help you and drive from

both sides at the same time, holding the cape chisel as indicated in Fig. 1.

Note: Using one chisel will cause the broken part to imbed itself in the side opposite the chisel.

4. If the broken part will not come loose, use nitric acid as explained in section "A".

Fig. 1.

C. <u>When the broken part is deeply imbedded</u>.

If broken parts cannot be reached as in "A" or "B", and the work is of such a nature that it can be heated without spoiling it, proceed as follows:

1. Heat the work to a dull red.

2. Cover with ashes, powdered charcoal, lime or asbestos, and allow it to cool slowly until no red shows when in a dark place.

3. Plunge into cold water.

4. Drill out broken part.

QUESTIONS:

1. What are the principal causes of broken taps?

2. Why can the otherwise hard tap or drill be drilled out after heating the work as in "C"?

Operation MS-14

TO MILL A KEYWAY IN A SHAFT

INFORMATION:

Pulleys, gears, hand wheels, and levers of various kinds are often keyed to the shafts on which they rotate. This prevents them from slipping when heavy loads are applied.

PROCEDURE:

1. Set the vise on the milling machine table so that the jaws are at right angles to the axis of the arbor. See wall chart, part (1).

Note: Numbers in parentheses in body of instruction refer to parts numbered on wall chart.

2. Clean arbor and spindle (2) and drive arbor into spindle by tapping lightly with a lead hammer.

3. Select a cutter of the desired width of keyway.

4. Set the cutter to the desired position on the arbor. This position can be varied by using collars of different lengths as indicated in Fig. 1. It should be as near the spindle as the work will permit.

Note: See that the cutter is set to rotate in the right direction. Cutter should be above the work and the front edge of the teeth rotate toward the work as it moves under the cutter. Fig. 2.

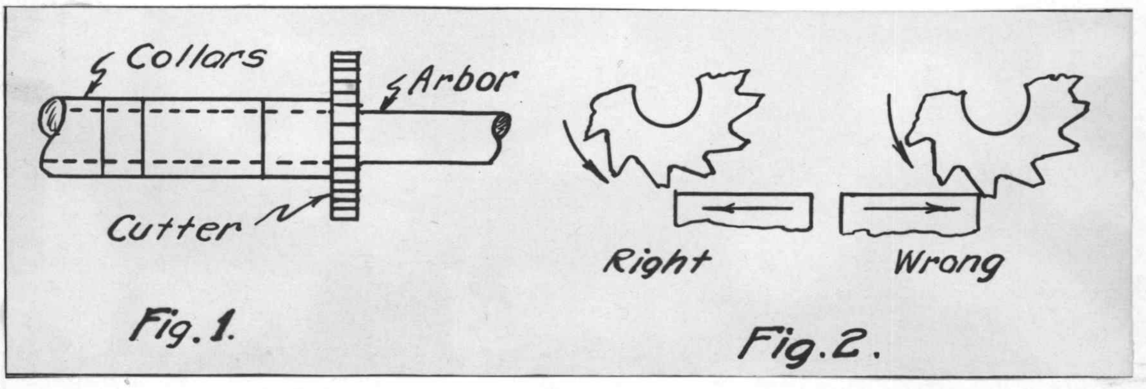


Fig. 1.   Fig. 2.

5. Fill in collars on arbor until they extend a short distance past the inner end of threads

on end of arbor.

6. Thread nut on end of arbor and tighten hard.

7. Oil end of arbor and adjust outer bearing support (37) to hold it firm.

8. Tighten bolts on arm and bearing support (35).

9. Set the cutter central over the shaft by raising the table and moving the cross feed until distance "A", Fig. 3, from the cutter to the fixed jaw "B", equals one-half the diameter of the shaft, less one-half the width of the cutter.

Example:

Shaft = $1\frac{1}{2}$" D. One-half of $1\frac{1}{2}$" = $\frac{3}{4}$"

Cutter = $\frac{1}{4}$" " " $\frac{1}{4}$" = 1/8"

$\frac{3}{4}$" - 1/8" = 5/8" for distance A.

10. Place shaft in vise. Tighten slightly. Tap down with lead hammer. Tighten hard. Shaft should then be exactly centered under cutter.

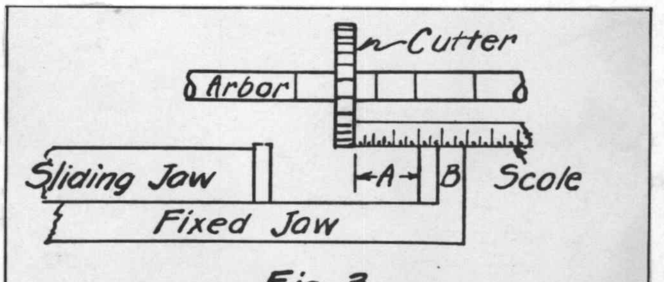


Fig. 3.

Note: If shaft rests too low in vise, use parallel bars to raise it.

11. Draw line for length of keyway.

12. Set the machine to the desired spindle speed. See Information Sheet MS-15.

13. Set the machine to the desired feed. See Information Sheet MS-15.

14. Put a few drops of cutting oil on the work.

15. Start the machine by moving friction clutch lever (2).

Note: Be sure you can stop machine quickly in case of accident.

16. Take a few trial cuts with hand feeds until cutter just exactly mills its own width, leaving only a flattened spot on the shaft without a groove. See Fig. 4.

17. Move work away from cutter. Set vertical dial at zero and raise the table the required number of thousandths to get desired depth of cut.

18. Feed work to cutter by hand feed, then throw in power feed (14) and mill keyway with one cut. When nearing line trip power feed by hand and feed to line by hand feed.

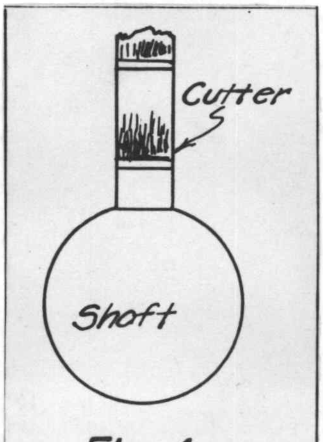


Fig. 4.

QUESTIONS:

1. What are some other methods of setting the cutter central?

2. Why does the speed box table give spindle speeds instead of F.P.M.?

3. What would happen if the cutter was put in "wrong side to"?

REFERENCES:

Burghardt, H. D. - Machine Tool Operation, p. 236.

Palmateer, T. J. - Machine Shop Practice, p. 97.

Information MS-15

MILLING SPEEDS AND FEEDS

GENERAL INFORMATION:

Speeds:

This is a variable quantity depending on the hardness of the material to be cut, depth of cut, width of cutter, stability of the machine, nature of work, and the kind of steel the cutter is made of.

Tables of cutting speeds are given in terms of F.P.M. or the number of feet per minute that the cutting edges of the cutter travel across the work.

The settings of the speed box on the milling machine are given in terms of spindle speeds. This makes it necessary to convert the F.P.M. of the tables to the terms of spindle speeds on speed box. By using the following formulas taken from Machinery's Handbook, it is possible to set the speed box directly on the average speeds required for the four most commonly used metals.

Spindle Speeds for Carbon Cutters

Cast Iron	Machine Steel	Tool Steel	Brass
$N = \frac{150}{D}$	$N = \frac{230}{D}$	$N = \frac{95}{D}$	$N = \frac{285}{D}$

N = revolutions per min.
D = diameter of cutter.

For high speed cutters, these speeds should be doubled.

Machinery's Handbook gives the speeds and feeds for a large variety of typical conditions, p. 866-868. Studying these pages carefully will give the beginner much valuable information.

Speed table (11) on wall chart gives spindle speeds, and position of the three levers, A,B,C, to obtain them. To obtain a given speed, unlatch lever "A" and lower it. Move index lever "B" to the required column under the speed table, then raise lever "A" to engage proper gears and latch it. Lever "C" throws back-gears in or out as required for a given setting.

Feeds:

Theoretically, the feed of a milling cutter is the thickness of the chip per tooth of the cutter. In "cone-driven" machines the amount of feed is dependent upon the spindle speed, and is usually rated as the amount the table moves per revolution of the spindle. In milling machines having a constant speed drive, the feed mechanism is usually independent of the number of revolutions of the cutter. In these machines, the feeds are arranged to move the table a certain distance per minute.

The proper rate of feed depends on the design of the cutter, the width and depth of cut, kind of material being milled, quality of finish desired, the rigidity of the work, the power of the machine, etc. As a general rule, a relatively low cutting speed and a heavy feed is used for roughing. For finishing, the speed is increased and the feed is diminished. It is seldom necessary to take more than two cuts. About 1/64" should be left for finishing.

Cutters are more often ruined from too high a cutting speed, rather than too coarse a feed.

Conditions vary so much that it is not possible to formulate tables for cutting feeds, but the typical examples of a large variety of conditions illustrated on pp. 866-868 of Machinery's Handbook will serve as guides for the beginner in setting feeds.

The wall chart feed table (13) gives the position of three levers D,E,F, to obtain twelve feeds. To obtain a given feed, unlatch lever "D" and raise it. Move lever "E" to required column under feed table, then lower lever "D" to engage gears, and latch. Lever "F" gives fast and slow motion.

QUESTIONS:

1. What speed and feed would be best suited for milling off the surface of a casting four inches wide, the cut being 3/16" deep?

2. What speed and feed would be best suited for milling a slot 1/2" wide and 1/2" deep on machine steel?

3. What advantages does the constant speed drive milling machine have over the cone pulley drive?

SECTION IV.

CONCLUSIONS

CONCLUSIONS

1. This method of presenting the work by means of individual instruction sheets should provide for individual help to each student so that he can progress in the shop at his own rate of speed.
2. It should enable the instructor to handle larger classes in a more efficient manner.
3. It places initiative upon the student rather than upon the instructor.
4. It gives the instructor more time for individual help where needed.
5. The trade analysis gives the instructor a bird's-eye-view of the total content of the trade. This should enable him to select the things he wishes to teach in a more scientific manner.
6. The operation sheets, in order to be effective, must be written in language easily understood. The easiest way is for the instructor to write down the things that he would say in a carefully planned demonstration and then to organize the content and arrange it in logical order for presentation in written form. The sheets should be profusely illustrated by drawings and photographs of the parts described.

7. When properly correlated with the personal efforts of the instructor, and when presented in a simple and logical manner, the instruction sheet plan will increase the effectiveness of instruction, especially with larger classes.

SECTION V.

BIBLIOGRAPHY

BIBLIOGRAPHY

1. Allen, Chas R. The Instructor, The Man, and the Job. J. B. Lippincott Company, Philadelphia, 1919. P. 3-114.

2. Bowman, Clyde A. Graphic Aids in Occupational Analysis. Bruce Publishing Company, Milwaukee, Wis., 1924.

3. Burghardt, Henry D. Machine Tool Operation, Parts I and II. McGraw-Hill Book Co., Inc., New York, 1919.

4. Cushman, Frank. Mathematics and the Machinist's Job. John Wiley & Son, New York, 1926.

5. Eaton, Joseph J. Machine Shop Science. Manual Arts Press, Peoria, Illinois, 1926.

6. Jones, Harry A. Machine Shop Practice, Book 1. Bruce Publishing Company, Milwaukee, Wis., 1926.

7. Oberg, Eric & Jones, Franklin D. Machinery's Handbook. The Industrial Press, New York, 1928.

8. Palmateer, T. J. Elementary Machine Shop Practice. The Manual Arts Press, Peoria, Ill., 1920.

9. Selvidge, R. W. How to Teach a Trade. The Manual Arts Press, Peoria, Ill., 1923.

10. Selvidge R. W. Individual Instruction Sheets. The Manual Arts Press, Peoria, Ill., 1926.

11. Smith, Robert H. Advanced Machine Work. Industrial Education Book Company, Boston, Mass., 1925.

12. South Bend Lathe Works, How to Run a Lathe. South Bend, Indiana, 1924.

13. Wentworth, George. Machine-Shop Mathematics. Ginn and Company, Boston, Mass., 1922.