

OPERATION AND INSTRUCTION SHEETS FOR THE  
ELEMENTARY PRINCIPLES OF MACHINE SHOP PRACTICE

by

EARL GILBERT DARBY

P. S., Kansas State College  
of Agriculture and Applied Science, 1923

-----

A THESIS

submitted in partial fulfillment of the  
requirements for the degree of

MASTER OF SCIENCE

Department of Education

KANSAS STATE COLLEGE  
OF AGRICULTURE AND APPLIED SCIENCE

1943

9/12/1943

Docu-  
ment  
LD  
2668  
T4  
1943  
D3  
C.2

TABLE OF CONTENTS

	Page
INTRODUCTION -----	1
METHOD -----	3
OPERATION AND INSTRUCTION SHEETS FOR MACHINE TOOL	
WORK SERIES -----	4
OPERATION SHEET FOR TEST SPECIMEN -----	5
INSTRUCTION SHEETS FOR TEST SPECIMEN -----	8
Finding Centers of Stock -----	8
Center Drilling Stock -----	10
Mounting Work Between Centers -----	12
Facing or Squaring Work Between Centers -----	14
Grinding Cutting Tools -----	16
Straight Turning Between Centers -----	19
Checking and Adjusting the Lathe to Eliminate	
Taper -----	21
Setting Lathe RPM For Proper Cutting Speed -----	23
Laying Off Distances on Work -----	26
Turning a Filleted Shoulder -----	28
Filing and Polishing on the Lathe -----	29
Adjusting Lathe to Cut Required Number of Threads	32
Set and Adjust an External Thread Cutting Tool ---	34
Check Setting of Lathe for Cutting Threads -----	36
Chamfering Stock Previous to Threading -----	37
Thread Cutting in General -----	38

Cutting Threads on Lathe Not Equipped with Threading Dial -----	40
Cutting Threads on Lathe Equipped with Threading Dial -----	42
OPERATION SHEET FOR WOOD LATHE TOOL REST -----	44
INSTRUCTION SHEETS FOR WOOD LATHE TOOL REST -----	46
Turning Cast Iron -----	46
Rough Turning Cast Iron Cylinder the Entire Length -----	48
Finish Turning a Cylinder to Fit Gage -----	50
Turning to a Square Shoulder -----	51
Chamfering an Edge -----	53
OPERATION SHEET FOR TAILSTOCK HANDWHEEL -----	55
INSTRUCTION SHEETS FOR TAILSTOCK HANDWHEEL -----	57
3-jaw and 4-jaw Chucks -----	57
Removing and Mounting Face Plates and Chucks -----	58
Truing Work in a Chuck -----	60
Facing Work in a Chuck -----	62
Spotting the Center and Drilling on the Lathe -----	63
Boring on the Lathe -----	65
Hand Reaming on the Lathe -----	67
Rough Forming on the Lathe -----	68
Finish Forming on the Lathe -----	69
OPERATION SHEET FOR TAILSTOCK CLAMPING SCREW -----	70
INSTRUCTION SHEETS FOR TAILSTOCK CLAMPING SCREW -----	72
Taper Turning in General -----	72

Calculation and Adjustment of Offset of the Tailstock for Taper Turning -----	74
Contour Forming -----	77
OPERATION SHEET FOR WOOD LATHE FACE PLATE -----	79
INSTRUCTION SHEETS FOR WOOD LATHE FACE PLATE -----	82
Counterboring on the Lathe -----	82
Setting Internal Threading Tool and Cutting Threads -----	84
OPERATION SHEET FOR TAILSTOCK SPINDLE SCREW -----	86
INSTRUCTION SHEETS FOR TAILSTOCK SPINDLE SCREW -----	88
Turning a Neck or Groove -----	88
Cutting Left-Hand Threads -----	90
OPERATION SHEET FOR WOOD LATHE LIVE CENTER BLANK -----	91
INSTRUCTION SHEET FOR WOOD LATHE LIVE CENTER BLANK -----	93
Checking a Taper With a Taper Gage -----	93
OPERATION SHEET FOR A V DRILL BLOCK -----	95
INSTRUCTION SHEETS FOR A V DRILL BLOCK -----	97
The Shaper in General -----	97
Clamping Stock in Shaper Vise -----	98
Adjusting Stroke and Positioning Shaper Ram -----	100
Tool Positions for Various Shaper Cuts -----	101
Arranging Feed for Various Shaper Cuts -----	103
SUMMARY -----	105
ACKNOWLEDGMENT -----	106
BIBLIOGRAPHY -----	107

## INTRODUCTION

The use of instruction sheets as a teaching device has been increasing in all branches of education. Such sheets are particularly adapted to shop courses where much good work has been done on the high school level. It was with a view of facilitating the teaching of the elementary principles of machine shop practice on the college level that this study and preparation of operation and instruction sheets were made.

The purpose is not to replace the personal instruction of the teacher but to supplement it. The pupil still has his opportunity to plan his work, and to realize fully his capacities. Williams (22, p. 86) says that instruction sheets furnish printed directions to be followed. Success in a great number of occupations depends upon the ability to understand and follow directions set forth in this way; consequently, the practice is of high importance.

According to Fryklund (7, p. 241), instruction sheets give the student a chance to regulate his speed in comprehension to suit his capacities. Students need training in reading habits.

By using instruction sheets the individual pupil may start a new piece of work when he is ready and complete it as soon as he can. He is not held back by the slower pupils in the class as he is when the work is conducted entirely by oral instruction. Van Westrienen (21, p. 237) makes the comparison clearly as follows:

The traditional shop course was formerly introduced to students by a long series of demonstrations covering every process and operation that will be required during the course. Active students resent having to listen when they might better be digging much of it out for themselves at the time they need it. This situation can be solved by means of adequate operation and instruction sheets.

When instruction sheets embody specific and brief directions on doing work, and furnish related information needed to make the performance of the work intelligent, they go far toward insuring economy of time and individual progress. Distinct instructional units, treated separately, may be used at pleasure, individually or in combination with other like units, and are readily adaptable to any wisely chosen series or group of jobs or projects used in shop practice. Written instructions usually represent more careful preparation than oral instruction; the brief and definite directions are carefully worded so that they may be understood readily and followed definitely and accurately. The instructor who prepares his teaching material in such form has adequate time to devote to situations which arise, and each pupil has ready-to-use instruction without delays of group or class instruction.

In the preparation of the following instruction sheets the purpose has been to furnish the means for raising the quality of class room instruction in the essential elementary principles of machine shop practice on the college level. Phillips (16, p. 321) outlines adequately the objectives of instruction sheets. He says properly prepared and used, they will:

1. Make instruction clear, specific and accurate.
2. Give information needed to supplement directions on how to do.
3. Economize the time of the pupil and the instructor.
4. Promote individual pupil progress.
5. Give flexibility to courses of instruction.

The course in machine shop practice as it now exists at this college consists of a series of exercises planned to cover essential operations in connection with the actual production job of making wood turning lathes. The objectives of this course as applied to engineering students are: (1) a general knowledge of common machines used in commercial production; (2) development of skill and precision used in factory mass production; (3) economical techniques in machine production; (4) knowledge and skill in selecting materials; (5) development of effective techniques in shop personnel management.

#### METHOD

The instruction sheets in this study were prepared from the elementary principles of machine shop practice which experience has shown to be essential, and from various textbooks, pamphlets, and directions published by different machine tool companies.

OPERATION AND INSTRUCTION SHEETS  
FOR MACHINE TOOL WORK SERIES

OPERATION SHEET FOR TEST SPECIMEN  
(Blue Print P-155)

Objectives:

1. Find centers and center drill ends of stock.
2. Mount and drive work between centers.
3. Square ends of stock to required length.
4. Rough turn a cylinder.
5. Check and adjust tailstock to eliminate taper.
6. Finish turn a cylinder to size.
7. Turn a filleted shoulder.
8. File and polish on the lathe.
9. Chamfer stock on the lathe.
10. Cut external threads on the lathe.

Tools and Equipment:

Lathe, hammer, bench plate, V block, surface gage, center punch, rule, calipers, micrometer, test nut, threading tool, combination drill, turning tool, fillet turning tool, file, abrasive cloth.

Operations:

1. Select stock.
2. Locate center of ends. I.S. 1.\*
3. Center drill ends. I.S. 2.
4. Attach dog and mount work on centers. I.S. 3.
5. Square ends, obtaining proper length. I.S. 4.
6. Re-drill center holes to proper size. I.S. 2.
7. Grind cutting tools. I.S. 5.
8. Straight turn between centers to obtain cylinder. I.S. 6.
9. Check and adjust for taper elimination. I.S. 7.
10. Set lathe RPM for proper turning speed. I.S. 8.
11. Rough turn entire cylinder. I.S. 6.
12. Finish turn entire cylinder to size. I.S. 6.
13. Lay off distances for threaded ends. I.S. 9.  
Note. Leave about  $1/32$  to  $1/16$  of an inch of material beyond the one-inch marks to assure having material for the fillets or  $1/8$ -inch radii.
14. Rough turn entire center section. I.S. 6.
15. Finish turn entire center section. I.S. 6.  
Note. Finish turn to about .002 of an inch oversize for filing and polishing.
16. Turn both fillets. I.S. 10.
17. File if necessary and polish center section. I.S. 11.

---

\* Instruction Sheet 1.

18. Adjust lathe to cut required number of threads. I.S. 12.
19. Set and adjust external threading tool. I.S. 13.
20. Check lathe setting for required number of threads per inch. I.S. 14.
21. Chamfer first end. I.S. 15.
22. Cut threads on first end to fit test nut. I.S. 16, 17, 18.
23. Using split-nut place driving dog on threaded end.
24. Chamfer second end. I.S. 15.
25. Cut threads on second end to fit split-nut. I.S. 16, 17, 18.
26. Have instructor check work.



Fig. 1. Test specimen.

## INSTRUCTION SHEETS FOR TEST SPECIMEN

## Finding Centers of Stock

## (Instruction Sheet 1)

After receiving stock, measure to see if it will finish to the required length. On a heavy grinder, remove the burrs and rough edges from the ends of stock. Chalk the ends so that the center locating lines will show distinctly.

The method used in locating the centers of the ends will be determined by available equipment. There are several good methods for locating centers: (1) hermaphrodite-caliper, (2) center-square, (3) surface-gage, and (4) divider.

In using the hermaphrodite-caliper (Fig. 2), set the caliper to about the radius of the stock. Place the caliper leg on the circumference at the extreme end of the piece and with the point draw an arc near the center of the end. Move the caliper leg about one-fourth of the circumference of the end each time and draw three more arcs. The four arcs will form an approximate square, the center of which is the required center.

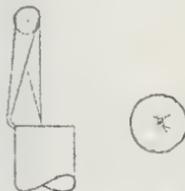


Fig. 2. Finding center of stock.

The center-square, when placed in contact with a cylinder (Fig. 3) gives two points of contact, a and b. If



Fig. 3. Finding center of stock.

these two points are joined, the line from a to b is a chord of a circle. If the chord is bisected at right angles, the line will pass through the center of the circle, so that the two intersecting lines will locate the center.

The surface-gage method (Fig. 4) requires a bench plate, surface-gage, and V block. Place the stock in the V block and set the height of the scriber point at the approximate center of the stock. Scribe lines with the stock in two or more positions. The intersection of scribed lines locates the center.

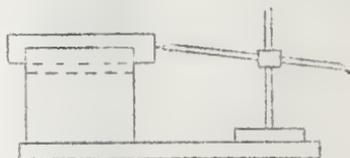


Fig. 4. Finding center of stock.

In using dividers, set them to approximately one-half the diameter of the stock and scribe four lines (Fig. 5). The center of the square locates the center of the stock.

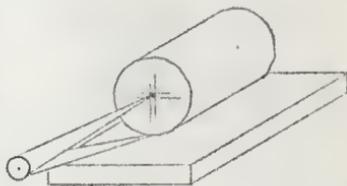


Fig. 5. Finding center of stock.

Center Drilling Stock  
(Instruction Sheet 2)

Before the stock may be center-drilled it is necessary to make a small indentation at the point where the drill is to enter the stock. Clamp the stock in a vise and select a center punch and hammer. Place the punch vertical to end of the stock and tap lightly with a hammer (Fig. 6). Note if the indentation is at the previously located center of the stock. If it is not, lean the punch slightly away from the direction in which the indentation is to move (Fig. 7). Tapping lightly with a hammer will move the point to the desired place. If the center punch is at the center of the stock, hold it vertical and strike it hard enough to make a center for guiding the drill.

There are various methods of center drilling but probably the most common one is to use a combination drill and counter-sink in the drill press. Clamp the stock in the drill press vise and insert the drill in the drill chuck. Drill the center hole to the desired



Fig. 6. Center punching stock.



Fig. 7. Shifting center point.

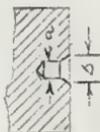


Fig. 8. Center correctly drilled.

size which is dependent upon the size of the stock to be turned. Figure 8 shows a hole properly drilled. The diameter of the hole at  $b$  is approximately one and one-half times the diameter at  $a$ .

Figure 9 shows a shallow hole which causes the stock to turn on the point of the dead center, thus producing undue wear.

A countersunk hole which is not large enough to give sufficient contact with the dead center is illustrated in Fig. 10.

A hole much too large to permit a finished piece of work is shown in Fig. 11.

Figure 12 shows the result of carelessness which is inexcusable.



Fig. 9. Hole too shallow.



Fig. 10. Countersink too small.



Fig. 11. Hole too large.

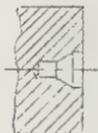


Fig. 12. An inexcusable hole.

## Mounting Work Between Centers

### (Instruction Sheet 3)

Work that is to be turned accurately and concentrically between lathe centers must be mounted correctly in order to provide a good bearing and support for the center holes of the job.

Some of the important factors to be considered in mounting a job between centers are: alignment of lathe centers, trueness of live center, condition of center holes, lubrication of tailstock center, adjustment of work between centers, and adjustment of lathe dog in regard to work and dog driver plate.

To check approximate alignment of lathe centers, slide the tailstock forward until the two centers are nearly touching, as shown in Fig. 13. Sight down over the centers and correct any error in alignment by adjusting the tailstock set-over screws (Fig. 14).

The lathe dog should be attached to the work with the bent tail pointing away from the job and in position to be driven by the driver plate.

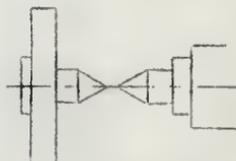


Fig. 13. Checking alignment of centers.

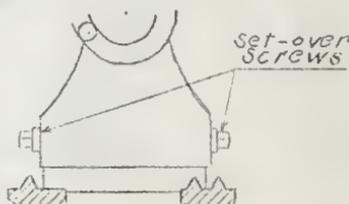


Fig. 14. Adjustment of tailstock.

Before mounting the work on centers, place a drop of oil in the center hole for the tailstock center. The tail of the lathe dog should fit freely into the slot of the driver plate so that the work rests firmly on both the headstock and tailstock centers, as illustrated in Fig. 15. Make sure that the tail of the dog does not bind in the slot of the driver plate.

When the work is supported on centers, adjust the tailstock center so that no play can be felt between the stock and the centers. At the same time, the work should be so free that the weight of dog tail would be sufficient to cause it to drop when moved to one side of the slot.

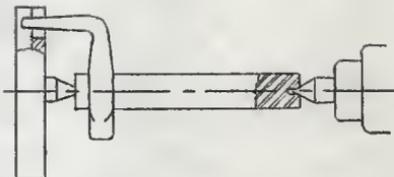


Fig. 15. Mounting work between centers.

Facing or Squaring Work Between Centers  
(Instruction Sheet 4)

Facing or squaring on the lathe is the process of machining the ends of the work flat, smooth, at right angles to the axis of the job, and to the required length. After the work is centered, usually it is faced or squared to provide a surface from which linear measurements may be taken.

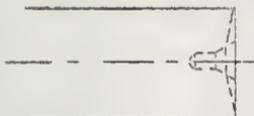


Fig. 16. Tailstock towards operator.

Before facing the work, it is necessary to adjust the tailstock so that the center is in alignment with the headstock center. If the tailstock center is off center towards the operator, the faced end of the stock will be concave, as shown in Fig. 16. If the tailstock center is off center away from the operator, the faced end will be convex (Fig. 17).

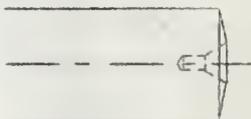


Fig. 17. Tailstock away from operator.

Most facing is done at the right end of the stock with a right-hand facing tool. An ordinary tool may be so ground as to be used for facing be-

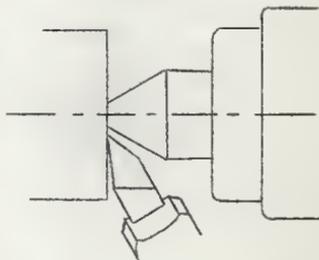


Fig. 18. Ordinary tool for facing.

tween centers. Such a tool is shown in Fig. 18. A special tool called a right facing tool may be used for the facing operation (Fig. 19).

In setting either tool for facing, the tool point must be set on the center of the work or on the tailstock center for height. The cutting edge of the tool is set against the end of the work at an angle slightly more than  $90^\circ$  to its axis to permit the point to do most of the cutting and to prevent the heel of the tool from rubbing

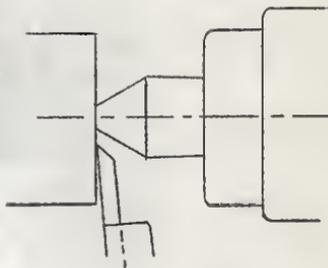


Fig. 19. Special tool for facing.

(Figs. 18 and 19). For a heavy roughing cut, the tool is generally fed in towards the center while for a light finishing cut it is generally fed out from the center. To assure a right angled face, the carriage should be locked in position by tightening the carriage clamping bolt and feeding the tool for the depth of the cut by means of the compound rest.

## Grinding Cutting Tools (Instruction Sheet 5)

A cutting tool carefully ground will stay sharp under correct working conditions for a considerable time, but as soon as it is noticeably dull it should be reground or the tool and possibly the work will be ruined. A dull tool tears rather than cuts the material, it springs the work, and makes a rough cut. To keep cutting tools sharp is a most important factor in efficient machine work.

The various clearance and rake angles must be correctly proportioned and ground to turn the chip properly and also preserve the cutting edge. Clearance angles which are excessive reduce the cross-section of the metal near the cutting edge so that the heat generated by the cutting cannot be dissipated rapidly. Figure 20 shows various clearance angles.

The round nose turning tool has clearance ground on both sides and on the front end. The front end is rounded and the top is either ground straight or with a right-hand or left-hand side rake, depending on the manner in which the tool is to be used.

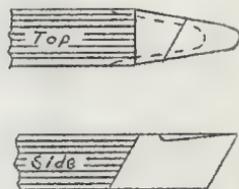


Fig. 20. Various clearance angles.

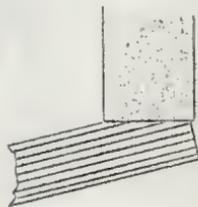


Fig. 21. Grinding side of tool.

When grinding a tool it should be held in the right hand and supported with the left. Press the left side of the tool against the face of the revolving wheel and move it back and forth across the face to grind the side to shape with side clearance (Fig. 21). Grind the right side of the tool in the same manner as the left.

Rough grind the shape of the rounded end. Then hold the tool lightly against the finer wheel and finish the end by swinging it from side to side, at the same time being careful to grind the required radius and front clearance (Figs. 22 and 23).

Hold the top of the tool flat against the wheel, as shown in Fig. 24. The top may also be ground at an angle to give side rake if it is desired.

To finish, hone the cutting edge of the tool by using a hand oil stone (Fig. 25). Honing improves the

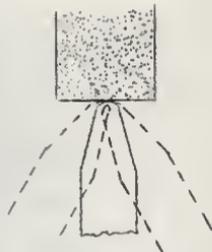


Fig. 22. Grinding end of tool.



Fig. 23. Grinding end of tool.



Fig. 24. Grinding rake on tool.

quality of the finish and lengthens  
the life of the tool.

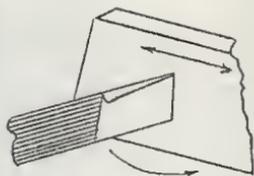


Fig. 25. Honing  
a tool.

## Straight Turning Between Centers

### (Instruction Sheet 6)

Straight or cylindrical turning on the lathe is the process of machining a revolving piece of work by feeding a cutting tool longitudinally along the piece, in order to produce sides that are parallel to the axis of the work.

Straight turning may be divided into two stages: rough and finish turning. Where much metal is to be removed, the roughing cuts should be as heavy as the machine, work, and tool will withstand. The surface of a rough turned job should not be so badly torn or roughly turned as to cause the keen edge of a finishing tool to dull quickly; nor be so heavy as to spring or bend the work or damage the lathe centers. The roughing cut is illustrated in Fig. 26. Ordinarily, the work is turned to within one-thirty-second of an inch of the finished size.

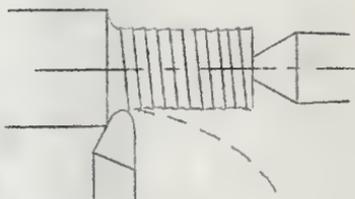


Fig. 26. Rough turning.

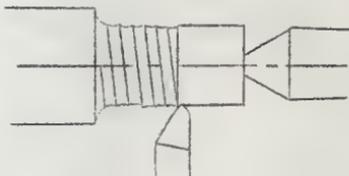


Fig. 27. Finish turning.

Finish cuts are made to turn the work round, smooth, and accurately to size. Light cuts are taken with a keen edged tool slightly rounded at the point. Finer feeds are used with increased spindle speeds. A finish cut is shown in Fig. 27.

Generally, one finish cut is all that is required. However, where the roughing cut has left a torn surface, two light cuts may be required to turn the surface smooth and accurately to size.

In starting straight turning between centers, select the proper turning tool and holder. Make sure that the tool is ground to the proper shape, rake, and clearance for either rough or finish turning. Set the

tool post at the left side of the T-slot in the compound rest. Adjust

the tool holder in the tool post so that the cutting tool is at right angles to the work or inclined slightly towards the tailstock to prevent its digging into the work when taking heavy cuts (Fig. 26). Set the cutting edge of the tool at the height of the center of the work, as shown in Fig. 28. Move the carriage along the ways to make sure that the tool can travel to the desired length without interference.

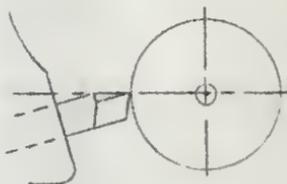


Fig. 28. Setting height of tool.

Checking and Adjusting the Lathe to Eliminate Taper  
(Instruction Sheet 7)

After taking the first cut across a shaft, check the diameter at each end of the cut with calipers or a micrometer to make sure that the lathe is turning straight. Sometimes when the position of the tailstock is changed for a different length of work, adjustments will be necessary. This is especially true on old lathes which may be worn or have burrs on the ways of the bed. The difference between the two diameters is the amount the tailstock should be adjusted.

Another method of checking for taper is to mount a piece of stock of approximately the same length as the work to be turned. Turn a portion of the shaft at A, Fig. 29, wide enough to allow the caliper or micrometer to measure the diameter. Do not disturb the setting of the tool but remove the



Fig. 29. Checking for taper.

work from between centers and move the carriage towards the tailstock to allow the work to be replaced on centers. Turn a portion of the shaft, as at B in Fig. 29. If the lathe centers

are in alignment, the diameter of the work at A and B will be the same.

An accurate method for checking the taper is to secure an alignment bar of approximately the same length as that of the work. Place the bar on the lathe just snug enough between centers to allow it to turn freely by hand. Clamp a tool holder in the tool post so that the butt of the holder may come in contact with the alignment bar. Move the tool holder to the alignment bar just snug enough to allow a piece of paper to slip between the bar and holder. Without changing the position of the holder, move the carriage to the other end of the bar and note if the paper slips between the bar and holder with the same ease as in the first case (Fig. 30). Adjust

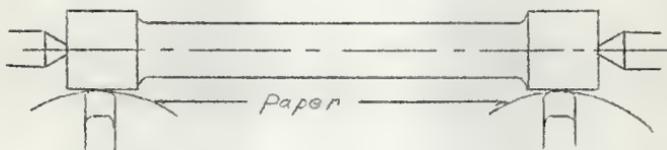


Fig. 30. Checking taper with alignment bar.

tailstock until the paper has the same degree of pull between the two ends of the alignment bar.

If it is necessary to make an adjustment, the tailstock clamping screws must be loosened so that the tailstock top may slide transversely to the ways of the lathe. Movement of the tailstock is accomplished by loosening one of the adjusting screws and tightening the opposite screw, as shown in Fig. 14 and Instruction Sheet 3.

## Setting Lathe RPM For Proper Cutting Speed (Instruction Sheet 8)

The engine lathe headstock is so constructed as to permit the changing of spindle speeds by means of a belt-driven cone pulley. A change in speed on the pulley can be obtained by shifting the belt to another step on the pulley. Slower speeds are obtained by engaging the back gears.

The power and motion of the feed mechanism is transmitted from the headstock to the carriage at various gear ratios for selected feeds.

The cutting speed for a material does not change but remains constant, while the RPM of the lathe spindle may be either increased or decreased depending upon the diameter of the work. The factors which determine cutting speed are: the kind and hardness of the material, the diameter of the work, the kind of material of which the cutting tool is made, the shape of the cutting tool, and the depth of the cut.

Feed depends upon the rigidity of the work, the manner in which it is held on the machine, the rigidity and shape of the cutting tool, and the cutting speed.

Coarse feeds and slow speeds are recommended for roughing cuts to prevent undue wear on the cutting edges of the tool and loss of time. The spindle speed is usually increased for finish cuts, while the amount of feed is decreased.

Cutting speeds for lathe work are always expressed in terms of feet per minute. The cutting speed is the number of feet of material which pass the cutting edge of the tool in

one minute as measured on the circumference of the work. Cutting speeds in feet per minute for various materials are given in Table 1.

Table 1. Cutting speeds of various materials.

Material	Turning and boring		Threading
	Rough cut	Finish cut	
Machine steel	90	100	35
Tool steel (annealed)	50	75	20
Cast iron	80	80	25
Brass	160	220	60
Aluminum	250	325	75

A simple method of computing the approximate RPM of the spindle is to multiply the cutting speed of the material by four and divide by the diameter of the work in inches.

Feed is the distance the tool advances for each revolution of the work. The feed may be thought of as the width of the chip. For example, if a 64th of an inch feed is used in turning a cylinder, it will take 64 revolutions of the work to advance the tool one inch. The width of the chip will be one-sixth-fourth of an inch.

The amount of feed depends upon a number of factors: the kind of material being turned, the cutting speed for the material, the depth of the cut, and the tool. For roughing cuts the feed should be as great as the machine will take and the cutting tool will withstand. For finish cuts the amount of feed is decreased, depending upon the finish desired.

When turning cast iron or any other material which has a scale on its outer surface, the tool is always set deep enough

to get under the scale for the first roughing cut.

If the beginner will calculate for the first few jobs the RPM necessary to give the required cutting speeds, he will soon be so accustomed to seeing the machine work properly that he will be able to tool up without making calculations.

## Laying Off Distances on Work (Instruction Sheet 9)

The three principal methods of laying off distance on work are with dividers, a hermaphrodite caliper, and with a steel rule and scribe.

The use of dividers in laying off a distance is illustrated in Fig. 31. The work is revolved on the lathe and chalk is applied at the point where a line is to be scribed. Set the dividers to the required distance and holding one point at the end of the work, allow the other point to make a mark around the work. Chalk makes the scribed lines more distinct.

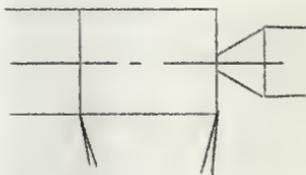


Fig. 31. Laying off distance on work.

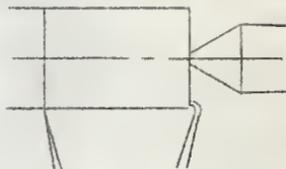


Fig. 32. Laying off distance on work.

Figure 32 shows the hermaphrodite caliper being used to mark a distance from the end of stock while the work is on the lathe. The work should be chalked. While the bent leg of the caliper is against the end of the stock, the work is rotated slowly while the straight leg scribes a line around the work.

The method of using a steel rule and scribe is illustrated in Fig. 33. To assure accuracy in marking, the end of the stock and the end of the rule should be placed against a solid straight surface to assure the rule and work being flush at their ends. A scribe is used to mark the distance desired.

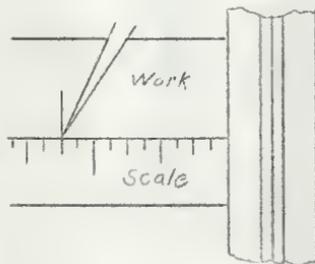


Fig. 33. Laying off distance on work.

## Turning a Filleted Shoulder

(Instruction Sheet 10)

Before turning a filleted shoulder, the finish diameter should be obtained. A slight amount of stock at the fillet should be left so the tool will have sufficient material to form the fillet.

Set a fillet tool on the center of the work in such a position as to permit the formation of a true arc.

Decrease the spindle speed so as to overcome chatter and to prevent the tool from digging in at the corner or at the finished diameter. Use a cutting compound to obtain a smoother finish.

Rough turn excess material left by the turning tool at the corner of the shoulder. Use both hand longitudinal and cross feed screws to guide the tool. Finish turn to the scribed lines. Considerable care must be exercised to prevent the tool from digging into the work.

Filing and Polishing on the Lathe  
(Instruction Sheet 11)

Filing on the lathe is a finishing operation primarily intended for removing tool marks from turned surfaces and for reducing diameters slight amounts. The mill file, with its single cut teeth, is better suited for this operation than a double cut file because it will not 'pin' as readily.

Filing on the lathe is performed at higher spindle speeds than those ordinarily used for turning purposes. A surface speed about double that of turning will usually be found sufficient.

A better appearance of the work will result if the file is rubbed with chalk before starting the filing operation. The file should be cleaned frequently with a file card and rechalked.

One mistake a beginner usually makes in filing is to move the file across the work too rapidly, thereby filing it out of round. A long slow stroke will always give best results. The file should not be lifted on the return stroke but the pressure should be relieved.

Another fault is that of bearing too hard when filing round work. The surface of the file at any given time touches only a very small surface of the work and bearing too hard will tend to file the piece out of round and also tend to 'pin' the file and scratch the work.

When filing, hold the handle of the file in the right palm. Extend the left arm over the work clear of the dog or chuck and hold the end of the file with the thumb on top and the fingers curled under the end. Place the file on the revolving work with the handle slightly inclined towards the headstock. Take long slow strokes at right angles to the work (Fig. 34). Do not over file and do not leave more than .001 of an inch for polishing.

When polishing, the speed can hardly be too great, but if a large piece is on the lathe and a high speed is wanted, care must be taken to have the work well balanced. Have plenty of oil on the tailstock center and do not adjust the tail center too tight.

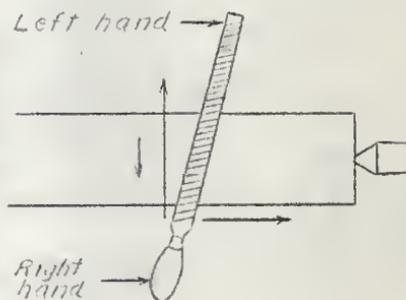


Fig. 34. Filing on lathe.

One of the best methods for polishing consists of using a strip of emery cloth held on a file in the same position as that for filing. The same strokes as for filing should be used.

Small jobs are polished by holding a piece of abrasive cloth over the work and applying pressure with the thumb and forefinger. A short strip of emery cloth should be used when it is held by hand. Hold the cloth in such a manner as to prevent it from winding around the work as this may cause injury to the operator. The cloth should be moved back and forth

slowly across the work to prevent the cutting of rings.

A finer grade and somewhat duller appearing surface can be produced by using oil on the cloth when polishing. Oil on the cloth tends to prevent deep scratches on the surface of the work. Continue polishing until the desired degree of smoothness is obtained. Use successively finer grades of abrasive cloth to produce a more highly polished surface.

Adjusting Lathe to Cut Required Number of Threads  
(Instruction Sheet 12)

Thread cutting on the lathe is a process of producing a ridge of uniform section by cutting a continuous groove around a cylinder as a cutting tool of required shape advances along the revolving work.

The speed at which the spindle turns in relation to the speed of the lead screw determines the distance the carriage and tool will advance when the split-nut is closed over the lead screw.

Change gears on a lathe are those gears whose position and size may be changed to establish varying ratios between the speed of the spindle and the speed of the lead screw.

Screw cutting lathes are made in two general types: the standard change-gear and the quick change-gear.

The standard change-gear lathe involves the problem of determining the gears to be used on the stud and lead screw. This may be done by referring to the index chart found on the front of the lathe. In the first column of the chart is found a list of the common pitches of threads; in the second, a list of the stud gears available; in the third, a list of the available screw gears. To select the gear of the proper ratio, follow down column one until the required number of threads per inch is found; then read the number directly opposite in the second column. This number represents the number of teeth on the gear to be placed on the stud. The number in the third column directly opposite is the number of teeth on

the gear which must be placed on the lead screw. When these two gears are mounted on the stud and lead screw respectively, as shown in Fig. 35, the gearing is set for cutting the required number of threads per inch.

When threads are to be cut on a quick change-gear lathe, the principle of the ratio between the speed of the spindle and the lead screw still holds. This type of lathe has change-gears built in as an integral part of the machine. They are contained in the

gear box, thereby making it unnecessary to remove gears and replace them with different sizes for thread cutting. Changes to accommodate a wide range of threads per inch can be made quickly by simply sliding the gears in the quick change-gear box by moving levers to a position as indicated on the index plate. Locate the desired number of threads per inch on the index plate. Follow the column in which this number occurs either to the left, right, up, or down to determine the necessary positions of various levers to cut the required number of threads per inch.

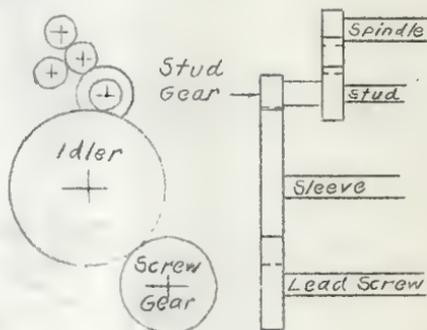


Fig. 35. Standard change-gear mechanism.

Set and Adjust an External Thread Cutting Tool  
(Instruction Sheet 13)

The engine lathe is almost invariably used for cutting screw threads on parts that are turned on the lathe. This method insures cutting the thread concentric with other turned surfaces.

The tool used to produce the American National Form Threads has cutting edges ground to conform to the standard profile of the thread. The cutting tool for this thread has sides ground to an included angle of  $60^\circ$ . The point is ground to a flat whose width is equal to one-eighth of the pitch of the thread. In common practice the point of the tool is usually ground sharp for finer pitch threads, and is sometimes stoned or honed with only a slight flat or radius instead of the required width of the flat.

The  $60^\circ$  angle of the threading tool should be checked by means of a center gage, as shown in Fig. 36. The center gage is also used to set the point of the cutting tool square with the work on the lathe.

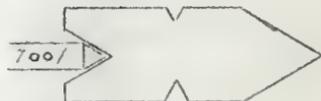


Fig. 36. Checking angle of tool.

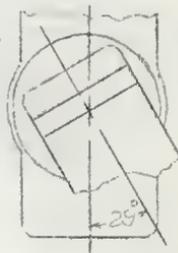


Fig. 37. Setting compound rest.

Before setting a threading tool the compound rest is swung around to the right to an angle of  $29^{\circ}$  (Fig. 37). The compound rest screw is used for adjusting the depth of cut and most of the metal is removed by the left side of the threading tool, as shown in Fig. 38. This permits the chip to curl out of the way better than if the tool is fed in straight. The right side of the tool will cut the thread smooth and produce a good finish, although it does not remove enough metal to interfere with the main chip which is taken by the left side of the tool.

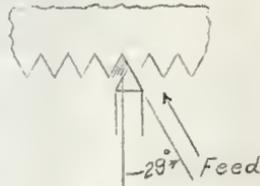


Fig. 38. Action of threading tool by using compound rest.

Secure the tool and holder in the lathe tool post and set the point of the tool at the height of the lathe center. Set the threading tool square with the work by placing the edge of the center gage against the turned side of the work and adjust the tool until the cutting point fits the V-shaped notch in the center gage (Fig. 39). When squaring the tool care must be exercised so as not to change the height of the tool point.

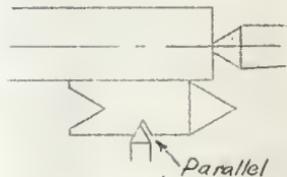


Fig. 39. Squaring tool point.

Check Setting of Lathe for Cutting Threads  
(Instruction Sheet 14)

Checking the number of threads which the lathe is set to cut should be a part of the procedure of anyone who is not familiar with the machine. This is best accomplished by closing the split-nut in the apron over the lead screw, turning on the power and allowing the spindle to revolve a few revolutions to take up the back-lash. After taking up the back-lash, stop the lathe and mark a spot on the dog driver, face plate or chuck. The tail of the dog in an up position is often used as a mark. Make a pencil line or lay a scale on the ways of the lathe at the right-hand end of the carriage. If the carriage moves one inch while the spindle revolves a number of times equal to the number of threads per inch to be cut, the lathe is set to cut the required number of threads.

Another check is to take a very light trial cut on the work just deep enough to scribe a line on the surface. To check the number of threads per inch, place a scale against the work (Fig. 40) so that the end of the scale rests on one of the scribed lines. Count the spaces between the end of the scale and the first inch mark, and this will give the number of threads per inch.

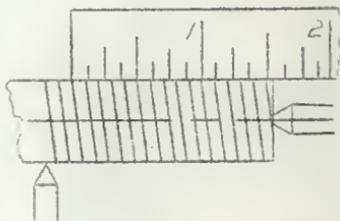


Fig. 40. Checking threads per inch.

**Chamfering Stock Previous to Threading**  
(Instruction Sheet 15)

The end of a piece of stock to be threaded should be chamfered before starting the threading operation. This gives the piece a more finished appearance and tends to prevent the formation of a large burr on the end of the stock.

Most thread chamfers are cut at an angle of  $30^\circ$  with the end of the stock and are cut to a depth approximately equal to the depth of the thread. The cutting edges of the threading tool make an angle of  $30^\circ$  with the end of the stock after the tool is squared up. To cut a chamfer move the tool to the end of the work and advance it longitudinally or transversely by hand until the desired chamfer is obtained (Fig. 41).

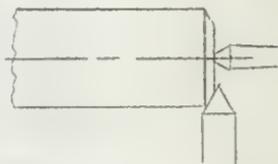


Fig. 41. Chamfering stock.

Thread Cutting in General  
(Instruction Sheet 16)

To cut a smooth and accurate thread on a lathe depends upon the operator's ability to recognize the various influences which have an important action on the tool when cutting. A beginner does not usually pay enough attention to small things and, in consequence, does not produce nice work.

The various forces acting on the tool which are of vital importance and cause a peculiar action on the cut are due to (1) the manner in which the tool is ground; (2) the direction of the cut; (3) the position and type of toolholder; (4) the fit of the ways of the machine; (5) the use of the proper cutting compound; and (6) a proper cutting speed.

Thread cutting on the lathe must be done at a slow RPM. The back gears of the lathe are put in operation to reduce the spindle speed. Threading speeds of various materials, expressed in feet per minute, are as follows: machine steel, 35; tool steel, 20; cast iron, 25; brass, 60; and aluminum, 75.

A cutting oil should be used when threading on steel. A smoother thread will be produced as the oil tends to prevent the tearing of the steel by the cutting tool. Apply oil before each cut.

If the lathe is equipped with a multiple hole face plate a hole should be marked with chalk to be certain of getting the dog tail back in the same place after the work has been removed from the centers for testing the thread with a test nut. Should the dog tail be placed in another hole or slot

the work will have been revolved through a part of a revolution in respect to the setting of the machine for the required number of threads.

Left-hand threads are readily cut on a lathe. All that is necessary is to reverse the rotation of the lead screw. This is done by shifting the reverse lever on the headstock. The only difference between cutting right and left-hand threads is that in the latter the tool must move towards the tailstock.

In considering the depth of cut in cutting threads, the tool is first fed to take a series of cuts, .003 to .005 of an inch. The distance the tool is fed in on the compound rest for each successive cut is decreased until the full depth of the thread is nearly reached. After most of the metal is removed, the thread may be finished to size by taking a few fine cuts of .001 of an inch or less by moving the tool straight in with the cross-feed screw.

If it is necessary for any reason to remove the tool before the thread is finished, the tool must be set accurately to gage and then, using one of the following methods, make sure the tool exactly enters the partly cut groove. Be sure all back-lash is taken up.

1. If the lathe is provided with a compound rest, adjust the tool to the desired position in the groove by manipulating the cross-feed screw and the compound rest screw.

2. If the lathe is not provided with a compound rest, loosen dog and turn the work until the tool enters the groove centrally.

Cutting Threads on Lathe Not Equipped with Threading Dial  
(Instruction Sheet 17)

After setting the threading tool and adjusting the lathe for the required number of threads per inch, thread cutting may begin.

Advance the threading tool until the point just touches the work, then note the micrometer reading on the cross-feed screw or set the micrometer collar to read zero.

Draw the tool back past the right-hand end of the work, then advance the tool point about .002 of an inch by turning the compound rest screw to the right.

Engage the split-nut and clamp it firmly to the lead screw, in which position it remains until the completion of the thread. Start the lathe and when the tool reaches the end of the threaded section quickly withdraw it by turning the cross-feed screw one of two revolutions to the left. Stop the lathe and reverse the spindle by shifting to the reverse pulley on the countershaft drive.

When the tool point on the backward travel is just past the end of the work stop the lathe before the point of the tool strikes the tailstock center. With the tool in this position advance the cross-feed screw until the micrometer collar reads the same as before beginning the first cut. Advance the tool for depth of second cut by turning the compound rest screw to the right about .002 of an inch. Start the second cut and when the tool reaches the end of the threaded section

quickly turn the cross-feed screw to the left and reverse the rotation of the spindle as before.

Repeat the above operation until the thread is cut to the desired depth or until a test nut fits the thread snugly.

Cutting Threads on Lathe Equipped with Threading Dial  
(Instruction Sheet 18)

A thread dial indicator, sometimes called a thread chasing dial, is a device used in thread cutting to indicate the position at which to engage the split-nut with the lead screw so that the cutting tool will always follow in the same groove as the previous cut.

The advantage in using the thread dial indicator is that the split-nut may be disengaged at the end of the cut to permit the carriage to be returned quickly to the starting position by hand. This saves time that would otherwise be required to bring the tool to the starting point by reversing the spindle.

The dial indicator indicates whether or not the work has turned a sufficient number of revolutions to permit the split-nut to be engaged with the lead screw at a position which will insure the correct tracking of the threading tool in the same groove that was previously cut.

As the lead screw revolves the dial on the indicator is turned and the graduations indicate the points at which the split-nut may be engaged. These points are reached when any graduation on the dial is just even with the line or point on the rim of the indicator.

When cutting an even number of threads per inch the split-nut is engaged as any graduation on the dial passes the line on the rim.

When cutting an odd number of threads per inch the split-nut is engaged as any numbered graduation on the dial passes

the line on the rim.

When cutting a fractional number of threads per inch the split-nut is engaged as either the even numbered or odd numbered graduations pass the line on the rim.

After setting the threading tool, adjusting the lathe for the required number of threads per inch, checking, and shifting the feed change lever to neutral, thread cutting may start.

Advance the threading tool until the point just touches the work, then note the micrometer reading on the cross-feed screw or set the micrometer collar to read zero.

Draw the tool back past the right-hand end of the work, then advance the point about .002 of an inch by turning the compound rest screw to the right.

Start the lathe and engage the split-nut on the proper graduation on the chasing dial. When the tool reaches the end of the threaded section withdraw it quickly by turning the cross-feed screw to the left a revolution or two. At the same time disengage the split-nut.

Draw the carriage back to the starting point with the apron handwheel. Advance the cross-feed screw until the micrometer collar reads the same as before beginning the first cut. Advance the tool for the depth of the second cut by turning the compound rest screw to the right about .002 of an inch. Start the second cut and when the tool reaches the end of the threaded section, turn the cross-feed screw quickly to the left and disengage the split-nut.

Repeat the above operation until the thread is cut to the desired depth or until a test nut fits the thread snugly.

## OPERATION SHEET FOR WOOD LATHE TOOL REST

(Blue Print L-54)

Objectives:

1. Turn cast iron.
2. Turn or square to a shoulder.
3. Use a snap gage.

Tools and Equipment:

Lathe, calipers, cast iron turning tool, surface plate, hammer, snap gage, V block, center punch, file, bench plate, center drill.

Operations:

1. Read Instruction Sheet 19 on cast iron turning.
2. Select stock.
3. Locate centers. I.S. 1.
4. Center drill both ends. I.S. 2.
5. Arrange to drive work between centers. I.S. 20.
6. Face or square right-hand end. I.S. 4.

Note. Use a cast iron turning tool to face the right-hand end. This will leave a small amount of metal around the center which will serve as a re-enforcement. This is later removed.

7. Rough turn cylinder to entire length. I.S. 20.
8. Check and adjust to eliminate taper. I.S. 7.
9. Rough turn entire cylinder to  $1/64$  of an inch oversize. I.S. 20.
10. Finish turn cylinder to fit gage. I.S. 21.
11. Turn to a square shoulder. I.S. 22.
12. Chamfer right-hand end. I.S. 23.
13. Finish facing right-hand end. I.S. 4.
14. Have instructor check work.

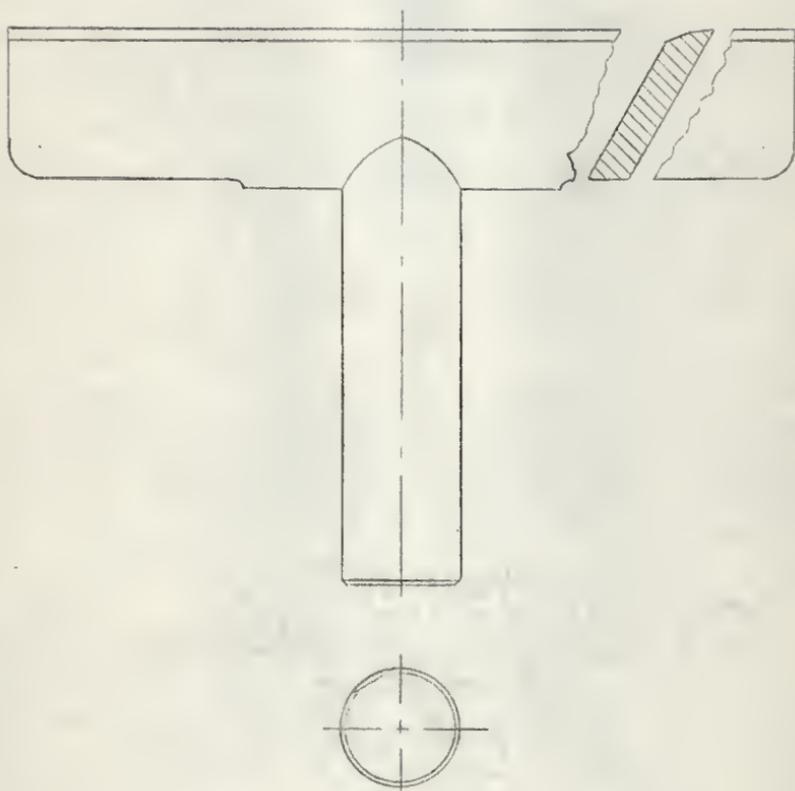


Fig. 42. Wood lathe tool rest.

## INSTRUCTION SHEETS FOR WOOD LATHE TOOL REST

Turning Cast Iron  
(Instruction Sheet 19)

In making an iron casting molten metal is poured into a sand mold. Hot metal coming in contact with cold sand causes the surface of the casting to become considerably harder than the interior. This hard surface is called scale or skin. Also a considerable amount of sand is fused with the iron in the surface of the casting. These conditions serve to render the surface of a casting very hard and brittle, and when machining it is important that the point of the cutting tool be well under this scale in order that the point of the cutting tool will not wear off and become dull too quickly.

In order to get the tool point under the scale it is necessary to take a deep cut as the first roughing cut. The RPM of the spindle must be reduced so that the machine may handle the heavy roughing cut. The engagement of the back gear is often necessary, depending upon the diameter of the piece to be turned.

Cast iron is always machined dry, no oil of any kind being used.

A cast iron turning tool for general turning is ground with a sharp round nose. No back or side rake is necessary because of the brittleness of the cast iron chip. The tool is left flat on top.

After the scaling operation is finished the spindle speed is increased until the cutting speed is somewhere between 60 and 80 feet per minute.

**Rough Turning Cast Iron Cylinder the Entire Length**  
(Instruction Sheet 20)

When turning the entire cylindrical part of the wood lathe tool rest special precautions are necessary in mounting the tool and holder.

The tool post must be placed in the left-hand end of the T-slot in the compound rest. Place the tool and holder in the tool post so that the cutting edge of the tool is to the left side of the compound rest (Fig. 43). These precautions are for the purpose of allowing the tool to cut the entire length of the cylinder to the shoulder before the arms of the piece being turned strike the compound rest.

After setting the tool as shown, engage the back gear so as to reduce the RPM of the spindle. The first cut must be under the skin of the tool rest. Allow the tool to cut along the cylinder, using the power feed, until the required distance from the end has been reached. When the tool has advanced this distance hold the apron hand wheel and slowly back the tool out from the shoulder by turning the cross-feed screw to the left.

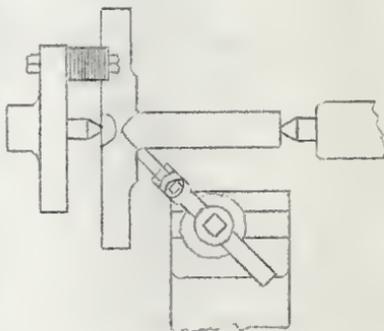


Fig. 43. Setting tool for tool rest.

Should the tool rest tend to clatter badly when shouldering the two arms, tighten the rest on its centers slightly. In all probability it will not be necessary to make another cut on the shoulders of the arms. Take succeeding cuts to the shoulder leaving a fillet at the end of the cut.

Finish Turning a Cylinder to Fit Gage  
(Instruction Sheet 21)

Calipers are not efficient for accurate measurements in the hands of amateur machinists. They are efficient for measuring stock, roughing cuts, lengths, and any dimensions that need not be extremely accurate. The caliper may be used if necessary for very close measurements but it is easier, quicker, and surer to use a micrometer or a gage.

For cylindrical work the snap gage is used and where a slight variation over or under nominal size is allowable a limit gage is used. Figure 44 shows such a limit gage. The 'go on' dimension of the gage must go on the cylinder with no application of force. It snaps on clear to the depth of the opening. The 'no go' section of the gage must not go on the cylinder. The difference in size between the 'go' and 'no go' is the tolerance allowed in the diameter of the cylinder.

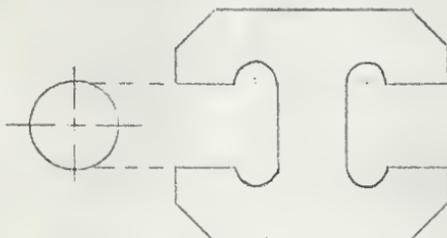


Fig. 44. Snap gage.

## Turning to a Square Shoulder

(Instruction Sheet 22)

A shoulder is the surface or step which joins two different diameters. Shoulders on turned work are formed to various shapes to suit the requirements of the particular part. They are formed to add strength to the part, to provide a square corner where parts are to be fitted together, or to add to the appearance of the job.

Figure 45 illustrates three common types of shoulders, namely, square shoulder, filleted shoulder, and angular shoulder.

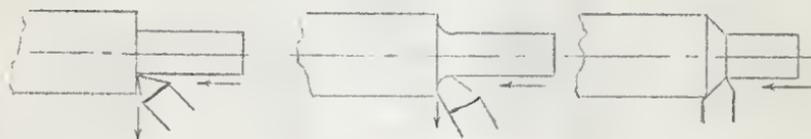


Fig. 45. Types of shoulders.

To lay out a shoulder, mark a line the required distance from the end of the work. If the work is of such a nature, chalk the spot where the line is intended to be marked. The line may be marked with a hermaphrodite caliper, a pair of dividers, or a rule and scriber. Figure 45 shows the type of tool to use in turning each type of shoulder.

A general method of squaring to a shoulder by using a special tool is shown in Fig. 46. The finishing tool is ground with just sufficient clearance on the front and side, and the corner is a right angle one-sixteenth of an inch back each

way, while the point is rounded with an oil stone.

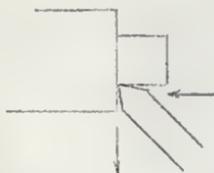


Fig. 46. Turning square shoulder.

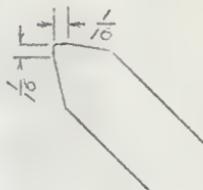


Fig. 47. Special squaring tool.

Chamfering an Edge  
(Instruction Sheet 23)

Chamfered edges are usually beveled at angles ranging from  $30^{\circ}$  to  $60^{\circ}$ , the most common being  $45^{\circ}$ . The purpose of chamfers is to remove sharp corners and add to the appearance of the job as shown in Fig. 48. They also provide safety and comfort in handling the finished parts. The chamfered ends permit turned parts to be fitted squarely against internal shoulders as shown in Fig. 49.



Fig. 48. Chamfered end.



Fig. 49. Chamfered end fits snugly.

The amount of chamfer and the angle at which it is turned may be specified on the drawing. However, in many cases, it is left to the judgment of the machinist to determine how much the end is to be chamfered.

In turning a chamfer the compound rest may be set at the required angle and the compound feed-screw used to feed the tool past the corner. Figure 50 shows a method of setting a square nosed or facing tool at an angle of  $45^{\circ}$  which is the most common angle.

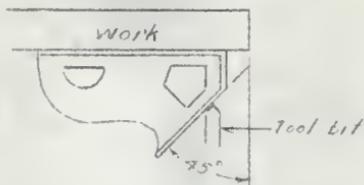


Fig. 50. Setting 45° chamfer.

## OPERATION SHEET FOR TAILSTOCK HANDWHEEL

(Blue Print L-44)

Objectives:

1. Chuck stock in four-jaw chuck.
2. Drill on the lathe.
3. Bore on the lathe.
4. Hand ream on the lathe.
5. Form to shape on the lathe.

Tools and Equipment:

Lathe, round file, 7/16-inch drill, 1/2-inch reamer, flat file, boring tool and holder, crescent wrench, handwheel holder, forming tool, round nose tool, drill chuck, emery cloth.

Operations:

1. Select stock.
2. Chip and file spokes and inside of rim.
3. Read '3-Jaw and 4-Jaw Chucks'. I.S. 24.
4. Remove face plate and mount 4-jaw chuck. I.S. 25.
5. True work in chuck. I.S. 26.
6. Face hub of wheel to required length. I.S. 27.
7. Spot center of hub and drill hole. I.S. 28.
8. Bore hole to take 1/2-inch reamer. I.S. 29.
9. Ream hole its entire length. I.S. 30.
10. Remove work from chuck and mount on handwheel holder.  
Note. Check out handwheel holder from tool crib.
11. Remove chuck and mount face plate. I.S. 25.
12. Mount work between centers. I.S. 3.
13. Form rough shape of rim. I.S. 31.
14. Finish form shape of rim. I.S. 32.
15. File tool marks on outside of rim. I.S. 11.
16. Polish rim of wheel. I.S. 11.
17. Have instructor check work.

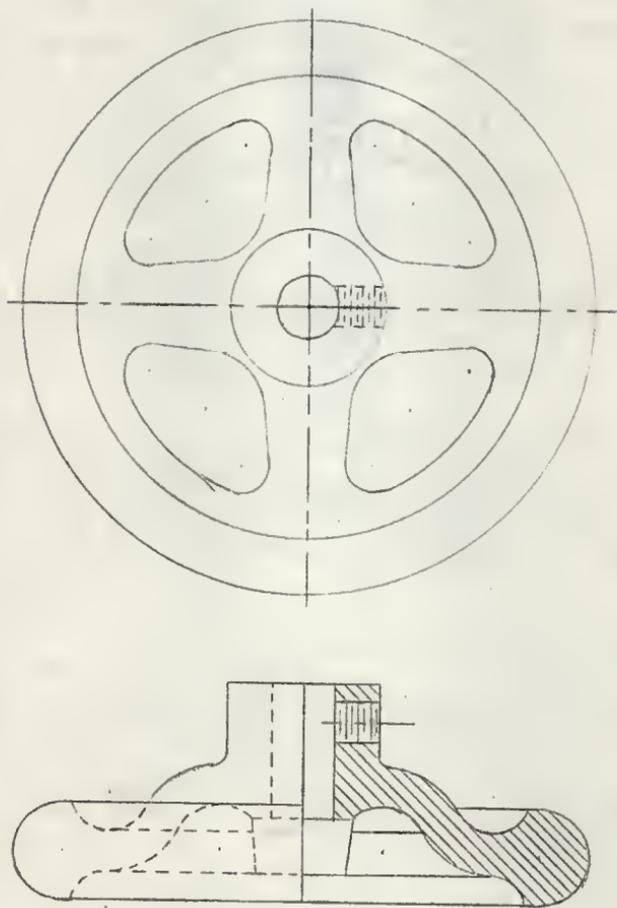


Fig. 51. Tailstock handwheel.

## INSTRUCTION SHEETS FOR TAILSTOCK HANDWHEEL

3-Jaw and 4-Jaw Chucks  
Instruction Sheet 24)

A chuck is a device which, when fixed to the spindle of a machine, is used to hold either a tool or material to be worked upon. The jaws of a chuck do the gripping and they are adjustable so that work of different sizes may be held. The jaws are operated either by screws or by a scroll. Gear blanks, pulleys, collars, bushings, etc., where the outside must run true with the hole are held in a chuck while they are being drilled, bored, reamed, or possibly threaded.

A 4-jaw Independent chuck has four reversible jaws, each of which may be independently adjusted. This type of chuck is recommended if the lathe is to have but one chuck as it will hold square, round, and irregular shapes in either a concentric or an eccentric position.

The 3-jaw Universal chuck is used for chucking round and hexagonal work as the three jaws move simultaneously and automatically center the work.

Since there is no way to adjust the jaws independently in a 3-jaw chuck, it is not used where extreme accuracy is required. The 4-jaw Independent chuck should always be used when work must be centered to run dead true because wear in the Universal chuck soon renders it inaccurate for exacting work.

Removing and Mounting Face Plates and Chucks  
(Instruction Sheet 25)

When removing face plates or chucks, extreme care must be taken to avoid damaging the threads in the plate or chuck back or those on the nose of the lathe spindle. Do not allow the plate or chuck to drop on the ways of the lathe as dents or marks tend to make for inaccurate work.

In removing a face plate or chuck, have at hand a block of hard wood, about two inches square, and long enough to rest on the back ways and reach the slot in the face plate or the jaw of the chuck when in a position corresponding to three o'clock (Figs. 52 and 53). Arrange the lathe for a fairly slow speed and running it backwards by hand allow the edge of the slot or jaw to strike sharply against the end of the block. This should jar the plate or chuck loose. If it doesn't, arrange the lathe for a slower speed and give the belt a harder pull.

Before mounting a chuck or face plate on the lathe spindle,



Fig. 52. Removing face plate.



Fig. 53. Removing chuck.

clean and oil the threads of the spindle and chuck back thoroughly. Also clean the shoulder of the spindle where the chuck back fits against it. A very small chip or burr at this point will prevent the chuck from running true. Cleanliness is very essential for accurate work.

In mounting a chuck hold it in the cradle of the right hand and arm against the spindle nose of the lathe. If a cradle block is at hand, use it to hold the chuck in position (Fig. 54). When the chuck is in position turn the lathe spindle with the left hand to screw the chuck on the spindle just tight enough to hold it securely.

Avoid running the lathe with power while screwing the chuck on the spindle and do not spin the chuck up to the shoulder as it may be very difficult to remove.

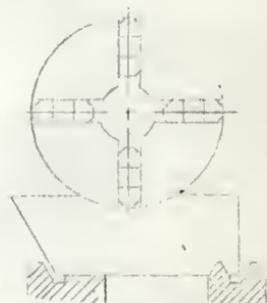


Fig. 54. Cradle block holding chuck.

Truing Work in a Chuck  
(Instruction Sheet 26)

Before placing work in a chuck adjust the jaws approximately to the size of the work by withdrawing or advancing the adjusting screws of each jaw to conform to the size of the work and the concentric grooves turned in the face of the chuck (Fig. 55).

Place the work in the jaws of the chuck and fasten it lightly by tightening the adjusting screw of two opposite jaws, for example, jaws 1 and 3. Tighten these just enough to hold the work securely. In a similar manner tighten jaws 2 and 4.

Remove the chuck wrench. Make a practice of doing this or you may forget it sometime and thus cause an accident.

Revolve the work slowly by hand or power and using the carriage as a support for the arm, bring a piece of chalk lightly in contact with the face of the revolving work (Fig. 56). The chalk will leave a mark on that part of the face which is farthest to the

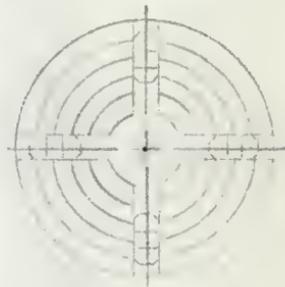


Fig. 55. Concentric rings on chuck face.

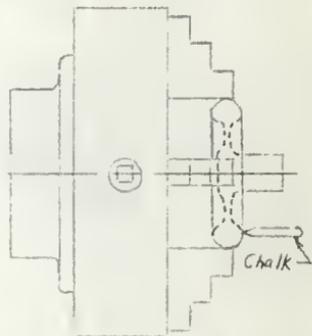


Fig. 56. Truing face of work.

right. Tap the chalked portion back to the left with a soft hammer. Rub off chalk and repeat until the face of the work runs parallel to the face of the chuck.

After getting the face of the work to run true hold the chalk against the revolving hub, as shown in Fig. 57. Stop the lathe and locate the high spot (chalk mark). Slightly loosen the adjusting screw of the jaw opposite the chalk mark. Turn the chuck a half turn and tighten the adjusting screw, thus pushing the work towards the center of the chuck.

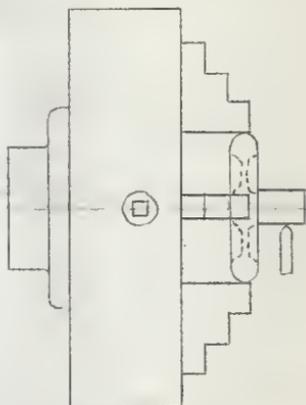


Fig. 57. Truing hub of work.

If the chalk mark extends as much as one-third the distance around the work, it will be necessary to loosen two of the adjusting screws opposite the chalk mark, then tighten the other two opposite screws. However, this should be done by adjusting one pair of jaws at a time. Continue in this manner until the work runs true, after which firmly tighten the adjusting screw of each jaw, then proceed with the machining operation.

Facing Work in a Chuck  
(Instruction Sheet 27)

Facing on the lathe is the process of machining an end or face in order to produce a smooth flat surface which is square with the axis of the job. Through the facing operation the work is reduced to a specified length and the faced end provides a surface from which measurements of length may be taken. The end is faced at right angles to the axis of the work as the cutting tool advances with the cross slide. Work held in a chuck is more conveniently faced than work held between centers.

Where a considerable amount of metal is to be removed a roughing cut should be made. A roughing cut is fed from the outer edge towards the center while a finishing cut is fed from the center out. The tool point must be set on center in order not to leave a small amount of metal at the center.

When facing, the carriage should be held in place by holding the longitudinal handwheel or clamping the carriage with the clamping screw. Either of these precautions is necessary to assure a face at right angles to the axis of the work.

Spotting the Center and Drilling on the Lathe  
(Instruction Sheet 28)

Before drilling on the lathe it is necessary to find the center of the work and make an indentation for the lips of the drill to be guided into the work at the center.

One method of centering the work is to use a flat drill for spotting the center. The spotting tool is mounted exactly on center so that both lips do the cutting (Fig. 58). The indentation at the center should only be deep enough to assure the drill being guided to the center of work.

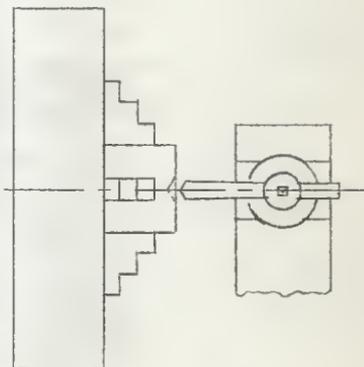


Fig. 58. Spotting center.

There are several methods of drilling on the lathe. One method is to make use of a drill, drill chuck, and drill chuck holder. This method has the advantage of the operator being able to make use of either straight shank or taper shank drills.

In making use of a straight shank drill, chuck the drill in the drill chuck and place the tapered shank of the chuck in the tapered hole in the drill chuck holder. Mount the assembled drill outfit on the lathe (Fig. 59).

In making use of a tapered shank drill, the tapered shank of the drill is placed in the tapered hole in the holder and the assembly mounted as shown in Fig. 59, except for the elimi-

nation of the drill chuck.

While either of these assemblies may be held by hand, it is advisable to rest the handle of the holder on some part of the cross slide and hold it against the tailstock center with the end of a tool holder clamped in the tool post.

With this set-up (Fig. 59),

the carriage offers enough resistance to prevent the drill from pulling away from the tailstock center, thus breaking the drill.

In drilling, pressure on the drill is obtained by turning the tailstock handwheel. At the same time advance the carriage with the longitudinal handwheel so that the tool holder keeps just ahead of the drill holder, thus preventing the holder from pulling away from the tailstock center.

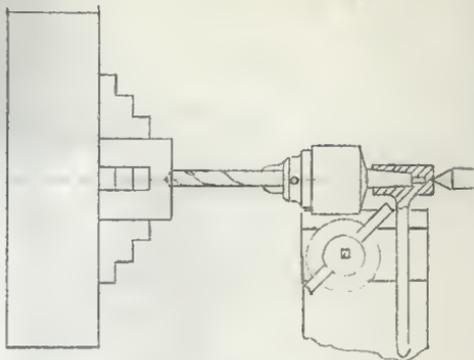


Fig. 59. Drilling on lathe.

Boring on the Lathe  
(Instruction Sheet 29)

Boring should not be confused with drilling because drilling usually refers to cutting a hole in a solid piece of material, while boring refers to enlarging a drilled or cored hole. Boring on the lathe is an internal turning process as compared with external turning on an outside diameter.

The boring operation is used principally to enlarge a hole, to machine a hole concentric with other turned surfaces, or to bore a hole in which a reamer may be started. In boring a hole to take a hand reamer it is left about .005 of an inch undersize or of such a diameter that about one-third of the tapered end of the reamer will enter the hole. A hand reamer is straight for nearly its entire length. It is slightly tapered, smaller toward the front, for a distance about equal to its diameter, to permit its entering the hole to be reamed.

The boring tool is a turning tool held in a bar or holder or forged on the end of a bar. It is ground like a turning tool that cuts from left to right. For boring small holes a light holder having a 'V' groove and a clamping yoke is used to support small diameter tools. These tools are usually forged.

Clamp the boring tool holder in the tool post at such an angle that the boring tool is parallel to the ways of the lathe. The boring tool is clamped in the holder with its cutting point on the horizontal center of the work (Fig. 60). Just sufficient length should protrude from the holder for the

cutting tool to pass through the length of hole to be bored (Fig. 61).

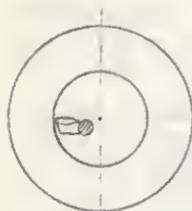


Fig. 60. Centering boring tool.

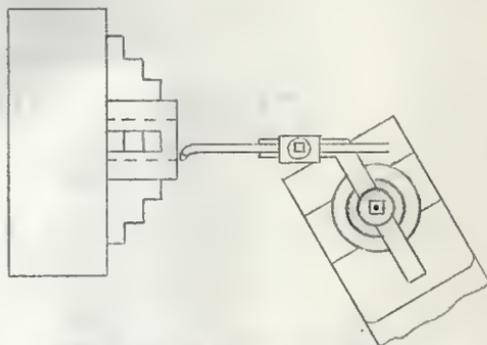


Fig. 61. Boring on lathe.

In beginning a cut, the carriage is moved forward so that the cutting edge of the tool is close to the edge of the hole to be bored. The lathe is started and a light trial cut is taken for a short distance into the hole. Stop the lathe and check the hole for size. Engage the longitudinal power feed and take a cut to the depth of the hole. Due to the tendency of a boring tool to chatter and spring away from the work under heavy cuts, the amount of feed should be decreased. Check the size of the hole before taking each succeeding cut and then bore to size.

Hand Reaming on the Lathe  
(Instruction Sheet 30)

Reaming is the process of finishing a hole to the required size by means of either a machine reamer or a hand reamer. Power is never used on a hand reamer.

After a hole has been bored to the correct size for receiving the tapered end of a reamer, the latter is started in the opening. To assure reaming a hole parallel to the axis of the lathe it is necessary to center the tail end of the reamer on the tailstock center. Make sure that the tailstock is set on center.

Reaming by hand is best accomplished by working on the rear side of the lathe. A wrench is placed on the square end of the reamer and a turn or part of a turn is taken clockwise with the right hand operating the wrench. At the same time the left hand operates the tailstock handwheel to take up the slack. Forcing the reamer too hard with the handwheel will place too much strain on the reamer, making it difficult to turn and possibly causing a break in the reamer. Continue reaming until the tapered end of the reamer is entirely through the hole.

To remove a reamer, continue turning it in a cutting direction, and at the same time gradually withdraw it until it turns out of the hole. A reamer is never turned backwards as that would quickly dull the tool.

Rough Forming on the Lathe  
(Instruction Sheet 31)

In rough forming a handwheel begin with a spindle speed reduced by use of the backgear. The first cut is generally a deep scaling one. To rough form the rim of the wheel, set a round-nose tool on center at an angle such that the tool may be used to face one side of the rim. Face both sides of the rim until the desired thickness is obtained. Care should be taken to remove about equal amounts of metal from each side.

After obtaining the desired thickness feed the cutting tool across the outside of the rim parallel to the axis of the lathe. Pay no attention to the curvature of the rim, but continue reducing the diameter of the wheel until it is about one-sixty-fourth of an inch oversize and somewhat rectangular in form (Fig. 62).

Further rough form the rim by using the hand longitudinal and cross feeds. The free hand forming should be made to conform as nearly as possible to the final shape. After obtaining this rough shape the piece is ready for a finish forming tool.



Fig. 62. Rough forming handwheel.

Finish Forming on the Lathe  
(Instruction Sheet 32)

Before a piece of stock is finish formed it should be rough formed by hand operation of the longitudinal and cross feeds with a round-nose tool.

To use a forming tool with a broad cutting edge reduce the spindle speed to lessen the chatter of the wheel. The tool must be set on the center of the work and squared with the axis of the lathe. Use the hand feeds. As the cutting progresses measurements must be made frequently to determine when the cutting is done.

In forming a handwheel care must be taken to get the diameter correct by very careful use of the calipers. The tool should be made to start cutting on both faces of the wheel at the same time. If the wheel is to be filed and polished it is not necessary to leave more than .005 or .010 of an inch for the final finish.

## OPERATION SHEET FOR TAILSTOCK CLAMPING SCREW

(Blue Print L-46)

Objectives:

1. Turn approximate taper by the offset tailstock method.
2. Form an irregular curve to fit a templet.

Tools and Equipment:

Lathe, bench plate, V block, surface gage, center punch, hammer, file, combination drill, rule, turning tools, calipers, micrometer, test nut, templet, emery cloth.

Operations:

1. Select stock.
2. Find centers and center drill ends. I.S. 1.
3. Square ends, obtaining proper length. I.S. 4.
4. Redrill centers to proper size. I.S. 2.
5. Rough turn entire cylinder. I.S. 6, 7.
6. Finish turn cylinder to major diameter. I.S. 6.
7. Lay off distance for shoulder. I.S. 9.
8. Rough turn shouldered end. I.S. 22.
9. Finish turn shouldered end. I.S. 22.
10. Square shoulder. I.S. 22.
11. Lay off distance of threads. I.S. 9.
12. Cut threads. I.S. 13, 14, 15, 16, 17, 18.  
Note. When the tool point reaches end of thread it should be withdrawn from groove before movement of carriage is stopped; otherwise, the tool point will cut a groove straight around stock. Either withdraw tool at end of thread quickly by turning the cross-slide handle while work is revolving, or shift off power before tool point reaches end of thread and finish the cut by hand power on the belt.
13. Use a split nut and reverse stock in lathe.
14. Read 'Taper Turning in General'. I.S. 33.
15. Calculate tailstock setover for required taper. I.S. 34.
16. Setover tailstock. I.S. 34.
17. Lay off distance for taper. I.S. 9.  
Note. Leave enough material at the small end of taper to form fillet.
18. Partially turn and check for proper taper. I.S. 34.
19. Finish turn taper. I.S. 6.  
Note. Leave last 1/8 of an inch on large end straight for forming round corner.
20. Turn fillet. I.S. 10.
21. File round corner at fillet end until templet fits. I.S. 35.
22. File round corner at end until templet fits. I.S. 11.
23. Polish all of handle to shoulder. I.S. 11.
24. Have instructor check work.

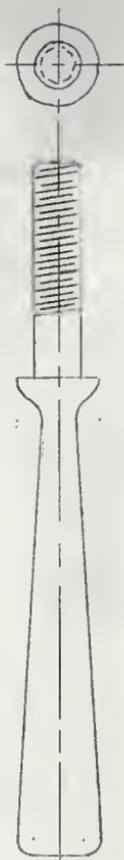


Fig. 63. Tailstock clamping screw.

## INSTRUCTION SHEETS FOR TAILSTOCK CLAMPING SCREW

Taper Turning in General  
(Instruction Sheet 33)

Taper turning is the process of producing a conical form which uniformly increases or decreases in diameter as the tool is fed longitudinally along the revolving work.

Tapers may be turned on a lathe by four different methods. The method used depends on the available equipment and the nature of the job. Short or abrupt tapers may be turned with a square nosed tool set at the required angle. The compound rest may be used for short tapers on which the length of the taper does not exceed the amount of travel of the compound. The use of the taper attachment is preferred to all other methods of turning tapers. However, not all lathes are equipped with a taper attachment. The offset tailstock method of taper turning is frequently resorted to because of the limitations of the form tool and compound rest methods and the fact that not every lathe is equipped with a taper attachment.

When the tailstock center of the lathe is offset the center line of the work and the line of travel of the turning tool are no longer parallel, and, as the work revolves and the tool moves longitudinally, a taper is turned (Fig. 64).

While the offset method is widely used for taper turning it has a number of disadvantages. When turning duplicate pieces slight variations in length of work and depth of center holes will result in variations in the amount of taper turned.

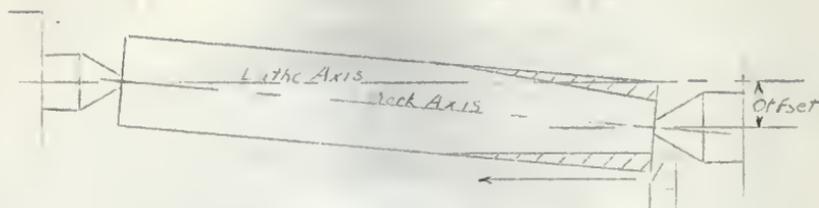


Fig. 64. Principle of taper turning.

Each time the tailstock is offset for taper turning, the center must be realigned for straight or parallel turning.

The offset method is limited to work held between centers; therefore, only external tapers may be turned.

Another objection to this method is that the center holes in the work do not bear uniformly on the lathe centers (Fig. 65). The center holes become distorted and often cause the centers themselves to become scored.

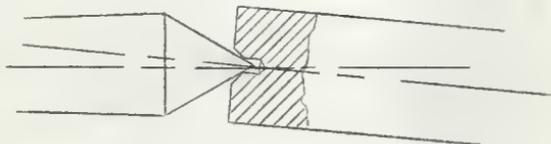


Fig. 65. Wear on lathe centers.

## Calculation and Adjustment of Offset of the Tailstock for Taper Turning

(Instruction Sheet 34)

The approximate amount that the tailstock should be offset for turning a given taper may be figured by two methods. In each case the length of the work is a factor in computing the amount of offset. The method used will be determined by the facts given concerning the taper.

To calculate the amount of offset when the taper per foot and length of work are given, divide the overall length of the work in inches by 12 and multiply the quotient by one-half the required amount of taper per foot.

To calculate the amount of offset required when the end diameters and lengths are given, divide the overall length of the work by the length of the tapered section and multiply the quotient by one-half the difference between the large and small diameters of the taper.

When measuring the work and calculating the offset, disregard eighths of an inch and take the nearest quarter inch under the dimension. The centers of the lathe enter the work a little way and will offset the error in disregarding the fraction.

To offset the tailstock towards the operator, secure the tool post in the compound rest by clamping a tool holder side-wise in the slot. Move the tailstock spindle out until it extends far enough to provide a surface from which measurements may be taken (Fig. 68). Tighten the spindle lock.

Move the tool post so that it nearly touches the tailstock spindle and then reverse the direction of the cross-feed screw to take up backlash. Set the graduated collar on the cross-feed screw at zero or note the reading of the collar. Move the tool post up to the tailstock spindle by turning the compound rest screw. Adjust until a paper drags slightly when drawn between the points of contact, as illustrated in Fig. 67.

Turn the cross-feed screw out until the graduations on the collar indicate that the tool post has moved a distance equal to the amount of offset. Loosen the tailstock clamping nuts and turn set-over screws to offset the tailstock. Adjust until the piece of paper drags when drawn between the tool post and spindle as before (Fig. 68).

To offset the tailstock away from the operator the procedure is the same as that above except that it is not necessary to reverse the direction of the cross-feed screw to take up backlash. Offset the tailstock a little more than the required amount and move the tool post in for the offset required; then adjust the tailstock until a piece of paper drags when drawn between the tool post and spindle.

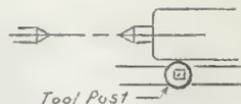


Fig. 66. Setting tailstock for taper.

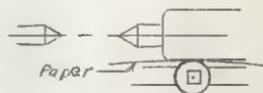


Fig. 67. Setting tailstock for taper.

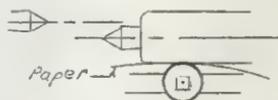


Fig. 68. Setting tailstock for taper.

Practically the same procedure may be followed for offsetting the tailstock by making use of a pair of inside calipers instead of a strip of paper.

Make certain that the tailstock clamping nuts are tightened before starting to turn the taper.

One method used to check the amount of taper being turned is to compute the amount of taper per inch or multiple thereof from the facts given. After cutting enough taper to measure, lay off the predetermined length with a pair of dividers and measure the diameter at each end of the length with a micrometer. Slight adjustments in the offset should be made if the taper is not correct.

Contour Forming  
(Instruction Sheet 35)

To form irregular curves first use a radius or fillet tool. Slow the RPM to reduce chatter which is caused by turning on a small diameter with a broad nosed tool. Turn the fillet or inside arc until it is tangent to the taper and the proper distance from the shoulder, as demonstrated in Fig. 69.

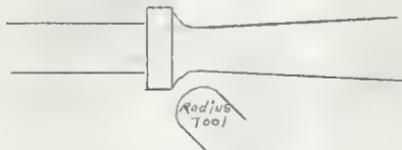


Fig. 69. Inside radius turning.

An outside arc is filed to shape. Care must be taken to file the arc without allowing the corner of the file to score the small end of the taper. One method to prevent scoring the taper is to hold with the left hand a strip of emery cloth around the small end of the taper. The smooth side of the emery cloth should be next to the metal. The left hand should be under the revolving work. Then file the arc to shape with the right hand holding the file (Fig. 70).

Outside arcs may be roughed to shape with a turning tool after which they are filed and then polished along with the rest of the handle. To polish a fillet, wrap emery cloth around a dog wrench or something smaller in diameter in order

to reach the inside arc.

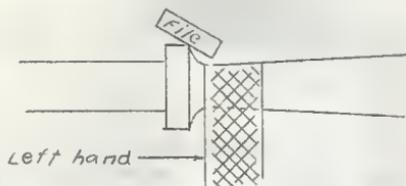


Fig. 70. Filing outside arc.

Check the arcs from time to time to determine whether or not the form fits the templet (Fig. 71).

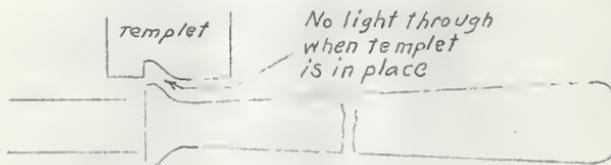


Fig. 71. Use of templet.

## OPERATION SHEET FOR WOOD LATHE FACE PLATE

(Blue Print L-61-62)

Objectives:

1. Counterbore a hole.
2. Use 'go' - 'no go' plug gage.
3. Cut internal threads.
4. Turn on a mandrel.

Tools and Equipment:

Lathe, boring tool and holder, plug gages, turning tools, calipers, combination square, mandrel, 1-1/2-inch dog, tap, 1/4-inch radius tool, 12-inch crescent wrench, inside threading tool.

Operations:

1. Select stock.
2. Remove face plate and mount 4-jaw chuck. I.S. 25.
3. Chuck stock in chuck with hub out.
4. True up stock in chuck. I.S. 26.
5. Arrange RPM of spindle for scaling cast iron. I.S. 8.
6. Scale entire surface on hub side. I.S. 19 and 20.
7. Face hub to 1/32 or 1/16-inch undersize from face of plate. I.S. 27.  
 Note: Leave rest of surface as it is. Reason for facing hub back undersize is that when scale is removed on other side of plate, side scaled first will warp. If first side had been finished to size, plate would always be warped. It is necessary to face hub undersize so that more material will have to be taken off to eliminate warp.
8. Bore cored hole to fit plug gage. I.S. 29.  
 Note: The 'go' end of gage must go through hole while 'no-go' end must not enter.
9. Counterbore hole to fit plug gage. I.S. 36.
10. Adjust lathe to cut required number of threads. I.S. 12.
11. Check setting of lathe for cutting required number of threads. I.S. 14.
12. Set internal threading tool and cut threads. I.S. 37.  
 Note: Chamfer shoulder before cutting threads. In cutting depth of threads, continue cutting until tool point begins to cut in counterbore.
13. Finish threads with a tap. I.S. 30.  
 Note: Hand tapping on lathe is accomplished by supporting tap on tailstock center and turning with wrench.
14. Remove work from chuck and mount on threaded mandrel.

15. Remove scale from the remainder of outside surface.  
I.S. 19 and 20.  
Note: After scaling remaining surface leave it as it is.
16. Turn hub to size. I.S. 6.  
Note: Leave enough material for forming fillet.
17. Finish face hub side of plate to size. I.S. 27.
18. Form fillet. I.S. 32.
19. Turn plate to diameter. I.S. 6 and 8.
20. Finish face front of plate to required thickness. I.S. 27.
21. Chamfer edge. I.S. 23.
22. Break all sharp corners with file.
23. Have instructor check work.

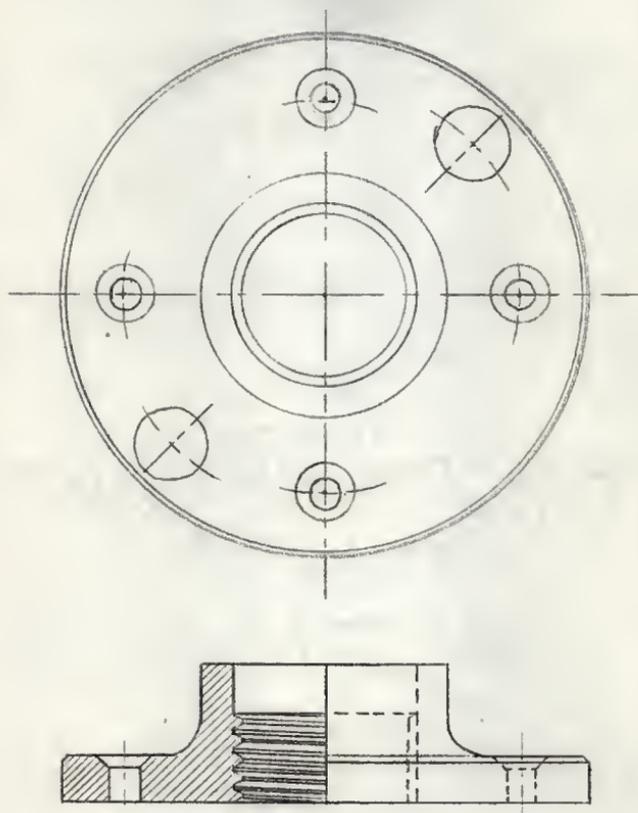


Fig. 72. Wood lathe face plate.

## INSTRUCTION SHEETS FOR WOOD LATHE FACE PLATE

Counterboring on the Lathe  
(Instruction Sheet 36)

Counterboring is another form of boring on the lathe which refers to enlarging the front of a bored hole to a definite size for a given depth (Fig. 73).

The counterbored hole has parallel sides and a shoulder which is either faced square or chamfered if the remainder of the hole is to be threaded. Counterboring involves boring and facing an internal shoulder.

The regular boring tool with a rounded cutting edge may be used to rough bore the counterbored hole. For the finish cut when the shoulder is to be square the tool should be ground to produce a square shoulder (Fig. 74). The same tool is used for facing the internal shoulder as well as for finish boring.

When measuring the diameter of a counterbore, use a pair of inside calipers or check your final diameter with a plug gage. To measure the depth of the counterbore use a scale, measuring from the shoulder to the outside edge. In some

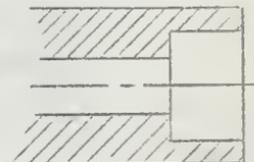


Fig. 73. A counterbore.

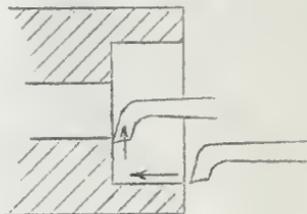


Fig. 74. Boring a counterbore.

cases it is necessary to lay a scale across the opening and use another scale to measure from the shoulder out.

## Setting Internal Threading Tool and Cutting Threads (Instruction Sheet 37)

Setting an internal thread cutting tool requires a great deal of care. The tool point must be set on center. This is best accomplished by placing the tool in the holder and the holder in the tool post. Turn the tool post so that the point of the tool may be elevated to the height of the tailstock center. After setting the tool on center carefully turn the tool post so that the threading tool is in approximately the correct position for thread cutting.

The tool must be squared or made perpendicular to the inside wall of the bored hole. By using a center gage against the face of the chuck, the face of the work, or inside the bored hole, the tool may be squared up with the axis of the hole (Fig. 75). The tool should be clamped as short in its holder as the length of the thread will permit. This tends to prevent unnecessary spring in the tool.

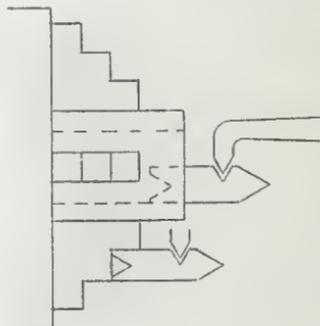


Fig. 75. Setting threading tool.

Cutting an internal thread of whatever shape is to cutting an external thread as boring is to turning--a little more care for the spring of the tool, and a little more difficulty in measuring. The compound rest is moved around to the left  $29^\circ$  instead of to the right, and the cross

slide is moved towards the operator for the cut, and away from the operator before reversing or moving the tool back to the beginning of the thread.

With the apron handwheel advance the tool to within one-fourth of an inch to the work, then with the cross-feed screw adjust the tool until the point is even with the near wall of the hole. Start the lathe and engage the half-nut when a line on the dial indicator is just even with the line on the rim. As the tool advances feed it into the cut by moving the compound rest towards the operator. When the threading tool reaches the terminus of the thread, quickly draw it away from the work by giving the cross-feed screw a revolution or two toward the right, and at the same time immediately disengage the half-nut.

Draw the carriage back to the starting point with the apron handwheel, set the cross-feed screw in the same position as for the first cut and advance the depth of cut with the compound rest screw about .002 to .005 of an inch.

Repeat these operations until the thread is cut to the desired depth.

Internal threads are best finished with a tap but if a tap is not available the thread must be bored to size.

## OPERATION SHEET FOR TAILSTOCK SPINDLE SCREW

(Blue Print L-42)

Objectives:

1. Turn neck or groove.
2. Cut left-hand threads.

Tools and Equipment:

Lathe, cutting tools, micrometer, center drill, surface gage, V block, bench plate, scale, threading tool, necking tool, file, test nut.

Operations:

1. Select stock.
2. Locate centers of ends. I.S. 1.
3. Center drill ends. I.S. 2.
4. Attach dog and mount work on centers. I.S. 3.
5. Square both ends, obtaining correct length. I.S. 4.
6. Re-drill center holes to proper size. I.S. 2.
7. Arrange RPM of spindle for proper cutting speed. I.S. 8.
8. Straight turn between centers to obtain cylinder. I.S. 6.
9. Check and adjust for taper elimination. I.S. 7.
10. Rough turn entire cylinder. I.S. 6.
11. Finish turn entire cylinder to major diameter. I.S. 6.
12. Lay off distances for shouldered ends. I.S. 9.
13. Rough turn both ends to shoulder, checking and adjusting for taper elimination. I.S. 6, 7.
14. Finish turn both ends, being careful to keep within proper tolerances. I.S. 6.
15. Square both shoulders. I.S. 22.
16. Turn small diameter on end to be threaded. I.S. 6.
17. Turn neck to the proper diameter. I.S. 38.
18. Adjust lathe to cut required number of threads per inch and check setting. I.S. 12, 14, 39.
19. Set and adjust an external threading tool. I.S. 13.
20. Cut threads until test nut fits. I.S. 16, 17, 18, 39.
21. Clean neck of burrs if necessary. I.S. 38.
22. Break all sharp corners with file.
23. Have instructor check work.

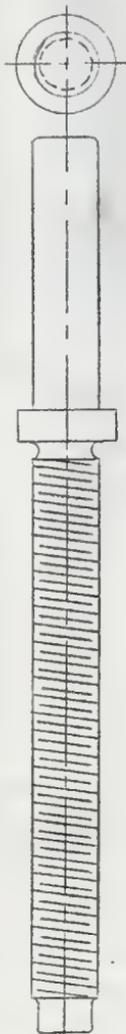


Fig. 76. Tailstock spindle screw.

## INSTRUCTION SHEETS FOR TAILSTOCK SPINDLE SCREW

Turning a Neck or Groove  
 (Instruction Sheet 38)

Necking or grooving is the process of turning a channel or furrow on a cylinder. The shape of the tool and the depth to which it is fed into the work governs the shape of the neck.

The square groove, the round groove, and the V-shaped groove are the three types most commonly used. These are illustrated in Fig. 77.

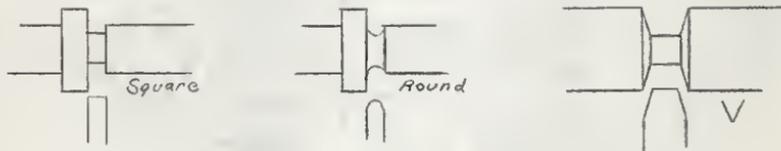


Fig. 77. Types of necks.

Necks or grooves are frequently cut at the end of threads or against shoulders in order to provide a channel into which the threading tool may run. The neck which is cut against a shoulder makes it possible to end a threaded part with a full thread. The neck also provides a clearance for the threaded piece to be screwed tightly against a shoulder.

When a grinding operation is to follow the turning process, a neck is cut at the shoulder end to provide a clearance for the grinding wheel in order to prevent hitting the shoulder with the side of the wheel.

The V-groove is used extensively on pulleys made for the V type of belt.

The spindle speed should be reduced in necking so as to prevent chatter and digging in.

## Cutting Left-Hand Threads

(Instruction Sheet 39)

Left-hand threads are cut in the same manner as right-hand threads except that the carriage must be made to travel from the headstock towards the tailstock, or from left to right. This is done by shifting the reverse feed lever in the train of gears on the headstock. By shifting this lever the lead screw is made to revolve in the opposite direction which causes the carriage to move to the right.

The threading tool must be fed into the work with the cross-feed screw and not the compound rest screw unless a tool with a right-hand rake is being used. In such a case the compound rest should be turned  $29^{\circ}$  to the left and the same procedure followed as in cutting right-hand threads.

In cutting an odd number of threads per inch, the half-nut lever should be engaged only when a numbered line on the dial passes the fixed point on the rim.

## OPERATION SHEET FOR WOOD LATHE LIVE CENTER BLANK

(Blue Print L-12)

Objectives:

1. Turn an exact taper.
2. Use external taper gages.

Tools and Equipment:

Lathe, hammer, bench plate, center drill, V block, rule, surface gage, calipers, micrometer, center punch, file, turning tools, rough Morse #2 taper gage, finish Morse #2 taper gage.

Operations:

1. Select stock.
  2. Locate center of ends. I.S. 1.
  3. Center drill ends. I.S. 2.
  4. Attach dog and mount work on centers. I.S. 3.
  5. Square ends, obtaining proper length. I.S. 4.
  6. Re-drill center holes to proper size. I.S. 2.
  7. Set lathe RPM for proper turning speed. I.S. 8.
  8. Straight turn between centers to obtain cylinder. I.S. 6.
  9. Check and adjust for taper elimination. I.S. 7.
  10. Rough turn entire cylinder. I.S. 6.
  11. Finish turn entire cylinder to size. I.S. 6.
  12. Read 'Taper Turning in General'. I.S. 33.
  13. Lay off distance for taper. I.S. 9.
  14. Calculate amount of offset for required taper. I.S. 34.
  15. Offset tailstock. I.S. 3, 34.
  16. Partially turn and check for proper amount of taper using taper gage. I.S. 40.
  17. Adjust, take light cut and recheck taper with gage. I.S. 40.
  18. Finish turn taper to fit gage, making sure of proper length of taper. I.S. 6, 40.
- Note: Length of taper is measured to bottom of chamfered shoulder. Tapered piece must enter gage until small end is flush with end of gage.
19. Reset tailstock on center. I.S. 34.
  20. Straight turn small end of piece. I.S. 6.
  21. Turn 45° chamfers. I.S. 23.
  22. Break all sharp corners with file. I.S. 11.
  23. Have instructor check work.



Fig. 78. Wood lathe live center blank.

## INSTRUCTION SHEET FOR WOOD LATHE LIVE CENTER BLANK

## Checking a Taper With a Taper Gage

(Instruction Sheet 40)

It is necessary always to test the amount of taper before turning the work to size because the lathe centers enter the work a short distance and because of the possibilities of other errors.

Taper socket gages are available for certain standard taper shanks. These gages measure the amount of taper and diameters. The distance to which the work enters the gage indicates whether or not it is machined to the correct diameter.

In order to check a taper, draw three light chalk lines about equidistant along the length of the tapered work (Fig. 79).

Insert the turned taper into the taper gage or the part to which it is to be fitted. Turn the tapered piece in the gage while adding a slight pressure, then withdraw from gage. Note whether the chalk marks have been rubbed off evenly along the length of the taper, or whether the contact is more pronounced at one of the ends.

If the contact is more pronounced on the smaller end of the taper the amount of taper is insufficient, as indicated at B in Fig. 80. If the contact is at the large end, the piece

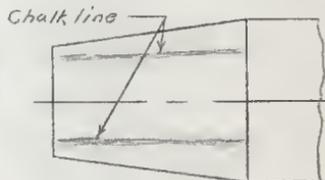


Fig. 79. Chalking a taper.

tapers too much (C in Fig. 80). At A in Fig. 80 is shown a correct taper.

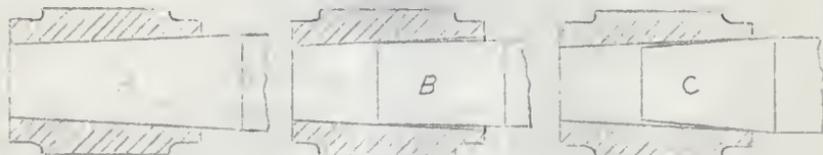


Fig. 80. Checking a taper.

Reduce the diameter on the end which bears too heavily by changing the tailstock offset to compensate for the error. Recheck with the gage and correct the offset until the taper fits the gage evenly (Fig. 80, A).

When the taper has been turned to the proper diameter, the tapered piece enters the taper gage until the small end is even with the smaller hole of the gage.

## OPERATION SHEET FOR A V DRILL BLOCK

Objectives:

1. Horizontal plane on shaper.
2. Vertical plane on shaper.
3. Angular plane on shaper.

Tools and Equipment:

Shaper, parting tool, parallels, turning tools, file, combination square, soft hammer, calipers, tool holders.

Operations:

1. Select cast iron V block blank.
2. Read 'The Shaper in General'. I.S. 41.
3. Arrange vise so that jaws are parallel to stroke.
4. Clamp V block in vise with bottom up. I.S. 42.
5. Adjust end position of ram. I.S. 43.
6. Mount cutting tool on shaper. I.S. 44.
7. Arrange feed for roughing cut. I.S. 45.
8. Rough plane bottom of block. I.S. 44, 45.
9. Finish plane bottom of block. I.S. 45.
10. Remove block from vise and re-clamp with a side up, using round against movable jaw. I.S. 42.
11. Rough plane first side. I.S. 45.
12. Finish plane first side. I.S. 45.
13. Rough and finish plane second side. I.S. 45.
14. Re-clamp block with V up. I.S. 42.
15. Rough and finish plane two top surfaces. I.S. 45.
16. With parting tool in center of block, cut relief slot at bottom of V. Use vertical feed.
17. Arrange swivel head and tool for angular cut. I.S. 44.
18. Rough plane first angular surface. I.S. 44.
19. Rough plane second angular surface. I.S. 44.  
Note: Two top surfaces of block must be same width.
20. Finish plane both angular surfaces, making certain that two sides of V make an angle of  $90^\circ$ . I.S. 44, 45.
21. Turn vise  $90^\circ$  and clamp block so one end projects outside of jaws. I.S. 42.
22. Arrange tool and swivel head for vertical cut. I.S. 44.
23. Rough and finish plane end, making certain it is at right angles with other surfaces. I.S. 44, 45.
24. Rough and finish plane other end making sure of squareness. I.S. 44, 45.
25. Break all sharp corners with file.
26. Have instructor check work.

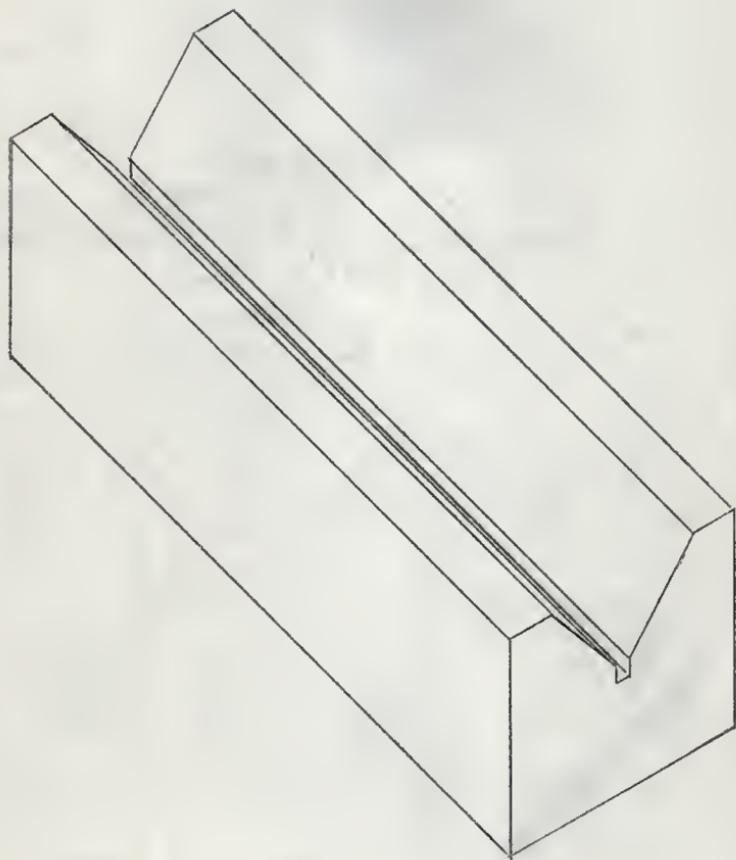


Fig. 81. V block.

## INSTRUCTION SHEETS FOR A V DRILL BLOCK

## The Shaper in General

(Instruction Sheet 41)

The function of the shaper is, primarily, to produce flat surfaces. The work is held on an adjustable worktable or more often in a vise fastened to the worktable, while the cutting tool, which is given a reciprocating motion, that is, caused to move forward and backward, peels off a chip on the cutting stroke. During the return stroke the feed operates to move the table and work the desired amount for the next cut.

Shapers are classified as to size, as a 14-inch or 20-inch shaper. The size is determined by the maximum length of the cut that may be taken and a standard shaper of a given size will hold and plane a cube of that size.

The shaper is especially adapted to small work which may be held in a vise bolted to the worktable. The tool head is so constructed as to permit horizontal, vertical, or angular cuts being taken. For toolroom work such as punch and die work, jig and fixture parts, and on short work for other special tools or machines, the shaper is practically indispensable.

The shaper cutting tool is easily ground to the desired shape for the cut to be taken and when dull may be quickly sharpened. Shaper cutting tools are ground with much less clearance behind the cutting edge than is the case of turning tools. Shaper tools should have a front clearance of about 7°.

## Clamping Stock in Shaper Vise

(Instruction Sheet 42)

Most shaper work is held in a vise which is bolted to the top of the table. However, the vise may be removed and work which is too large or otherwise impracticable to hold in the vise may be bolted to the top or side of the table, or to an angle plate or any special plate or other holding device fastened to the table.

The body of the vise may be swiveled on the base plate to any desired angle, the graduations in degrees showing the angular setting. This swivel feature is often useful for beveling ends, planing adjacent faces at other than  $90^\circ$ , but most of the work is done with the vise jaws either parallel with or at right angles to the direction of the cut. It is quicker to plane a surface cutting in line with its length than in line with its width, although the work is far more rigid when being cut against the vise jaw; but the friction between the work and the jaws is usually sufficient to hold the work when cutting lengthwise.

Often in clamping work in the vise it is necessary to use parallels under the block (Fig. 82). This may be essential for two reasons: to allow space to measure the work and to raise the top of work above the top of the vise jaws.

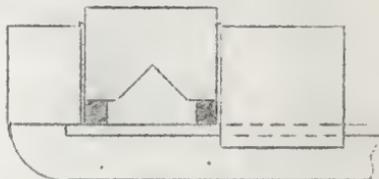


Fig. 82. Clamping in shaper vise.

The surfaces of rough castings are very hard and irregular and if placed in the vise they will mark the faces of the vise jaws and make them rough and inaccurate for fine work. It is advisable when the work is rough to place a piece of cardboard between the work and the face of the vise. The cardboard holds the work securely and forms a cushion for the irregularities on the work, enabling the vise to exert a pressure over a greater area.

After planing the first side of the block it is removed from the vise. In order to plane the second side, the block is placed in the vise with the planed bottom next to the fixed jaw. Parallels are used if necessary.

To prevent the work from lifting when being tightened in the vise, a round bar is sometimes used between the movable jaw and the work so that when pressure is applied the bar will roll against the work (Fig. 83).

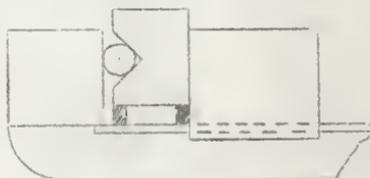


Fig. 83. Clamping in shaper vise.

## Adjusting Stroke and Positioning Shaper Ram (Instruction Sheet 43)

When planing on a shaper it is necessary that the length of stroke be adjusted so that time will not be wasted due to the tool being idle a portion of the cutting stroke.

To determine the length of stroke necessary, measure the work parallel to the stroke and add one inch.

On the right-hand side of some shapers and on top of the ram on others is a scale for setting the machine for length of stroke.

Figure 84 shows the scale as found on the right-hand side of some shapers. To adjust the stroke, loosen the locknut, place the wrench on the second nut and turn until the correct reading in inches registers at zero. Tighten the locknut.

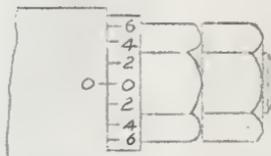


Fig. 84. Stroke indicator.

The stroke may not be in the proper place for the work after the correct length is obtained. Before positioning the stroke, a tool holder and tool should be clamped in position. By means of the belt move the ram to the extreme rear end of the stroke, that is, the beginning of the stroke. Loosen the clamp on the top of the ram and push or pull ram until the tool comes to within a half or three-fourths of an inch from the end of the work. Tighten the clamp securely to prevent the shifting of ram position when a cut is taken.

Tool Positions for Various Shaper Cuts  
(Instruction Sheet 44)

Shaper tools include various types of tools, some forged from solid steel, or tools ground to various shapes and used in various types of toolholders. The proper setting of the toolholder while cutting is important if good work is to be produced.

The toolholder should be placed in a position so that the cutting edge is not directly beneath the pivot point of the toolholder which is at the center of the tool post (Fig. 85). If by any chance the tool moves owing to the pressure of the cut, it will move away from the surface instead of under cutting. Do not allow the cutting edge to project too far from the tool post, that is, catch it short and clamp it tight. Be sure the toolhead slide is not run down too far by the down feed screw as this causes weakness and undue strain. Raise the table if necessary.

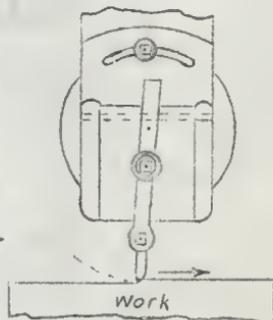


Fig. 85. Setting tool in shaper head.

When planing a horizontal surface the tool is adjusted to take the depth of cut desired by means of the downfeed screw, and the work is fed horizontally by hand until the cut is started, then the power feed is engaged.

To plane cast iron set the tool to take a chip deep enough to get under the scale. Should a portion of the surface be

low and the tool rubs on the scale, another cut should be started provided the cut will not make the work undereize.

When making vertical or angular cuts it is necessary to use an angled toolholder. When making either of these cuts it is very necessary to swivel the apron. Figure 86 illustrates the set-up for a vertical cut. When the top of the apron is moved in a direction away from the surface of the cut, the tool block and tool will hinge in a direction up and away from the work during the return stroke.

The entire head is swiveled over to the number of degrees required when making an angular cut. As in the vertical cut, the apron is swiveled away from the surface of the cut (Fig. 87).

When the head is set over for an angular or vertical cut and the tool slide fed down too far there is danger of the ram striking the work. Care must be exercised when setting up to have the slide high enough at the start to prevent any interference with the ram as the cut proceeds.

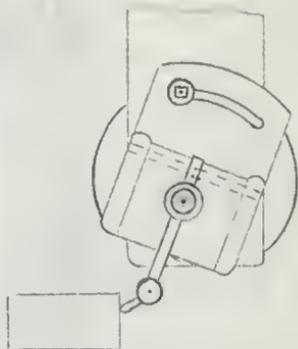


Fig. 86. Setting for vertical cut.

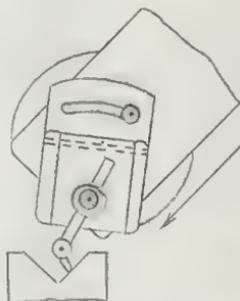


Fig. 87. Setting for angular cut.

**Arranging Feed for Various Shaper Cuts**  
(Instruction Sheet 45)

The shaper is designed to move slowly on the forward or cutting stroke and quickly on the return or non-cutting stroke. The feed should operate when the tool is on the return stroke; otherwise, the feed mechanism may be damaged.

When planing steel the direction of the feed is governed by the way in which the tool is ground. Never feed the work against the rake of the tool. In cast iron planing a tool with no rake is used and the feed may be in either direction.

If the feed occurs while the tool is cutting the feed mechanism would be operating under an unnecessary load. Therefore, the feed should always occur on the back stroke of the ram.

Figure 88 shows the mechanism for feeding and timing the feed. A rocking lever or reversing mechanism is operated from

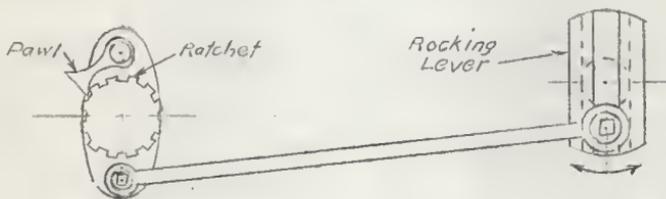


Fig. 88. Shaper feed mechanism.

the center of the crank rotation and rocks back and forth about its center at the same time that the ram moves backwards and

forwards. A pawl which operates the feed ratchet is connected by a rod to the rocking lever.

The amount of feed is governed by the distance the rod is off center of the rocking lever.

Timing the feed is accomplished by moving the rod to one side or the other of the center. The pawl and ratchet must advance the table and work when the ram is on the back stroke. Observation before starting the cut will determine whether or not the shaper is timed correctly.

For a roughing cut the connecting rod is set far enough off center so that the pawl will move the ratchet a distance of two or three notches. Ordinarily a one-notch feed is used for a finishing cut.

The direction of the movement of the table and work is changed by simply throwing the pawl over to the other side of the ratchet (Fig. 88).

## SUMMARY

There is here presented a set of operation and instruction sheets to facilitate the teaching of the elementary principles of machins shop practice with a view of:

1. Giving a general knowledge of common machines used in commercial production.
2. Developing skill and precision used in factory mass production.
3. Imparting economical techniques in machins production.
4. Acquiring knowledge and skill in selecting materials.
5. Developing effective techniques in shop personnel management.

## ACKNOWLEDGMENT

Indebtedness is acknowledged to Dr. V. L. Strickland, Professor of Education, and to Mr. W. W. Carlson, Professor and Head of Shop Practice, for their valuable suggestions and criticisms throughout the study, and to Dr. J. E. Ackert, Dean of the Graduate School, for his aid in preparing the manuscript.

## BIBLIOGRAPHY

1. American Technical Society.  
Lathe job training units. Dunwoody series. Machine shop training jobs. Chicago. Amer. Tech. Soc. 143 p. 1942.
2. American Technical Society.  
Shaper and planer job training units. Dunwoody series. Machine shop training jobs. Chicago. Amer. Tech. Soc. 115 p. 1942.
3. Brennan, John B.  
Can we improve our instruction sheets? Indus. Arts and Vocat. Ed. Mag. 26:87-89. March, 1937.
4. Burghardt, Henry D.  
Machine tool operation, Vol. I. New York. McGraw-Hill. 390 p. c. 1936.
5. Burghardt, Henry D.  
Machine tool operation, Vol. II. New York. McGraw-Hill. 400 p. c. 1937.
6. Ewing, Claude H.  
Should only individual instruction be used? Indus. Arts and Vocat. Ed. Mag. 30:16-17. January, 1941.
7. Fryklund, Verne C.  
Factors affecting the use of instruction sheets. Indus. Arts Mag. 16:241. July, 1927.
8. Fryklund, Verne C.  
The instructional unit. Indus. Arts and Vocat. Ed. Mag. 22:201-203. June, 1933.
9. Henig, Max S.  
Individualizing the instruction. Indus. Arts and Vocat. Ed. Mag. 24:325-326. November, 1935.
10. Jones, Harry A.  
Machine shop practice, Vol. I. Milwaukee. Bruce Pub. Co. 167 p. c. 1926.
11. Jones, Harry A.  
Machine shop practice, Vol. II. Milwaukee. Bruce Pub. Co. 207 p. c. 1930.
12. Landis, Russell H.  
Instruction sheet rating scale. Indus. Arts and Vocat. Ed. Mag. 24:203-204. July, 1935.

13. Le Blond Machine Tool Company.  
A manual of lathe operations and maintenance of a modern geared head engine lathe. Cincinnati. The R. K. Le Blond Machine Tool Co. 116 p. c. 1940.
14. New York (State) University. State Education Department. Bureau of industrial and technical education.  
Suggested job sheet units for beginners in machine shop practice; vocational education program for national defense industries. New York. New York (State) University. State education department. 50 p. 1940.
15. New York (State) University. State Education Department. Bureau of industrial and technical education.  
Suggested unit course in lathe work for beginners in machine shop practice; vocational education program for national defense industries. New York. New York (State) University. State education department. 164 p. 1941.
16. Phillips, Ernest C.  
Job sheets and operation sheets as teaching devices. Indus. Arts Mag. 14:321-325. September, 1925.
17. Smith, Robert E.  
Information and operation units in machine woodwork. Wichita, Kansas. McCormick-Mathers. 159 p. 1938.
18. Smith, Robert H.  
Text book of advanced machine work. Boston. Industrial Education Book Co. 1619 p. c. 1930.
19. South Bend Lathe Works.  
How to run a lathe. South Bend, Indiana. South Bend Lathe Works. 126 p. c. 1939.
20. Turner, Frederick W., Ferrigo, Oscar E., and Bertrand, Aldrick.  
Machine shop work. Chicago. Amer. Tech. Soc. 400 p. 1941.
21. Van Westrienen, Harold J.  
Preparation and use of lesson sheets. Indus. Arts and Vocat. Ed. Mag. 20:237-242. July, 1931.
22. Williams, Burton T.  
Individual instruction sheets. Indus. Arts and Vocat. Ed. Mag. 26:86-87. March, 1937.