CHAPTER 2

# Shaper Work

**Shaper Cutting Tools.** The variety of cuts that may be made in a shaper on any of the metals used in machine work calls for tools of various shapes. Shaping, that is, cutting on a shaper, can be done to the right or to the left. It also includes roughing cuts, finishing cuts, slotting, contouring, undercutting, dovetailing, and a variety of operations. Tools are ground differently for the different cutting operations. Tools can be made from solid bars of steel or they may be made from smaller pieces of tool steel, called *bits*, which are ground to the desired shape and held by being clamped in a toolholder. The large, solid tools are especially good for very heavy work because they carry away the heat from the cutting edge of the tool more rapidly. There are also toolholders using forged bits. The toolholder with the ground bit is probably the most popular combination on a shaper.

The shape of the cutting tool varies with the character of the work. The general shapes of shaper-cutting tools for shaping cast iron and mild steel as recommended by The Cincinnati Shaper Company are shown in Figs. 2-1 and 2-2. Study these shapes very carefully. It will prove worth while during your apprenticeship as well as for later work on the shaper.

The shape of the tool is also determined by the type of work that is to be done. For the production of an ordinary flat surface, the tool is either right-hand or left-hand. The left-hand tool is more common because it permits the operator to see the cut better than the right-hand tool does. A dovetailing tool is, naturally, quite pointed. A finishing tool is the reverse, because a broad-nosed or square-nosed tool will largely eliminate feed marks, whereas feed marks will be more noticeable with a pointed tool.

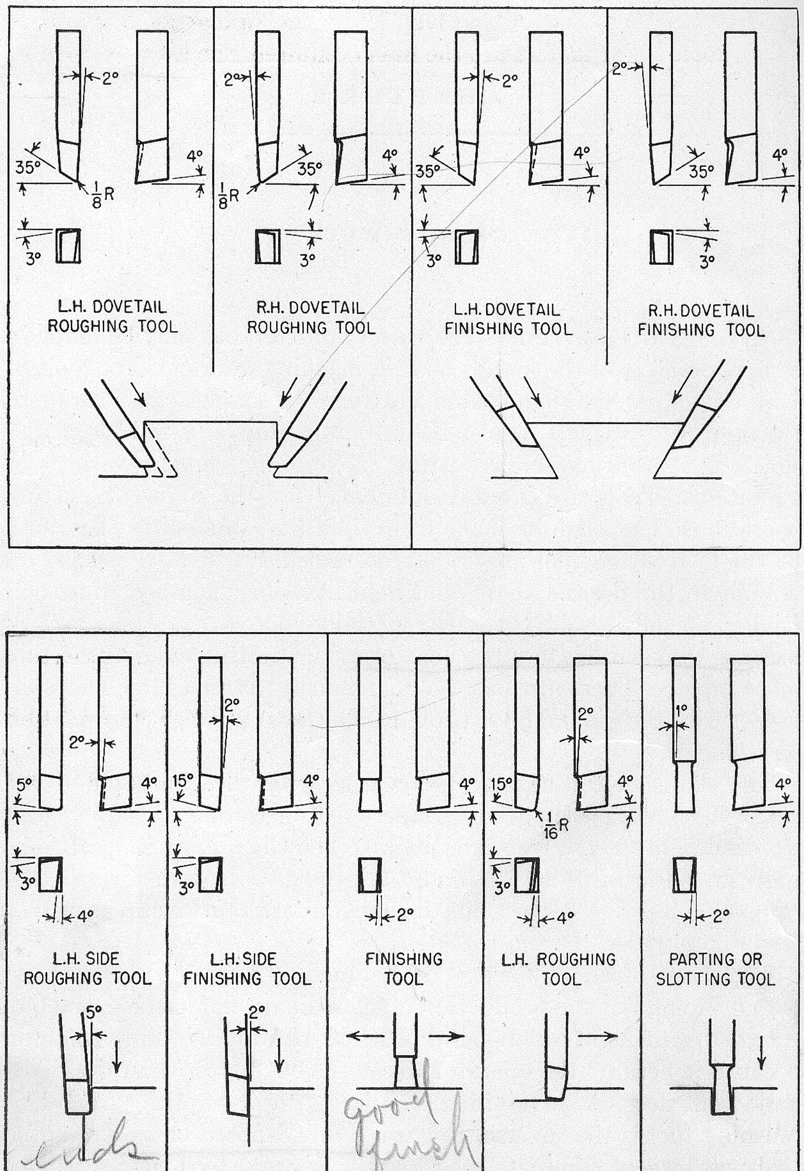



**Fig. 2-1.** Shaper tools for cutting cast iron. (*The Cincinnati Shaper Company*)

There are other factors that help in the determination of the shape of the tool. These factors are the finish required, the kind of material

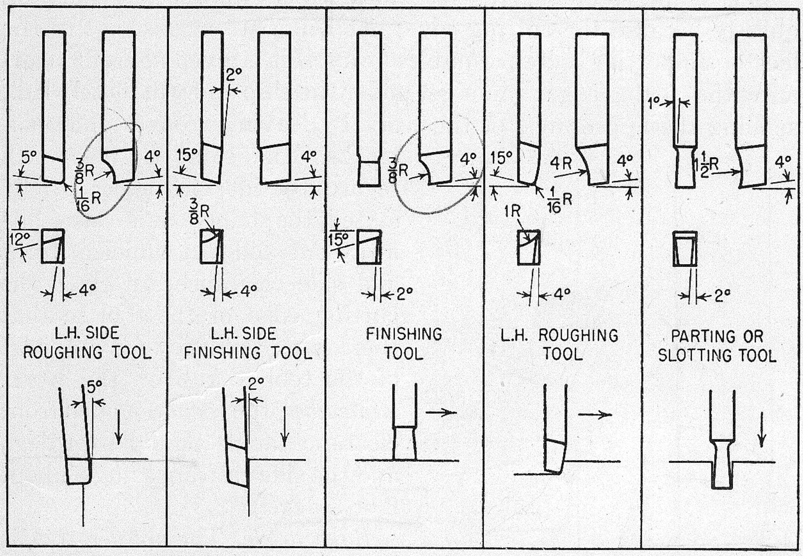



**Fig. 2-2.** Shaper tools for cutting mild steel. (*The Cincinnati Shaper Company*)

being cut, and the condition of the machine, as well as feed and speed.

The elements of a shaper tool or a planer tool, that is, the front rake, front clearance, side rake, etc., are in the same relative positions as on the lathe tool, regardless of the fact that the shaper tool when in use is held vertically, while the lathe tool is held horizontally.

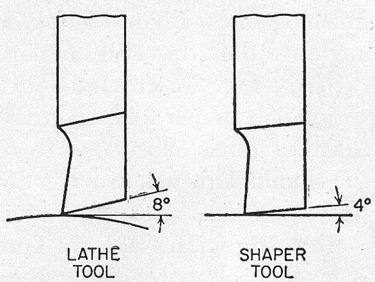



**Fig. 2-3.** Comparison of front clearance angles of lathe and shaper tool bits.

*Clearance Angles.* There is no rocker in the tool posts of the shaper, hence the tool cannot be adjusted for clearance; the proper clearance angles must be ground on the tool. As shown in Fig. 2-3, the front clearance angle is 4 deg.

Since the shaper feed does not operate during the cut as does the lathe feed, a side clearance of 2 or 3 deg. is sufficient.

If a shaper tool is given *too much* front clearance, it will dull quickly because the cutting edge, not being strong because of the lack of supporting metal, crumbles away; if it is given *no* front clearance, the cutting edge cannot get under the chip and will merely rub, spoiling the appearance of the work by leaving grooves and toolmarks. The same is true with regard to side clearance. Briefly stated, the shaper cut is a straightaway cut and just sufficient front and side clearance are given the cutting edge of the tool so that there is no tendency for any part of the tool to rub on the work. *Remember*, the recommended front clearance angle should be 4 deg., and the side clearance angle about 2 or 3 deg.

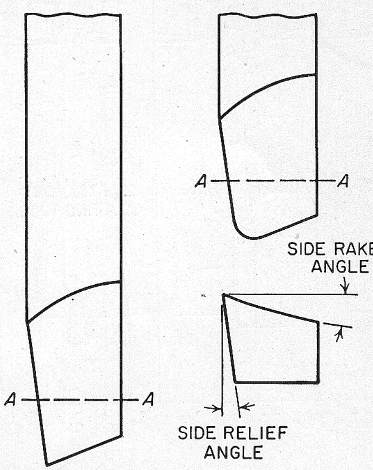



**Fig. 2-4.** Rake angle and side relief angle on shaper tools. *A–A* are cross sections of the tools.

*Rake angle.* The shaper tool is usually given side rake of 10 deg. or more, depending on the kind of tool and on the hardness of the metal to be machined, but no front rake is given except on finishing tools. Figure 2-4 shows a side-rake angle and a side-relief angle on the cross section *A-A* of the tools shown directly above.

Study Fig. 2-5 carefully for a simple explanation of the cutting action of a shaper tool when a plane surface is being machined. Note that the tool is offset so as to get the tool point toward the center of the shank. This will prevent the tool from digging. Most shaper and planer manufacturers recommend this type of tool for general work.

**Shaping with Carbide Tools.** Almost any type of material that is machinable with high-speed-steel cutting tools can be economically machined with carbide tools. In situations where the life of the tool is short, as for machining chilled cast iron, die steels, etc., the carbide tool is more efficient and economical.

In order that a shaper may be suitable for carbide shaping, it must be capable of speeds exceeding 100 ft. per min. because it has been determined by experiment and actual industrial conditions that 100 ft. per min. is the absolute minimum speed at which carbides can be economically used. At slower speeds, there is no appreciable difference as to cost of operation between the high-speed tools and the carbide tools.

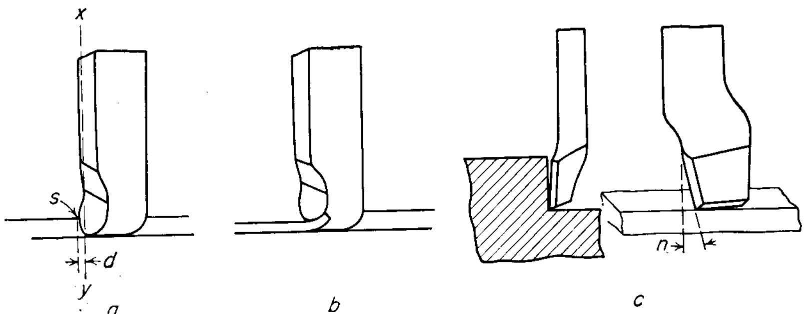



**Fig. 2-5.** Cutting action of tool when machining a plane surface. Note in view (*a*) the line *xy* is parallel to the base of the tool and therefore the tool has no front rake. Note also that since the cutting edge is given side rake, the start of the cut is made at *s* and the distance *d* is traversed before the full depth of the cut is taken; thus the tool enters the work gradually and prevents the shock of having the full cutting edge strike the metal at once. The way in which the chip curves is shown in (*b*). In (*c*) is shown a side tool with the cutting edge ground flat to take a finishing cut, say, ¼ in. or more wide. Note the angle of shear *n* to give an easy start and finish of the cut. A side tool, ground in this way, is much used in shaper and planer work for finishing cast iron.

The shaper must also be capable of producing uniform feeds. This condition must be met; if it is not, excessive tool wear will result because of the varying feeds between strokes.

The shaper must be equipped with a good tool lifter (Fig. 2-6). Sintered tools will not stand up if allowed to drag on the return stroke. On account of the high speed of the machine, it is almost impossible for the operator to lift the clapper box manually on the return stroke of the ram; the tool lifter does exactly that.

The shaper must be in excellent operating condition to ensure accurate work without chatter marks and, at the same time, guarantee long tool life.

The motor should be capable of supplying the needed horsepower when carbide tools are used, because the use of such cutting tools demands greater power. The horsepower requirements will vary in direct proportion to the speed of the machine. Shaping at 150 ft. per min. requires $1\frac{1}{2}$ times the horsepower used at 100 ft. per min.

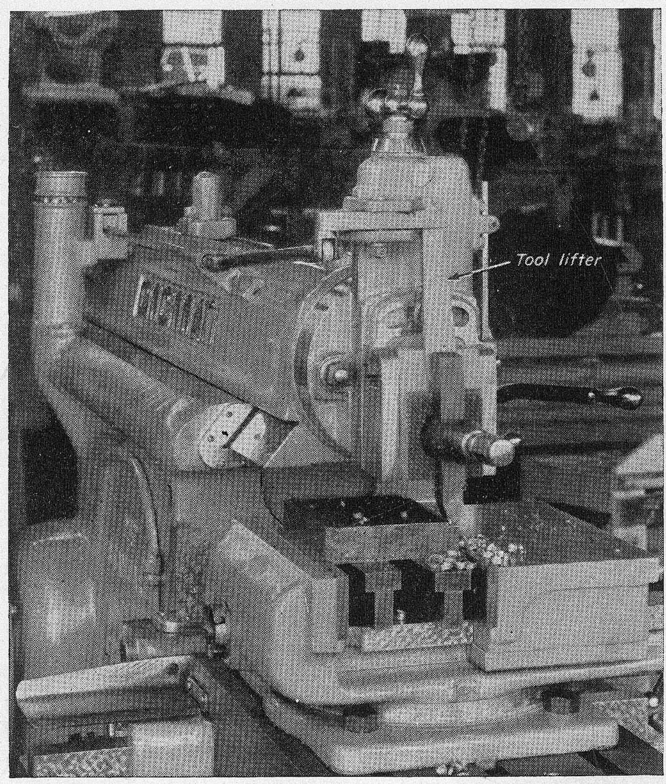



**Fig. 2-6.** A tool lifter attached to a shaper. (*The Cincinnati Shaper Company*)

**Right-hand and Left-hand Tools.** When a job is being set up in any machine, it is best, if possible, to arrange the work and also the tool in such a way that the operator can readily see the cut from his normal position at the machine, that is, from the position in which he controls the machine. For this reason it is customary when taking a horizontal cut on the shaper or planer to start the cut on the side toward the operator and, when shoulder or similar cuts are to be made, to arrange the work so that these cuts will come on this side. Many shaper jobs, however, include tongues, grooves, and angles which involve cuts on both sides of the work. Since in work of that kind it often makes for greater accuracy and speed to *machine in one setting of the work all the surfaces possible*, it is necessary to have right-hand and left-hand cutting tools. The terms *right-hand* and *left-hand* as applied to shaper or planer tools are derived from lathe tools of similar shape; that is, a right-hand cutting tool is one that cuts from right to left and a left-hand cutting tool is one that cuts from left to right.

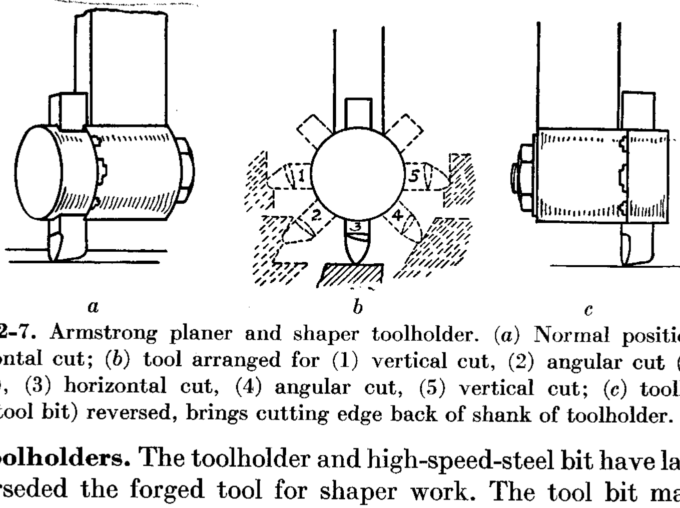



**Fig. 2-7.** Armstrong planer and shaper toolholder. (*a*) Normal position for horizontal cut; (*b*) tool arranged for (1) vertical cut, (2) angular cut (inside angle), (3) horizontal cut, (4) angular cut, (5) vertical cut; (*c*) toolholder (and tool bit) reversed, brings cutting edge back of shank of toolholder.

**Toolholders.** The toolholder and high-speed-steel bit have largely superseded the forged tool for shaper work. The tool bit may be ground to the shape required to accomplish the desired result for practically any operation. Figure 2-7 shows a patented toolholder (Armstrong) which, in the smaller size, is used for shaper work and, in the larger size, is very efficient for use in the planer. The construction of this toolholder permits the tool bit to be securely and rigidly held in any one of the five positions shown in *b*, so that horizontal, vertical, or angular cuts, either right-hand or left-hand, may be made. Another advantage of this toolholder lies in the fact that for heavy cuts the toolholder may be reversed in the tool post (and, of course, the tool bit is also reversed, [*c*, Fig. 2-7]). Since the cutting edge is then back of the shank of the tool, the tendency of the tool to chatter or to "dig in" is eliminated. In any case the tool bit should not be allowed to project too far as this will result in unnecessary spring.

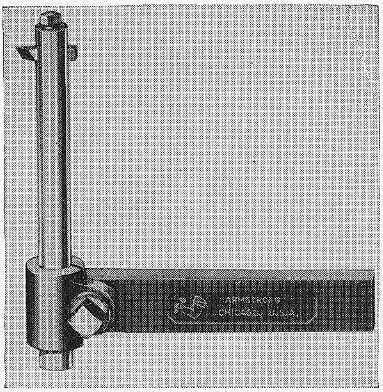



**Fig. 2-8.** Another type of shaper toolholder. (*Armstrong Bros. Tool Company*)

*The lathe-turning toolholder and bit* can be used as a shaper tool, provided the tool bit is not given too much clearance, especially too much front clearance. Of course, the position of the bit set at an angle of 20 deg. with the shank of the tool gives a front rake which, while not desirable, is not prohibitive for light cuts.

Other types of toolholders are used in shaper work, and some of them are shown in Figs. 2-8, 2-9, and 2-10. The toolholder shown in Fig. 2-8 should remind you

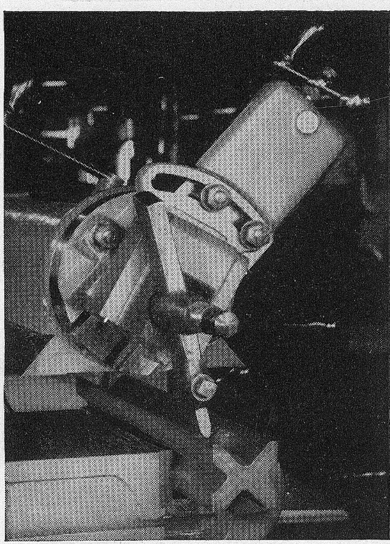


**Fig. 2-9.** A large lathe-type toolholder being used in a shaper. (*The Cincinnati Shaper Company*)

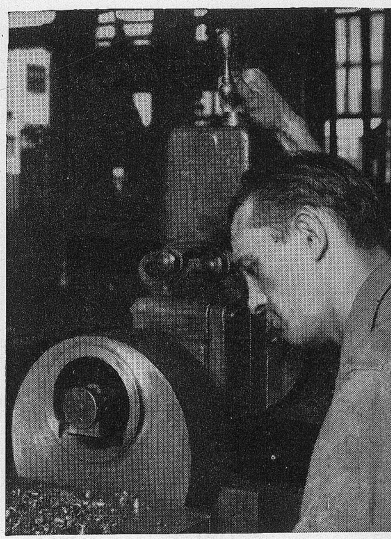


**Fig. 2-10.** Cutting a keyway. (*The Cincinnati Shaper Company*)

of the boring bar used in the lathe. This particular type of holder is used whenever it is necessary to hold the tool far away (comparatively speaking) from the tool post. It is also used whenever the shape of the piece to be machined is such that the cutting must take place in a part of the work away from the tool post. Figure 2-9 shows a large Armstrong holder being used. This is similar to the lathe toolholder but is larger and, most important, the toolbit is *not* at an angle to the holder but parallel to it. Figure 2-10 shows a toolholder similar to that shown in Fig. 2-7 but heavier in construction. The arm is also longer and heavier. With this toolholder, the clapper box must be locked in position.

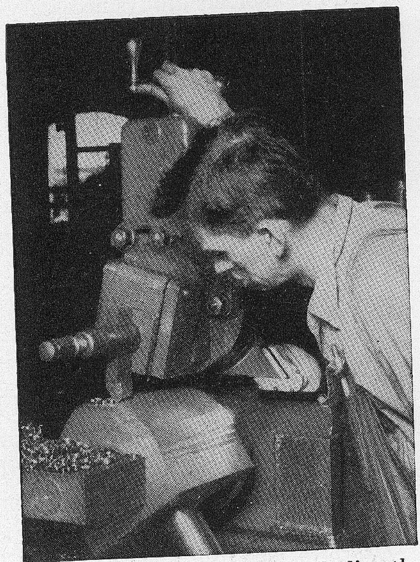


**Fig. 2-11.** Tool bit clamped directly to tool post in shaper. (*The Cincinnati Shaper Company*)

There are many instances in which a toolholder is not used. Figure 2-11 shows such an instance. Here the shaper operator is using a solid cutting tool held directly in the tool post. The tool is strong and heavy enough so that no toolholder is necessary. This tool is commonly used for very heavy work.

## QUESTIONS ON SHAPER CUTTING TOOLS

1. Why is it necessary to have many different-shaped tools for work on the shaper?
2. Why is it necessary, at times, to use forged tools?
3. What factors determine the shape of a cutting tool to be used in the shaper?
4. What size of angle is recommended for the front clearance? for the side clearance?
5. What would happen to a tool with a front clearance of 15 deg.?
6. Name some important considerations that must be given when a carbide-tipped tool is being used in a shaper.

7. What is a tool lifter? How does it help the tool?
8. What is meant by a right-hand and a left-hand tool?
9. How may tools be held in the tool post of the shaper?
10. Are toolholders always necessary? State why.

**Speeds and Feeds.** The reason for machining metal parts (in any machine tool) is usually twofold: (1) to remove surplus metal to bring the work to a given size, and (2) to produce a smooth finish on the surface. To accomplish these results, at least two cuts, one or more roughing cuts and a finishing cut, are nearly always necessary. To operate the machine efficiently to produce these results means a reasonable understanding of the proper speed, feed, and depth of cut for roughing and also for finishing.

To understand the cutting speed is comparatively easy; it depends almost entirely on three things: (1) the kind of material being cut is a factor, the softer the material the faster the speed at which it may be cut; (2) the amount of material being removed in a given time is another factor, a light cut may usually be taken at a greater speed than a heavy cut; and (3) the kind of steel from which the cutting tool is made is a very important consideration because a high-speed tool will cut at least at double the speed of a carbon-steel tool.

*Depth of Cut and Feed.* The value of any machine tool depends upon its power, the strength and rigidity of its construction, the rapidity and smoothness of its action, its convenience in operation, and its accuracy. Modern shapers may be classed as particularly rugged machines, carefully designed and accurately built. *Whenever a considerable amount of metal must be removed, the shaper should be made to work during the roughing cut; that is, the cutting speed should be suitable and the depth of cut and feed should be proportioned to remove as big a chip as the shaper will drive, provided that the nature of the work, the way it is held, and the strength of the tool will permit.* This point is entirely basic. It should be emphasized. It applies to other machining as well, on other machine tools. You can "make time" in roughing only. Finishing cannot be hurried. It is impossible to give a rule for the depth of cut or the amount of feed, or for a proportion of feed and cut, but the following suggestions may help the beginner.

With a given tool and the given amount of metal to be removed per cut, a really coarse feed and less depth are not so efficient as a finer feed and a deeper cut for two reasons: (1) the thick chip does not curl so easily and takes more power, and (2) the tear in the metal is greater, thus producing a rougher surface. A safe rule to follow is to give as much feed as is consistent with the surface desired, and then all the depth of cut that the tool and the motor will stand, provided that amount of metal must be removed.

The following table of (average) cutting speeds and feeds of cutting tools made of carbon and high-speed steels is given for the convenience of the beginner.

**Table of Speeds and Feeds for Carbon and High-speed Tools**

| Cutting tool | Cast iron | | Machine steel | | Carbon steel | | Brass | |
|---|---|---|---|---|---|---|---|---|
| | Speed | Feed | Speed | Feed | Speed | Feed | Speed | Feed |
| High-speed steel | 60 | 0.085 | 80 | 0.060 | 50 | 0.050 | 160 | 0.050 |
| Carbon steel | 30 | 0.060 | 40 | 0.050 | 25 | 0.040 | 100 | 0.050 |

The usual practice is to run the shaper too slowly. It is well for the beginner to calculate the number of strokes necessary to give the proper cutting speed for the work at hand until he gets accustomed to seeing the shaper move fast enough.

For cutting with a carbide-tipped tool, the following chart shows the recommended feeds and speeds with relation to depth of cut as

**Chart of Feeds and Speeds for Carbide Tools**

| Material | Depth of cut, in. | Feed per stroke, in. | Maximum speed, ft. per min. |
|---|---|---|---|
| C.I., hard | 1/32–1/2 | 0.005–0.020 | 100 |
| C.I., soft | 1/32–1/2 | 0.005–0.020 | maximum available |
| Cast steel | 1/32–1/4 | 0.005–0.010 | 150 |
| S.A.E. 1020 steel | 1/32–7/16 | 0.010–0.015 | 100 |
| S.A.E. 1045 steel | 1/32–3/8 | 0.010–0.015 | 150 |
| S.A.E. 1080 steel | 1/32–1/4 | 0.005–0.020 | 150 |
| Die blocks | 1/32–1/4 | 0.005–0.020 | 150 |
| Brass, hard | 1/32–7/16 | 0.010–0.015 | 100 |

practiced in industry. Consult this chart whenever this type of cutting tool is being used.

Compare both charts and note the differences in speeds, feeds and depths of cut when the various types of cutting tools are being used. Mass production demands (in most cases) that the carbide-tipped tools be used.

*Cutting-speed Calculations.* The calculations for cutting speeds for shaper work are more involved than those for lathe or drill-press work, because the shaper cuts only during the forward stroke and, further, because the return stroke is faster than the cutting stroke.

Given the ratio of return-stroke *time* to cutting-stroke *time* as 2:3, the sum of the terms of the ratio equals 5; and $\frac{2}{5}$ of the time equals the *time* of the return stroke; and $\frac{3}{5}$ of the time equals the *time* of the cutting stroke.

Given the length of stroke in inches and the number of strokes per minute, their product gives the number of inches cut during one minute of the machine's operation. Since cutting speed is expressed in feet per minute, this must be multiplied by $\frac{1}{12}$ to convert the inches to feet. As noted above, the actual time of cutting this distance is $\frac{3}{5}$ min. Therefore, since distance divided by time equals rate, divide the distance (in feet) by $\frac{3}{5}$ (that is, multiply by $\frac{5}{3}$) and the result will be the cutting speed. Instead of multiplying in every problem first by $\frac{1}{12}$ and then by $\frac{5}{3}$, it will be quicker to multiply by 0.14 which amounts to the same thing ($\frac{1}{12} \times \frac{5}{3} = 0.14$ approximately). Hence the following:

RULE 1: To obtain cutting speed (CS), *number of cutting strokes* and *length of stroke* being given:

Multiply the number of strokes per minute ($N$) by the length of stroke in inches ($L$) and the product by 0.14.

FORMULA: $0.14NL = CS$

EXAMPLE: Length of stroke 8 in. Number of strokes per minute 30. What is the cutting speed?

SOLUTION: $8 \times 30 \times 0.14 = 33.6$ ft. per min.

RULE 2: To obtain the number of strokes necessary, required cutting speed and length of stroke being given:

Multiply the cutting speed by 7 and divide by the length of the stroke in inches.

FORMULA: $N = \frac{CS \times 7}{L}$

DERIVATION: From Rule I, $0.14NL = CS$, or

$$N = \frac{CS}{L \times 0.14} = \frac{CS}{L} \times \frac{1}{0.14} = \frac{CS \times 7.2}{L}$$

and for practical purposes $N = \frac{CS \times 7}{L}$ is near enough.

EXAMPLE: How many strokes are required to shape cast iron with a high-speed tool (60 ft. per min.), with a stroke of 5 in.?

SOLUTION: $\frac{60 \times 7}{5} = 84$ strokes.

## QUESTIONS ON SPEED, FEED, AND DEPTH OF CUT

1. The proper cutting speed for a given job depends on three factors. What are they? Give an example of each.
2. What is a safe proposition to follow concerning the feed and depth?
3. About what cutting speed will be practical to start with on cast iron? Is the tool you are to use carbon steel or high-speed steel?
4. May it possibly be wise before long to change to a faster speed? To a slower speed? Give reasons.
5. How many strokes per minute are necessary to give the required cutting speed?
6. Do you suppose a machinist would use a formula to calculate the number of strokes necessary? How would he go about it? Of what value is the formula to the beginner?

**Holding the Work.** Most shaper work is held in a vise which is bolted to the top of the shaper 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 the side of the table, or to an angle plate or any special plate or other holding device which can be fastened on the table.

*Shaper Vise.* Figure 2-12 shows a large-opening vise used in shaper operations. The principal parts are the base, the body, the fixed jaw, the movable jaw, the screw, and the plates which are attached to the face of the jaws. The base is bolted to the table of the shaper and is graduated through an arc of 180 deg., with a zero (0) position on the right and left sides and the 90-deg. mark in the front, midway between the two zero marks.

The body may be swiveled on the base plate to any angle desired, graduations in degrees showing the angular setting. This swivel feature is often useful for beveling ends, shaping adjacent sides or faces at other than 90 deg., etc., but most of the work is done with the vise jaws either parallel with or at right angles to the direction of the cut. Figure 2-13 shows a piece of work held at an angle to the direction of cut other than 90 deg.

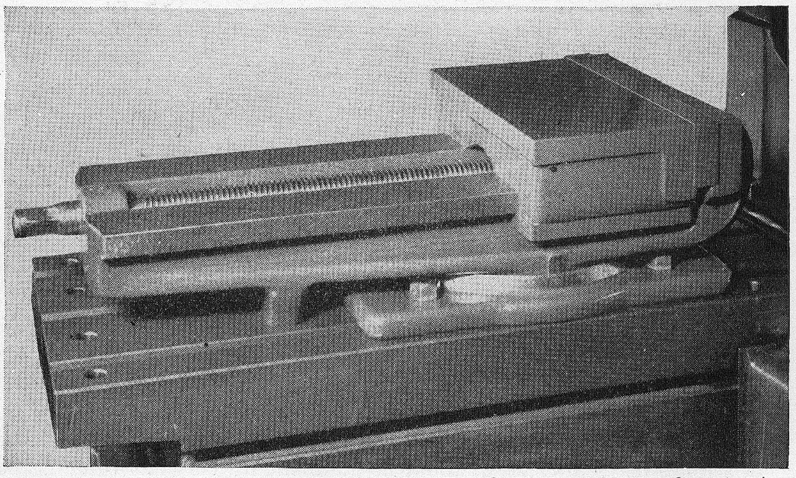


**Fig. 2-12.** A large-opening shaper vise. (*The Cincinnati Shaper Company*)

The shaper vise is especially strong, the jaws are long and deep, and the adjustment is sufficient to take work of a considerable width. The jaws are usually hardened to prevent them from being scored and dented.

There are shaper vises equipped with double screws rather than with a single screw. Some of these vises, in addition to holding straight work, are used to hold pieces having a slight taper. Figure 2-14 shows a vise with a double screw holding a tapered piece.

Figure 2-15 shows a side vise, (*a*) showing the side which holds the work and (*b*), the other side of the vise. This vise has many uses in machine-shop practice.

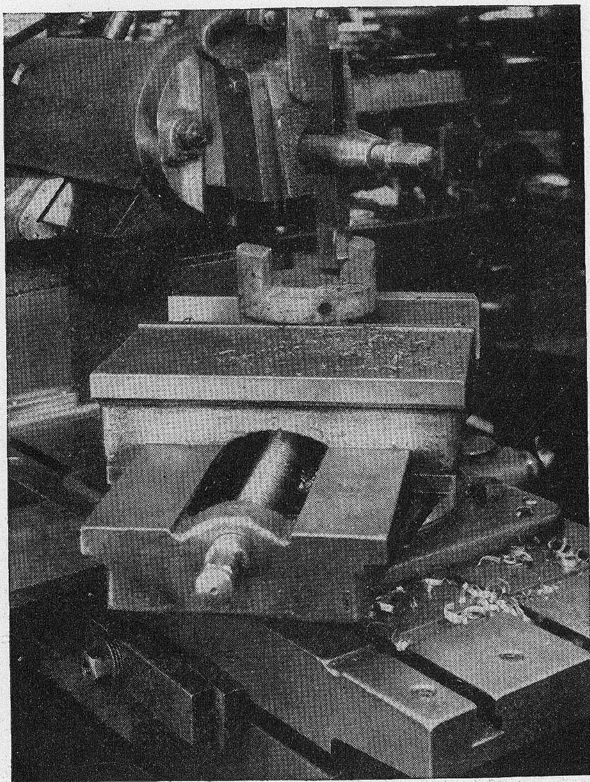


**Fig. 2-13.** Shaper vise set at an angle. (*The Cincinnati Shaper Company*)

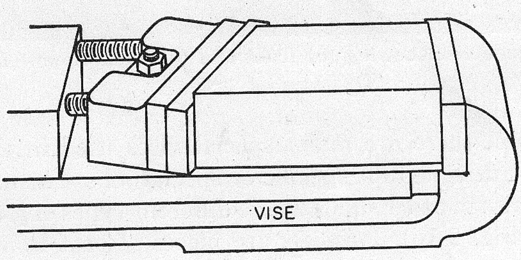



**Fig. 2-14.** Taper work with a double-screw vise. (*The Cincinnati Shaper Company*)

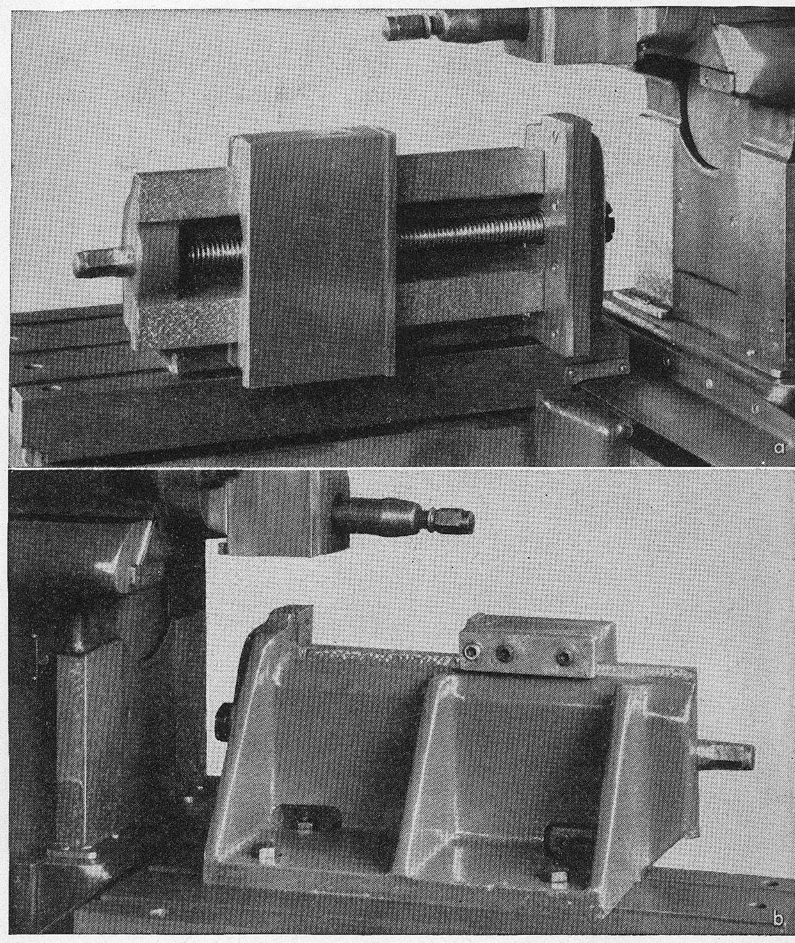



**Fig. 2-15.** A side vise showing (*a*) front and (*b*) back views. (*The Cincinnati Shaper Company*)

Then, again, there are many special vises used in the machine shop that are designed for specific or special jobs. Such a vise has a very limited use in other shops where different types of jobs are done.

*Parallels* (Fig. 2-16). Parallels are pieces of cast iron or steel of rectangular cross section, of considerable length in proportion to their width and thickness, with opposite sides parallel and adjacent sides square. They are used to raise the work to the required height in the vise or otherwise to bolster and level it. Parallels are made in pairs. Two or more pairs may often be used together.

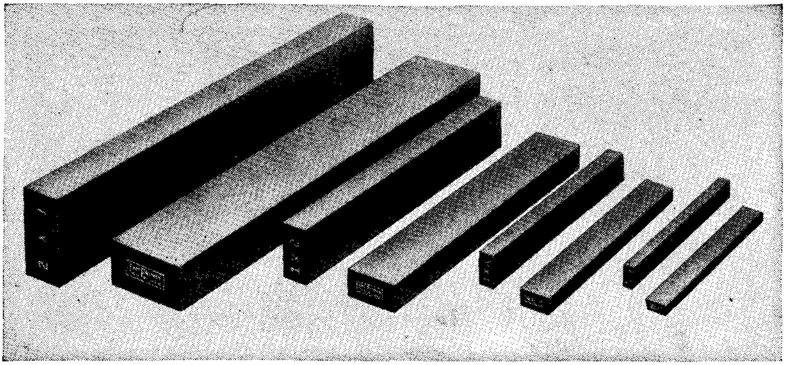


**Fig. 2-16.** Several sizes of parallels. Note sizes marked on edges. (*The Taft-Pierce Manufacturing Company*)

Another type of parallel is the *angular parallel* (Fig. 2-17). These parallels are similar to the regular parallels except that one side of each parallel is machined "out of square," with the adjacent side at a certain angle.

*Angle Plates* (Fig. 2-18). Angle plates are of any size required and are usually made of cast iron. An angle plate is composed of two members, or wings, the outer surfaces of which are machined flat at an angle of 90 deg. to each other. When in use, one surface is bolted to the table and the work is fastened to the other surface. Some angle plates have one of the inner surfaces finished, which permits work being clamped or bolted to this surface when desirable. Holes are drilled where necessary for the clamping bolts. Sometimes tapped holes are more convenient for the purpose of clamping the work. Often C clamps are used for this purpose.

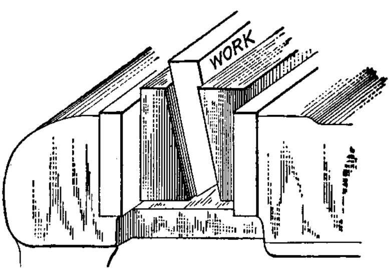



**Fig. 2-17.** Application of angular parallels.

*Hold-downs or Grippers* (Fig. 2-19). Hold-downs, or grippers, are thin pieces of approximately triangular cross section, of the length

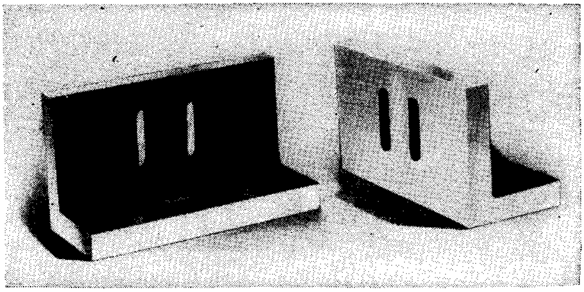


**Fig. 2-18.** Angle plates. (*The Taft-Pierce Manufacturing Company*)

desired (6 in. more or less), used most frequently to hold thin pieces in the vise. The narrow edge is rounded and the opposite edge is beveled about 2 deg. toward the bottom. This ensures the work's being held down on the bottom of the vise or on a parallel, as the case may be. Hold-downs are especially valuable when parallels of the required height to raise the thinner pieces just above the vise jaws are not available. They are very useful also when it is desired to finish only two opposite surfaces of a piece.

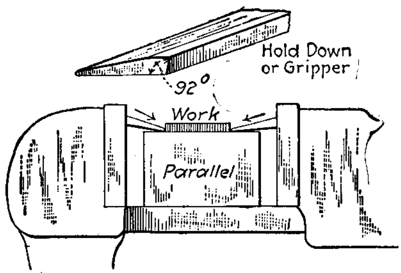



**Fig. 2-19.** Action of hold-downs.

*Toe Dogs* (Fig. 2-20). Toe dogs also are used to hold thin pieces in place. These dogs are held on the table and, with the help of two adjusting screws, the thin work is held in place.

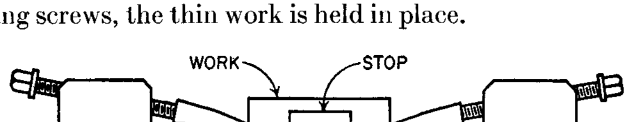



**Fig. 2-20.** Holding thin work with toe dogs. (*The Cincinnati Shaper Company*)

*Shaper-index Centers* (Fig. 2-21). Shaper-index centers are very useful for certain curved surfaces that are partially cylindrical but have projecting portions and consequently cannot be turned in a lathe. The splined shaft in the illustration is a good example of the use of index centers. They may often be used for finishing surfaces of pieces held on a mandrel more advantageously than the work

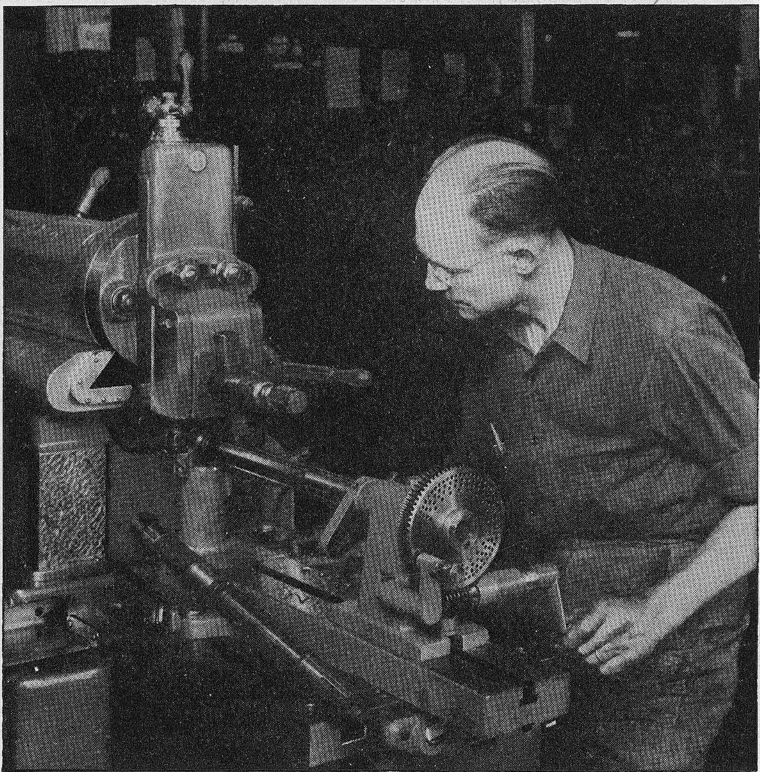


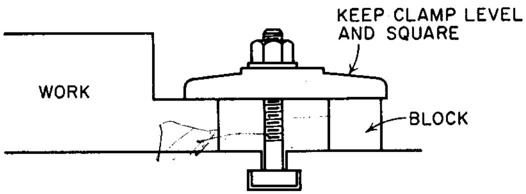
**Fig. 2-21.** Using index centers to cut splines. (*The Cincinnati Shaper Company*)

could be done in a milling machine. The construction of the index head and tailstock permits of a variety of indexing[1] operations.

Notice the various circles of holes in the plate that revolves on the worm. Notice, too, the pin that is used to fix the position of the plate. This procedure is further explained in Chapter 9 of this volume.

[1] Indexing is the method used in accurately dividing the circumference of a circle into any number of equal parts.

**Clamping Work to the Table.** Clamping work to the table properly is the first prerequisite for accurate work done on the shaper. If the work is *not* clamped properly, many things may happen. First of all, the work may move out of position and then it is usually spoiled; second, the tool may get caught in parts of the

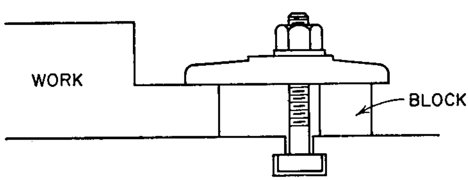



RIGHT

CLAMPING EFFECT IS ON WORK

**Fig. 2-22.** Correct method of clamping work to the table of a shaper. (*The Cincinnati Shaper Company*)

clamping devices and so it may be broken; third, somebody may get hurt when things start to fly. The object in proper clamping is to have the work absorb the clamping effect; that is, the pressure in clamping must be *on* the work, *not* on the block used in clamping. Remember this *always*.




WRONG

CLAMPING EFFECT IS ON BLOCK

**Fig. 2-23.** Incorrect way of clamping work to a shaper table. (*The Cincinnati Shaper Company*)

Figure 2-22 shows the proper way to clamp work to the shaper table and Fig. 2-23 shows an incorrect way to clamp work to the table. Note the positions of the work in the two illustrations. In the right way (Fig. 2-22), the clamp is set directly over the work and the bolt is in the middle of the setup; while in the wrong way of clamping (Fig. 2-23), the clamp is *not* directly above the work and the bolt is *not* in the middle of the setup. Clamping pressure must be evenly distributed over the work and block when clamped if it is to hold properly.

The description and use of some of the common clamps used in machine-shop work follows.

*Clamps*, or *straps*, as they are sometimes called, are designed to hold workpieces in place when they are being machined. These clamps are of various designs and shapes but all have the same function—to hold work in place.

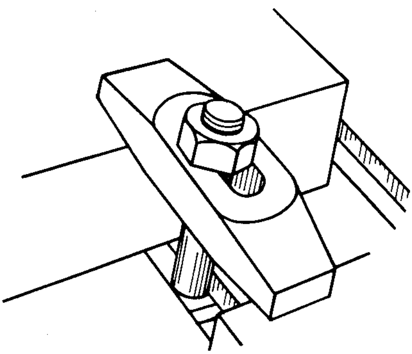


**Fig. 2-24.** The plain clamp.

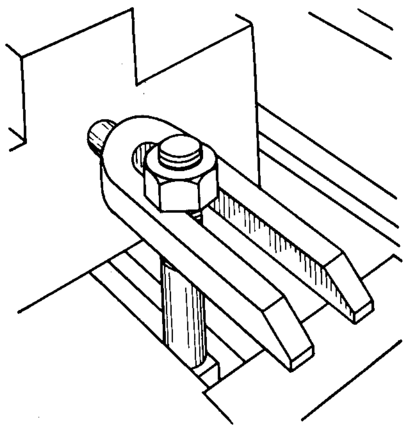


**Fig. 2-25.** The U clamp.

Some of the more common types of clamps and their uses are described here.

The *plain clamp* or *strap* (Fig. 2-24), which is strong, is used for general clamping purposes. It has an elongated slot through which the T bolt passes. The shape of the workpiece to be clamped must be such that the clamps will not interfere with the machining operation when the clamps are used.

The *U clamp* (Fig. 2-25), another type, is especially useful when the nut on the T bolt does not have to be removed. This is an advantage, in that the bolt may be set in the best position for clamping without having the nut removed from the bolt, and then the clamp is placed under the nut. Care must be taken not to exert great pressure on this type of clamp, because it has a tendency to spread and bend under such conditions.

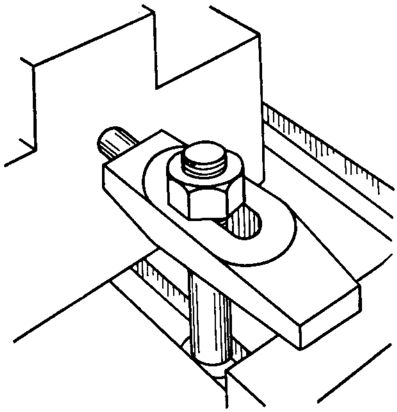


**Fig. 2-26.** The finger clamp.

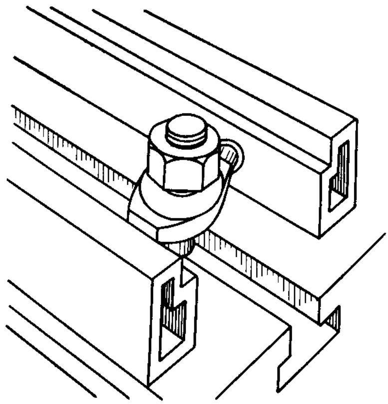


**Fig. 2-27.** The double-finger clamp.

The *finger clamp* (Fig. 2-26) is used where a cored or drilled hole is available as a support for the finger. This is a convenient way in which work may be clamped without interfering with the shaping operation.

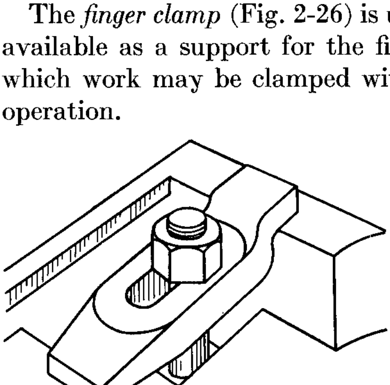


**Fig. 2-28.** The gooseneck clamp.

The *double-finger clamp* is similar to the single-finger clamp and is used in the same way. In this clamp two fingers may be inserted in the holes for support (Fig. 2-27).

The *gooseneck clamp* (Fig. 2-28), sometimes called an *offset clamp*, has the advantage of being below the surface that is being machined, when used in clamping.

The *C clamp* (Fig. 2-29), which is used, as a rule, when other clamps and bolts cannot conveniently be employed, gets its name from the fact that it is shaped like the letter "C." The parts to be held together are clamped between the pad and the end of the screw. When pressure is applied by turning the screw, the pieces are held together. These clamps come in many sizes, the smaller ones designed for light work and the heavier ones for heavy-duty work.

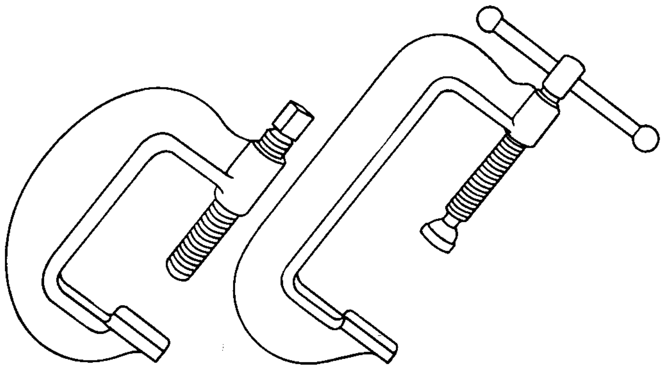


**Fig. 2-29.** C clamps.

## CAUSES OF INACCURATE WORK

**Inaccuracy in Vise or in Vise Setting.** In most shops it may be assumed that the shaper vise, as it is arranged, is right enough for the average job, but it may happen to be necessary to machine a piece which must be especially accurate—square and to an exact size—in which case it will probably be advisable to test the bottom of the vise (on which the work rests) for parallelism, and also to test the solid jaw, to make sure that it is square.

### *HOW TO TEST THE WORK SEAT*

1. Open the vise wide and set the vise jaws approximately parallel with the direction of the stroke.
2. Be sure that the bottom of the vise is clean and smooth. This is important.
3. Select two parallels high enough to project above the top of the vise jaws and long enough to extend about an inch or two beyond the width of the vise. Be sure that the parallels are clean and smooth.
4. Set these parallels as shown in Fig. 2-30, one against each of the vise jaws.

5. Select an indicator (one with a dial face preferred) and make sure that the pointer on the dial does not stick.

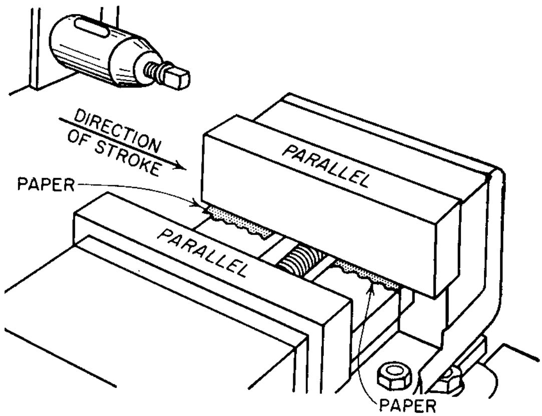



Fig. 2-30.

6. Arrange the indicator as shown in Fig. 2-31.
7. Bring the indicator contact point down to the parallel, using the feed handle of the machine; bring it down slowly until there is contact with the parallel.

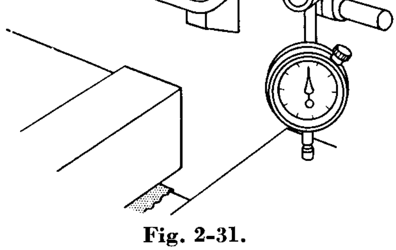


Fig. 2-31.

8. Give the feed handle a *slight* turn so as to get a reading on the indicator (about 0.010 in.)
9. Adjust the dial so that the *zero* mark of the dial is set at this reading.
10. Get readings at points *A*, *B*, *C*, and *D* (Fig. 2-32).
11. If the readings at all four places are the same (zero), the work seat is parallel.
12. If the readings are not the same, adjust by using paper shims between the worktable and the vise.

13. If paper shims are used, check the four points once more, to make sure.

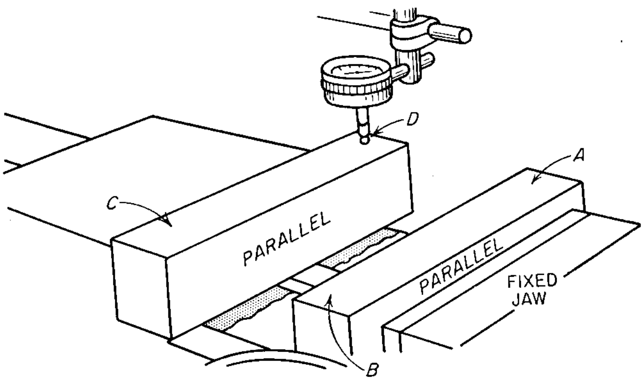



Fig. 2-32.

*To Test the Solid Jaw of the Vise.* If the face of either of the jaws of the vise is dented and scored, it should be repaired. If the solid jaw is not square with the seat, it is impossible to clamp the work against the jaw and to machine it square. To test for squareness takes only a few minutes. Follow this procedure:

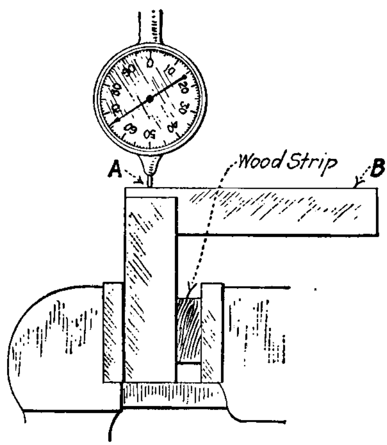



**Fig. 2-33.** Testing the solid jaw of a vise for squareness.

1. Clamp the beam of a precision square against the solid jaw (with a piece of wood between the movable jaw and the square), as shown in Fig. 2-33.
2. Arrange the indicator and move the worktable the distance *A* to *B*.
3. If the indicator registers the same at both ends of the blade, the jaw is square. It will be best to try the jaw near each end and in the middle.
4. If necessary, shim the jaw until it is square.

*To Set the Vise Parallel with Direction of Stroke.* While the graduations on the swivel plate are accurate enough for nearly all purposes, occasionally a cut, for example, a shoulder, must be made exactly parallel with the edge located against the jaw or the work may be spoiled. To test for this position is very simple. Arrange the length of stroke to about the length of the jaw, hold the indicator in the tool post and slowly run the shaper by hand to note if the indicator registers the same at both ends of the jaw. If necessary to make adjustment, clamp the vise lightly and tap with a babbitt hammer until the setting is correct, then clamp tight and test once more.

*To Set the Vise Square with Direction of Stroke.* To test and, if necessary, to correct the setting, the indicator is arranged as before, but, the vise being turned around 90 deg., the *worktable* instead of the *ram* is moved by hand to show the movement, if any, of the indicator needle.

*NOTE:* An angle plate or similar holding device when clamped to the worktable may be tested for square or may be set square or parallel with the direction of the stroke in exactly the same way as the vise.

**Chips and Burrs as a Cause of Inaccurate Work.** One of the most frequent causes of damaged or spoiled work is failure on the part of the operator before clamping the work to remove the burrs and clean the chips from the work and also from the holding device, whatever it is—vise, fixture, chuck, or clamp of any description.

*Chips.* Steel chips are worse than cast-iron chips, but if either are pinched between a finished surface of the work and the vise jaw, both the work and the jaw will be damaged, and possibly the work will be thrown out of true enough to ruin it. If chips are allowed to get under the parallels, or between the parallels and the work, it is obvious that the work will not seat properly and the finished surface cannot be accurate.

*Burrs.* Particularly on steel and wrought metal, the last few strokes tend to roll the metal over the corner, forming a burr. This burr is more or less difficult to remove, depending a great deal on the sharpness of the shaper tool. If the surface $x$, Fig. 2-34, over which the burr is rolled, is the next surface to be machined, it will cause no trouble, but if surface $x$ is to be used as a seat for finishing the opposite side, or if $y$ is to be used as a seat, the burr must be removed.

Burrs can be largely eliminated by cutting down the cross feed to a minimum on the last few strokes. Sometimes the heavier burrs are removed with a special burring chisel similar to a wood chisel; the lighter burrs are easily removed with a fairly fine file. In either case, be very careful not to spoil the corner.

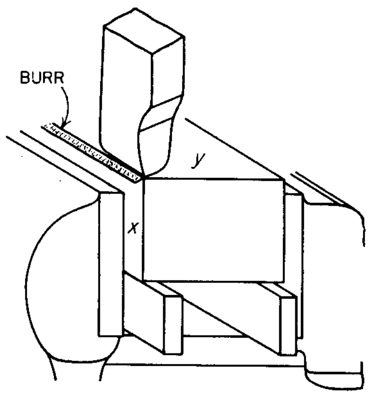


Fig. 2-34.

Then there is another kind of burr, the kind thrown up by making a nick or dent in a piece of metal. For example, pinching a rough forging or casting between the soft vise jaws without using protecting pieces will dent the jaws and throw up burrs; likewise, pinching a chip between the vise jaw and the finished surface. Dropping a parallel so that it strikes the machine may nick it and throw up a burr, and hammering a rough piece down on the parallel will do the same.

It is certain that if the work itself, the holding device, and the parallels are not clean and otherwise in good condition, at least two evils will result: (1) the work will be inaccurate, damaged, and possibly spoiled, (2) the parallels, vise, etc., will be damaged.

## *PRELIMINARY HINTS ON SHAPER WORK*

1. Keep the machine clean and well oiled.
2. Use the proper wrench or handle and, when they are not in use, keep them where they belong.
3. A vise jaw that is scored and dented and out of true is a disgrace in any shop. A real mechanic is careful. Use brass or copper or cardboard to protect the jaws when clamping the rough surfaces of bar stock, castings, or forgings.
4. Parallels should be kept clean, free from burrs, straight, parallel, and square. Examine them before using and be sure they are at least clean and free from burrs. Do not hammer a rough piece down on a parallel.

5. Be sure there are no chips on the seating surfaces, or the clamping surfaces of the vise, parallels, and work.
6. Carefully remove the burr caused by any previous cut if it will interfere with the proper seating or clamping of the work.
7. Select the proper tool, grind it carefully and oilstone it. A workman is often judged by the tools he uses.
8. To seat the work use a babbitt hammer or babbitt ball. Do *not* use a wrench.
9. Do not hammer the work with the babbitt, tap it just hard enough to seat it. Do not tighten the vise again after seating the work, as this is likely to lift the work a trifle.
10. Tissue paper "feelers" between the parallels and the work are often very useful to determine if the work is properly seated.
11. Do not pinch a thin piece of work too tight or it will buckle more or less and be out of true when the pressure is released.
12. Be sure the top of the table and the bottom of the vise plate are clean and also free from burrs before resetting a vise that has been removed from the worktable.
13. When setting the tool to a surface already finished (or to a size block), be sure the tool block is firmly seated, place a piece of tissue paper under the cutting edge, and then feed the tool down to pinch the paper lightly.
14. When setting up irregular work, be sure the head and also the bottom of the ram will clear the work during the whole length of stroke and the whole width of the cut.
15. Be sure, at all times, that the tool block works freely and seats properly. Failure to do this has caused a lot of spoiled work.
16. Do not hammer the side of the apron to swivel it. If the edge of the seating surface of the apron is dented and burred it will cause the tool block to bind in the box.
17. Maintain taper gibs, pins, and other take-up devices.

**Length of the Stroke.** The stroke of the shaper ram is dependent upon the length of work that is to be machined. On mass-production jobs, once the length of the stroke is set, it is not changed until that particular job is completed. But in the school shop, the length of the stroke is usually changed with each different job, which usually entails the machining of very few pieces. The apprentice should know how to change the stroke without too much help. It is a simple matter, and a little practice under the supervision of the instructor or the foreman should do the trick. Study the following procedure carefully.

### *HOW TO ADJUST THE LENGTH OF THE STROKE*

1. Loosen the clamping nut that locks the stroke setting shaft in position. If the shaper on which you are working has automatic clamping, like the one shown in Fig. 1-3, adjust the stroke-adjusting shaft (part 9, Fig. 1-3).

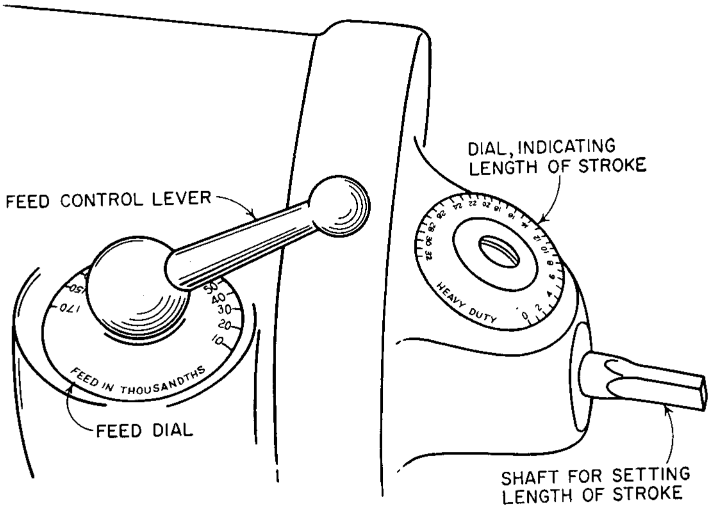



**Fig. 2-35.** Cincinnati stroke-adjusting shaft and feed dial. (*The Cincinnati Shaper Company*)

2. Using the crank provided for turning the stroke setting shaft, turn this shaft *in the direction* desired until the pointer on the stroke indicator dial (part 7, Fig. 1-3) registers the desired length of stroke. *Remember to take care of the added length for overtravel.*
3. If the crank is not needed, turn the shaft until the required reading is registered on the stroke indicator dial (Fig. 2-35).
4. After setting the desired length of stroke, tighten the clamping nut if one is used.

## SOME SHAPER OPERATIONS

**Horizontal Cut.** When the work is fed in a horizontal direction under the reciprocating cutting tool, the surface produced is a horizontal flat (or plane) surface. Most of the work done in the shaper is of this description. The length of the stroke is set for approximately ¾ in. longer than the work and the position of the stroke is such that ½ in. of this extra length comes at the beginning of the cut, to allow the tool block to seat properly for the next cut. If a given piece may be machined either crosswise with a short stroke or lengthwise with a longer stroke, other things being equal, it is better to take the longer stroke. To shape, for example, a piece 3 × 6 in., twice as much time will be wasted cutting air when cutting crosswise as when cutting lengthwise. This is illustrated in Fig. 2-36 where the shaded portion *B* shows air cut with cross stroke and *A* the air cut with the lengthwise stroke.

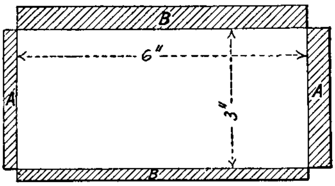


**Fig. 2-36.**

However, there is still another factor. You usually make fewer strokes cutting in the 6-in. direction than crosswise, because the strokes per minute are not necessarily in the proportion of 3 to 6 in. This becomes especially apparent in finishing.

All work, however, cannot be machined with a lengthwise stroke. It is often more convenient (and makes for better practice), when extremely heavy cuts are being taken and when the work has only a very small gripping surface, to set the vise jaws perpendicular to the ram and use crosswise strokes, shorter strokes. With this setup, the work is less likely to slip in the vise.

An important point in shaper construction and operation may be emphasized here. The shaper manufacturer takes the utmost care to have the clapper block fit the box. The bearing surfaces are scraped to provide the best of sliding fits with no shake, the axis of the hinge pin is exactly at right angles and consequently the block *hinges freely* in the box during the return stroke and is rigidly supported during the cutting stroke. The bearing surfaces should be wiped clean and a very little oil applied at least once a week. If the bearings are allowed to become dry or gummed with old oil, or if for any other reason the block does not always seat properly, trouble will surely result.

The position of the operator is at the right front of the machine with the speed and feed changes within easy reach. A low stool should be provided. In order that the depth of cut, the action of the tool, etc., can be more readily observed, the cut is usually started on the right side (the side nearer the operator), the feed of the table is arranged to move the work toward the operator *on the return stroke*, and the left-hand tool is used.

The smaller pieces or any pieces that will tend to tip under the pressure of the cut are best held with the vise jaws at right angles to the thrust.

There is practically no difference in roughing steel or cast iron except the cutting speed. For roughing plane surfaces of either cast iron or steel, the tools illustrated in Fig. 2-1 may be used. They must be held in a suitable holder.

### *PROCEDURE FOR TAKING A HORIZONTAL CUT*

1. Thoroughly clean the vise and remove all burrs by scraping.
2. Thoroughly clean the work and remove all burrs by filing.
3. Select a pair of parallels wide and long enough to have the work project above the vise jaws (see Fig. 2-37).
4. Place the work in the vise as shown in Fig. 2-37 and tighten securely.
5. Make sure that the vise jaws are set perpendicular to the ram (see Fig. 2-37).
6. Select a roughing tool ground to the form shown in Fig. 2-1, page 26.
7. Select the proper toolholder, if one is needed, set tool in toolholder, and clamp. Whether a toolholder is needed or not, clamp the tool (or toolholder) in a vertical position (Fig. 2-38) or pointing *very slightly* in a direction *away* from the work, so that if, by *any* chance, the tool moves owing to the pressure of the cut, it will move *away* from the surface instead of undercutting it, as shown in Fig. 2-39. *Note the directions of the arrows.*

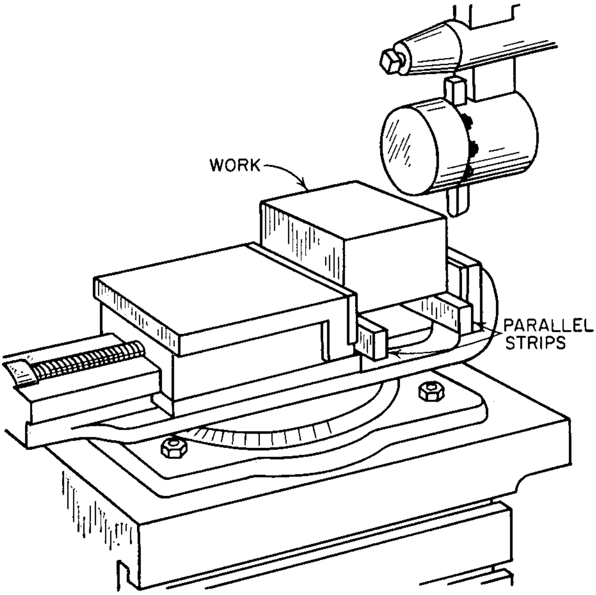



**Fig. 2-37.** Work in position for shaping.

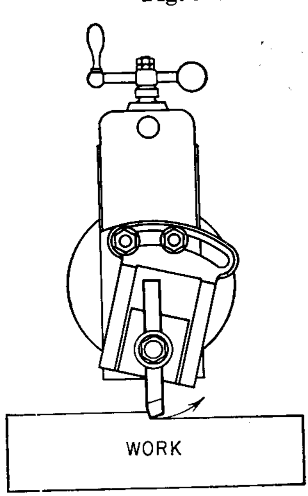



**Fig. 2-38.** The correct method of clamping a tool. (*The Cincinnati Shaper Company*)

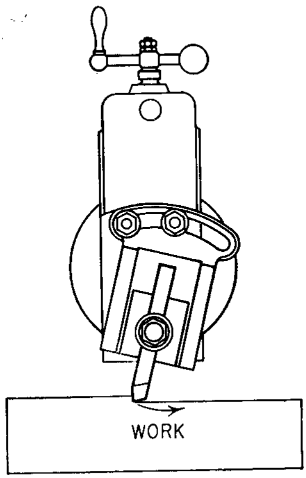



**Fig. 2-39.** An incorrect method of clamping a tool. (*The Cincinnati Shaper Company*)

8. Do not allow the cutting edge to project too far from the toolholder or the tool post—"catch it short"—and clamp it tight (Fig. 2-40).
9. *Be sure the tool-head slide is not run down too far* (Fig. 2-41), *as this causes weakness and undue strain. It is much better to take time to raise the worktable than to allow the toolslide to project below the head or have the tool project too far.*

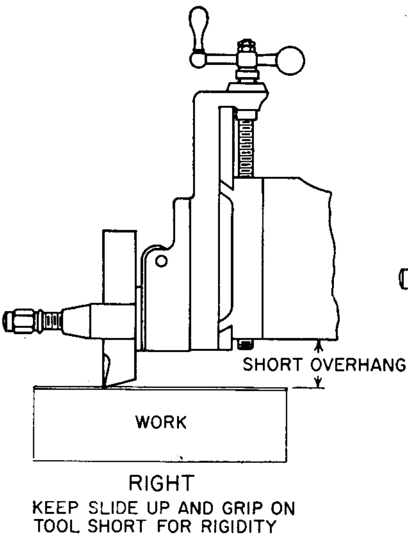



**Fig. 2-40.** Correct amount of projection of the tool from the toolholder or tool post. (*The Cincinnati Shaper Company*)

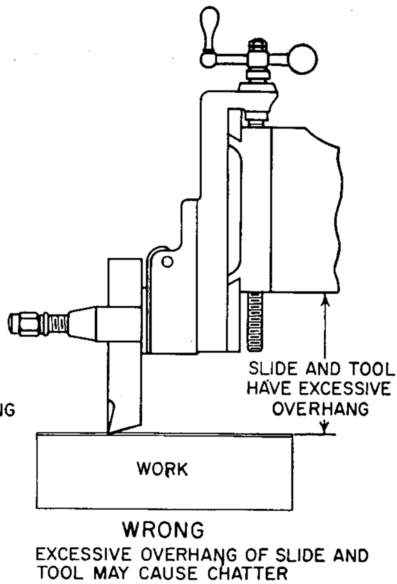



**Fig. 2-41.** Incorrect amount of projection of the tool from the toolholder or tool post. (*The Cincinnati Shaper Company*)

10. Adjust the depth of cut to be taken by means of the downfeed handle.
11. Start the machine and feed the work by hand (cross-feed) until the cut is started, then, *and not until then*, throw in the power feed.
12. When the cut is completed, stop the machine and inspect the work.
13. If more stock is to be removed, repeat steps 10, 11, and 12.

When cast metals are being machined, the edge at the end of the cut should be beveled with a chisel or an old file about 45 deg., practically to the depth of the cut (see Fig. 2-42); otherwise chunks of the corner will break out below the surface, leaving the edge ragged.

Cast-iron scale is hard and gritty. Set the tool to take a chip deep enough *to get under the scale.* If during the cut a portion of the surface is low and the tool rubs on the scale, the cutting edge will very soon be ruined. Provided that it will not make the work undersize, take a deep chip, and if necessary reduce the amount of feed, but *get the roughing cut under the scale if possible.*

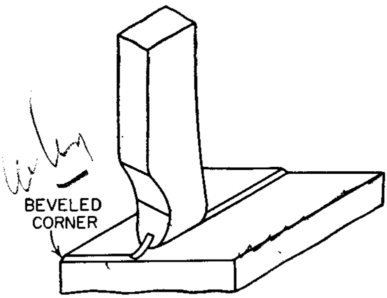


**Fig. 2-42.**

The finishing cuts should always be light. For finishing steel or wrought iron a fine feed will give the best result. A tool of substantially the same shape as the roughing tool, but with a narrower rounded end and a greater rake angle, produces an excellent finish; or if desired, the shear tool shown in Fig. 2-1, finishing tool, may be used.

The accepted commercial machine finish on flat cast-iron pieces of any considerable size is a surface that feels smooth and shows feed marks 3/8 in. or more apart. This finish is obtained by the scraping action of a broad square-nosed tool. This tool may be a forging, or the tool bit fitted to any one of a number of kinds of toolholders may be ground to shape. With a sharp tool, 0.002- to 0.004-in. chip, and 3/8 in. or more feed, a beautiful finish may be obtained. Use a slow speed and feed by hand. If the surface left by the roughing tool is badly torn, it may be necessary to take two cuts.

For finishing cast iron, the edge at the beginning of the cut should be filed slightly bevel so that the cutting edge of the tool will not strike the scale. Keep oil off cast-iron work; remember, even oily finger marks may defeat a good finish.

**Sharpening a Square-nose Tool for Finishing Cast Iron** (Fig. 2-43). Grind the front of the tool flat with 4- or 5-deg. clearance and round the corners slightly, oilstone the top and set in the tool post as nearly correct as can be judged (cutting edge flat on surface to be finished). Place a sheet of heavy paper on the work and on the paper a good oilstone. The paper is to keep oil off the work. Raise the tool block, that is, hinge it forward, and bring the oilstone and the paper under the cutting edge of the tool. The tool block is now probably hinged forward 15 deg. or more; raise the slide until it only hinges forward a very little (about 4 deg.). Bearing lightly against the tool, rub the oilstone back and forth between the paper and the cutting edge. Lift the tool, that is, hinge it way forward occasionally, and note when it is oilstoned enough; then remove the oilstone and the paper, and allow the tool block to fall back into place. It is obvious that the tool is sharp and that the cutting edge is parallel with the work and has the proper clearance (about 4 deg.). Feed down carefully to the work and take a very light chip, a coarse feed, and a slow speed.

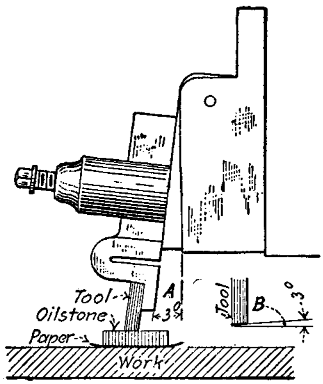



**Fig. 2-43.** If the tool is oilstoned when hinged forward, say 3 deg., as when at *A*, then, when it is seated, as during a cut, it will have a 3-deg. clearance, as shown at *B*.

**Setting the Head for Vertical and Angular Cuts.** The downfeed is used for vertical cuts, such as finishing the sides of tongues and grooves, squaring shoulders, squaring ends, cutting keyways, and occasionally for cutting off. It is used also for angular cuts, such as fairly wide beveled edges and ends, and for dovetails.

Except in the case of cutting off or a similar operation, or where the surface being machined is not much over ¼ in. deep (or high) it is very necessary to swivel the apron when using the downfeed. This is illustrated in Fig. 2-44. When the top of the apron is moved in a direction *away* from the surface of the cut, the tool block and the tool will hinge in a direction up and *away from* the work during the return stroke. This is true in angular (bevel) cuts as well as vertical cuts (see Fig. 2-45).

The setup for an angular cut with the head swiveled and the apron also set over sometimes appears awkward and wrong. It may help the beginner to imagine the angular cut as a vertical cut and set the

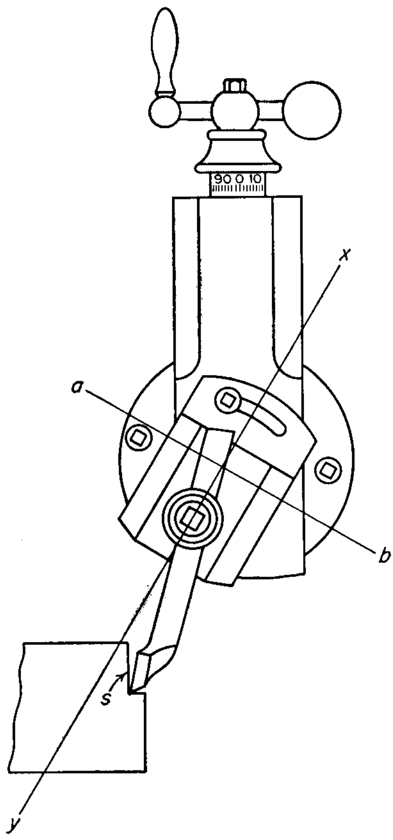



**Fig. 2-44.** Apron swiveled for a vertical cut. The axis of the hinge pin is in line with *ab*. The direction in which the tool block may rise on the return stroke is in a plane *xy*, at right angles to *ab*. If this plane is tipped away as illustrated, the tool will tend to raise in a direction away from the surface *s* and will not rub. If the plane is vertical, the tool will rub along the surface *s* on the return stroke.

apron accordingly. For all vertical or angular cuts it is important to understand and remember the following:

RULE: Always set the *top* of the apron in a direction *away from* the surface of the cut to be taken.

Although the construction permits of considerable downfeed of the head, it is not good practice to use the head with the slide run down much below the swivel plate, because in this position it is not so strong and rigid as when backed up by the ram. Sometimes it may be advisable or even necessary, but in no other case than for a finish cut.

Be careful, when setting up, to have the slide high enough at the start for either a vertical or an angular cut, so that this weakness or this interference will not result during the cut.

**Shaping Vertical or Angular Surfaces.** A *vertical cut* is made in the shaper (or planer) by setting the head exactly on zero, arranging the apron so that the tool will clear the work on the return stroke, and feeding down.

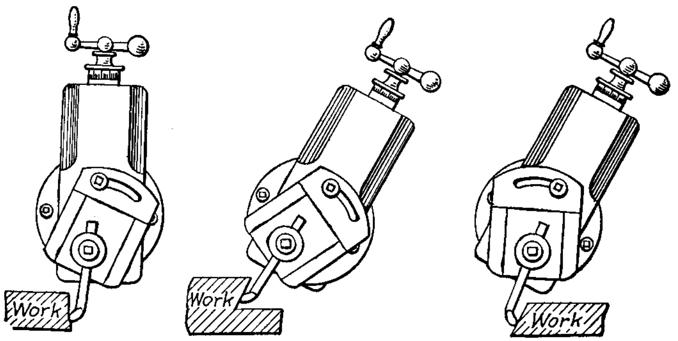



**Fig. 2-45.** Note that in each case the top of the apron is set over in a direction away from the surface being cut.

An *angular cut* is made by *swiveling the head* of the machine, arranging the apron so that the tool will clear the work on the return stroke, and feeding down.

It should be understood that it is not always necessary to take an angular cut to produce an angular surface. An *angular surface* is one that is neither parallel nor square to a given base or other surface. It may be machined in several ways:

1. The work may be supported on a tapered parallel (this is often called a taper cut).

2. A layout line indicating the position of the surface to be machined may be scribed on the work, and the work held in the vise with this line horizontal and the regular power feed used (for either taper or angle).

3. The work may be held in angular parallels (Fig. 2-17).

4. The vise may be swiveled to an angular setting.

5. Some shapers are provided with the universal table illustrated in Fig. 1-11.

6. The head of the shaper may be swiveled as shown in Fig. 2-45.

Except for a downcut on a piece held in a vise which has been swiveled to a given angle (as in (4) above), all of the first five methods suggested require only the regular horizontal cut. The last method (5) involves the angular setting of the swivel head and is very properly called "an angular cut."

Attention is called to the setup of the shaper (or planer) for the downcut for producing either a vertical cut (*a*, Fig. 2-45) or an angular cut *b* or *c*, in the same figure.

## QUESTIONS ON SHAPER WORK I

1. What precautions should be taken regarding the vise jaws? Why?
2. What are parallels used for? How should they be cared for? Why?
3. Is it good practice to pound rough castings or forgings or bar stock down on parallels? How should they be protected?
4. What are "hold-downs" or "grippers"? How are they used? When are they used?
5. State four ways in which the vise may be "out" enough to cause inaccurate work.
6. Explain how you may test the work seat.
7. Explain how you may test the solid jaw.
8. How do you set the vise jaw *exactly* at right angles with the direction of the cut?
9. Why is it necessary, in order to do good work, to keep the vise jaws clean?
10. What causes a burr on the work? There are times when it is unnecessary to remove this burr. Explain. If you have several pieces, when do you remove the burrs?
11. How much longer than the length of the cut do you set the length of the stroke? Why?
12. How is the tool arranged in the tool post for a horizontal cut?
13. Frequently one sees the tool slide run down 2 or 3 in. below the head. What does this indicate? What is the remedy?
14. When taking a cut in cast iron, why should you, whenever possible, cut under the scale the first cut?

*15.* What is the proper way to take a finishing chip on cast iron?
*16.* Explain how to sharpen the square-nose tool for a shaper cut.
*17.* How does the clapper block fit the clapper box?
*18.* Are the bearing surfaces of the block and the box smooth and clean? When and how should these surfaces be cleaned and oiled?
*19.* How may the apron be swiveled? How much?
*20.* Explain how hammering the side of the apron may prevent the proper seating of the tool block.
*21.* Why is the clapper box made so it can be swiveled on the head?
*22.* What is the rule for setting the apron when taking a vertical cut or an angular cut? Why is this rule important?

**Shaping a Rectangular Block or Similar Piece Square and Parallel.** Machine one side, preferably one of the larger surfaces (1)

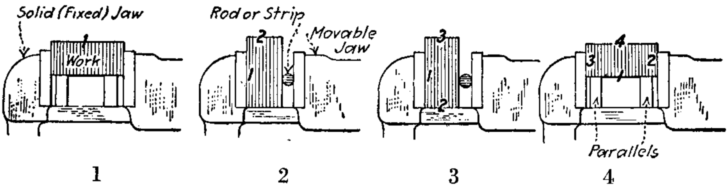



**Fig. 2-46.** The four successive steps in planing the sides of a rectangular block.

in Fig. 2-46, then using this surface as a seat against the solid jaw, plane the adjacent side (or edge) (2). If the shaper vise jaw is square and smooth and if the surface first finished is clean and free from burrs and properly seated against the vise jaw, the second surface machined will be square with the first surface. In order to make sure that the surface first machined is properly seated against the vise jaw, it is customary to use a rod or strip between the movable vise jaw and the work. This will obviate any tendency for the work to change its position, owing to any "give" in the movable jaw.

Next, place the second finished surface down on the bottom of the vise, or on parallels if necessary, and the first surface against the solid vise jaw as before, with the rod or strip between the movable jaw and the work, and tighten the vise. With a babbit hammer tap the work down in the vise to make sure that it is properly seated on the bottom, and plane surface (3). If the vise jaw is square and the tool is sharp and if care is taken to clean the surfaces of the finished work from burrs and chips, the two edges just machined should be parallel, and both square with the first side machined. Now place the first machined surface down on suitable parallels, clamp the work between the jaws *without* the rod or strip, and with a babbitt hammer, tap (not pound) the work until it is properly seated. If the vise is true and the work is seated on both parallels so that neither

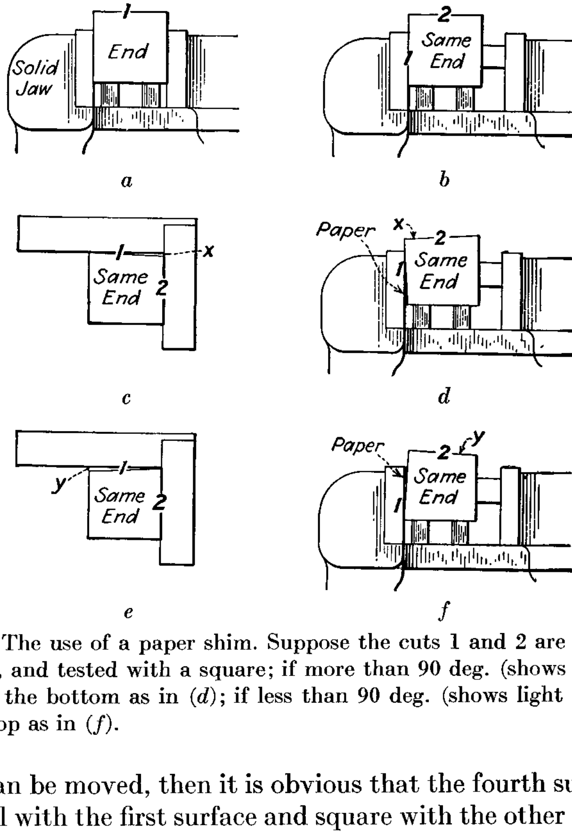



**Fig. 2-47.** The use of a paper shim. Suppose the cuts 1 and 2 are made as in (*a*) and (*b*), and tested with a square; if more than 90 deg. (shows light at *x*), shim along the bottom as in (*d*); if less than 90 deg. (shows light at *y*), shim along the top as in (*f*).

parallel can be moved, then it is obvious that the fourth surface will be parallel with the first surface and square with the other two sides.

It is better to seat the work on two parallels rather than on one, for the reason that it is easier to judge if the work is properly seated. Further, it may be desirable to measure the piece with a micrometer or caliper; this may be more readily accomplished if there is a space between the two parallels or between one parallel and the vise jaw.

*Adjacent Surfaces.* If a piece of work having one side machined is to have either or both adjacent surfaces machined square with this side, the work must be set up properly. If for any reason the work is not "square in the vise," the adjacent surface, when machined, will not be square with the surface already machined. If, for example, the solid jaw is "out of square" and the work tightened against the faulty surface, the work will be as much out of square as the vise jaw.

In testing the solid jaw of the vise, it was assumed that when the solid vise jaw is not square and true, time should be taken to correct the fault. This is not always possible; frequently it is advisable to shim the work in the vise rather than shim the vise jaw. This may be done, with paper usually, as shown in Fig. 2-47.

It is important to understand, further, that unless the *bottom* side is square with the side against the solid jaw, both of the parallels under the work will not be tight. This is shown in an exaggerated manner in Fig. 2-48. No amount of hammering will "squash" the steel and seat the work on both parallels when the work itself is out of square or when it is held out of square, but a few taps with the babbitt will seat the work on both parallels when conditions are right.

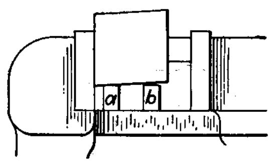


**Fig. 2-48.**

*Squaring the Ends.* The ends may be machined square in two ways, the shorter pieces by taking the cut horizontally across, and the longer pieces by cutting vertically downward. The short piece is set in the vise, either on the bottom of the vise or on a suitable parallel, and a finished edge or side set perpendicular by means of a machinist's square as illustrated in Fig. 2-49. Hold the square down hard on the parallel and the piece of work hard against the blade of the square and tighten the vise lightly. Check the setting, tap the work one way or the other if necessary, then tighten securely. If this is properly done, and the vise jaws clean and square, the end when machined should be square with the surfaces already machined. To finish the other end it is necessary merely to seat the work on the finished end, tap carefully with a babbitt hammer to make sure that it is seated, and finish to the length required.

If the work is too long to finish the ends in this manner, it may be set lengthwise in the vise, with one end projecting in a position to be finished by a vertical cut. Use parallels to raise it substantially flush with the tops of the jaws and allow it to project from the end only a short distance. A forged tool like that shown in Fig. 2-50 may be used in this operation. Tighten the vise securely. Run the tool slide well up toward the top, swivel the apron, and adjust the tool. For the reason that the tool will probably have to project some little distance from the tool post in order to take the cut to the bottom of the piece without interference, a feed and chip somewhat lighter than for horizontal machining will be advisable. Care must be taken not to break out the corners at the end of the cut if cast metal is being machined. An excellent finish may be obtained on cast iron with a

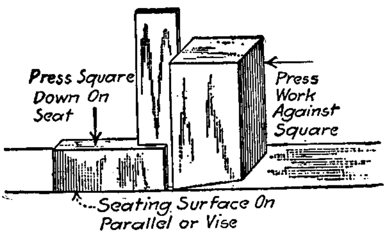



**Fig. 2-49.**

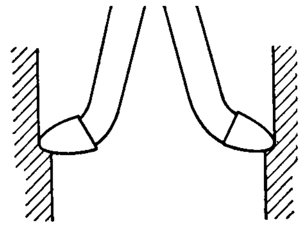



**Fig. 2-50.** A down-cutting tool.

side tool; have about $\frac{1}{4}$ in. of the cutting edge ground straight and set vertically; take a very light chip and a half turn of the downfeed screw for feed.

**Shaping an Irregular Cut.** A narrow irregular surface may be finished very efficiently with a forming tool. It will be better to hold the forming tool in a toolholder. Even if only a few pieces are to be shaped, it will probably be worth while to make a suitable forming tool. When machining a wider irregular cut, it usually is customary to lay out the irregular shape on the end of the work and machine to this line. When shaping an irregular piece to such a line, it is a good plan to rough to within a $\frac{1}{16}$ or $\frac{1}{32}$ of the line and then with a file bevel the edge to the line at an angle of 45 deg. or more, as illustrated in Fig. 2-51. With a suitable tool, with a round nose if convenient to use, machine off the bevel. If the bevel only is removed, then the surface is finished to the layout line. It is easier to see the bevel and gage the cut than it is to split the line without the bevel. When machining a wide irregular cut of a curved outline, the vertical *hand feed* may be employed in connection with the *power table feed. It is easier and better to feed down than up;* therefore, start at the highest part, feed

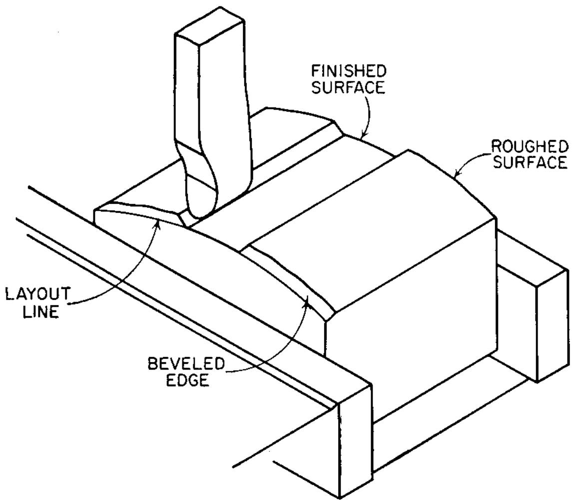



**Fig. 2-51.**

down by hand, and feed the table in the desired direction, either by hand or by power, usually by power.

**Shaping Tongue and Groove.** In nearly all cases where such operations as tongues and grooves (Fig. 2-52) or dovetails (Fig. 2-60)

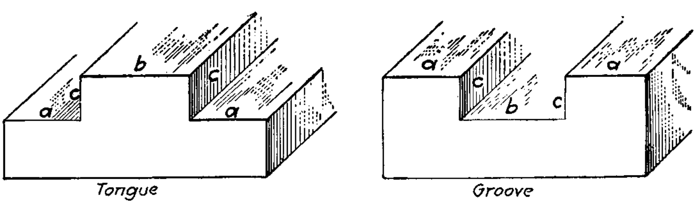



**Fig. 2-52.**

are to be made, a base surface is assumed to be finished. Also the surface where the tongue or groove or dovetail is to be cut is roughed to within $1/32$ in. The other surfaces are usually machined square with the base surface at the same time if they are ever to be machined. Assuming that these surfaces are already machined, proceed with the tongue or the groove, whichever is preferred. The tongue is considered easier because there is more room for the tool.

When shaping tongues and grooves or other shoulder operations, the roughing cuts should be made fairly close to the dimensions required, using the regular shaper tool with a small radius wherever convenient. When finishing, the surfaces *a* and *b* (Fig. 2-52) are machined to the degree of accuracy desired, but on each of these surfaces there is usually left a thousandth or two for the fitter to file or scrape off. For the distance from the surface *a* to surface *b* the graduations on the downfeed screw may be accurate enough, or if desired a size block (Fig. 2-53) may be used.[2] If the work is cast iron, the tool illustrated in Fig. 2-54 may be used for finishing all the surfaces *a*, *b*, and *c*. If the work is steel, the finishing tools for the corners (and also for the bottom surfaces) may be shaped like (1) and (2) in Fig. 2-55. The cutting edges for side and bottom are represented by *x* and *y*, respectively. These tools are given 15- or 20-deg. side rake.

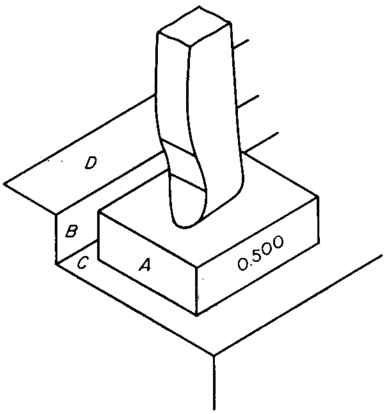



**Fig. 2-53.** Use of size blocks, *A*, ½-in. size block; *B*, shoulder; *C*, finished surface; *D*, surface to be finished ½ in. from *C*.

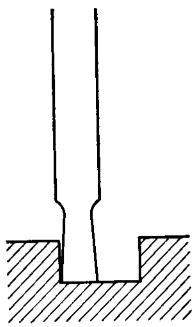



**Fig. 2-54.** A squaring tool.

[2] A set of gage blocks or "size blocks" is of great value to gage the setting of the tool for shoulders or similar projections. Gage blocks of any desired size, hardened, ground, and lapped for extreme accuracy, may be purchased, but for ordinary shaper work a piece of cold-rolled steel of the required thickness will answer.

Many machinists prefer to finish the smaller shoulder cuts in steel as well as in cast iron with a square-nosed tool without rake (Fig. 2-54). This saves changing tools and is satisfactory for small jobs and light cuts.

*Horizontal Surfaces of Tongue and Groove.* To finish the horizontal surfaces, proceed as follows:

1. If the shoulder is under ½ in. high, use a square-nose tool and have the apron in normal (vertical) position. Be sure the tool is properly ground and set "square."

2. If the shoulder is over ½ in. high and the right- and left-hand shoulder tools are used, have the apron "set over" in the right direction and then set the tool square.

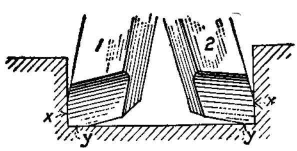



**Fig. 2-55.** Tool bits for finishing square-shoulder cuts. Surfaces $x$ and $y$ are at an angle of 90 deg. or a trifle less.

3. When using the graduations on the downfeed for gaging the vertical distance, first machine the top surface of the tongue (*b* in Fig. 2-52) to the finish size, then run the tool down the right distance and finish surface *a*, beginning at the edge and feeding toward the center. Both surfaces *a* may be finished with one setting of the tool.

4. When using a size block (Fig. 2-53) to gage the vertical distance, first machine surfaces *a*, then set the tool by the size block, and finish surface *b*.

5. When shoulder tools are used it will, of course, be necessary to reset the apron and change the tool to machine the second surface *a*. In this case, set the second shoulder tool to touch tissue paper on the finished surface and machine the second surface exactly in line with the first.

*Vertical Surfaces.* 1. When a vertical surface over ½ in. high is to be finished, it is usually necessary to set over the apron to allow the tool to clear the work on the return stroke; it is advisable to do this even when finishing the horizontal cut to a fairly high shoulder.

2. For the smaller jobs, say a tongue not over ½ in. high, the square-nose tool in Fig. 2-54, page 68, may be used for all surfaces (no setover of the apron is needed).

3. For the larger jobs the shoulder tools (Fig. 2-55) are best. Set over the apron according to the rule on page 61.

4. Whichever tool is used, first cut away (with this tool and hand feed) most of the fillet left by the round-nose tool when roughing. Then finish the vertical surface to the layout line, or to measure or gage if advisable, and feed down to the horizontal line.

5. To finish the second vertical surface if the apron has been set over for the first, reverse the position of the apron and change the tool.

6. If the square-nose tool is used without setting over the apron, merely run the tool up and over to the beginning of the other vertical surface and machine down to make the tongue the correct size.

**Fig. 2-56.** Adjustable parallels.

*Shaping the Groove.* Instructions for shaping the groove are similar to the above for shaping the tongue except for the roughing cut. To rough the groove proceed as follows:

With a cutting-off or similar tool, cut slots *inside* the layout lines for the groove, one slot on each side nearly to the bottom of the groove. Then cut away the metal remaining between the slots, possibly with the tool just used if only a small amount of stock is left. This will leave the groove roughed out nearly to the layout lines.

When finishing the bottom of the groove with shoulder tools (Fig. 2-55), finish half or more with one tool and the remainder when the apron is reset and the tool changed.

*Taper Parallels or Adjustable Parallels* (Fig. 2-56). These parallels are useful in gaging the width of a slot or a groove; slip one past the other until the slot is filled, then measure over the two with a micrometer. Possibly in a wider groove a straight parallel may be necessary to help fill the width of the groove.

**Shaping Slots, Keyways, Etc.** A keyway tool looks like a short cutting-off tool and has the same clearance angles. For shaping slots, keyways in shafts, or similar cuts, the average 14-in. shaper will carry a tool $\frac{1}{4}$ in. wide in steel or cast iron, provided a fairly light chip—0.005 to 0.010 in.—is taken. For a wider slot, two cuts or more may be necessary. If more than two cuts are necessary, take the out-side cuts to (or splitting) the layout lines, then remove the metal left between.

*Taking Cuts Which End in the Metal.* When making a cut which terminates in the metal (Fig. 2-57), it is necessary to drill a hole, and

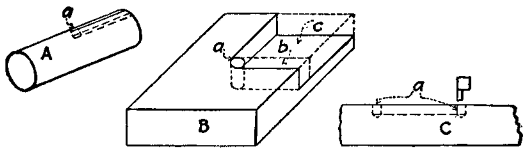



**Fig. 2-57.** In *A*, the diameter of the drilled hole (*a*) equals the width of the keyway. In such a job as *B*, first drill a hole at (*a*), say ¼ in. in diameter; next, machine the slot (*b*); then machine the remainder (*c*), the cut ending in slot (*b*). Such a job as *C*, requires a drilled hole at the beginning and also at the end of the keyway. Keyways such as those shown in *A* and *C* are cut more easily in the milling machine, but occasionally must be cut in the shaper or the planer.

in wide cuts to machine a groove, at the end of the cut, for the reason that if the chips are not cut off, they will remain to clog the cut and soon break the tool. Occasionally it is required to machine a groove, a keyway for example, somewhere between the ends of a rod or shaft (*C*, Fig. 2-57). In such a case, holes should be drilled at the beginning and end of the slot.

*NOTE:* Modern shapers are constructed to permit the end of a shaft to extend beneath the ram as far as desired.

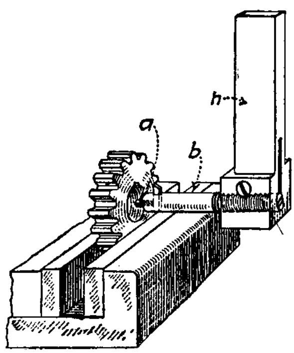



**Fig. 2-58.** Planing a keyway in a gear (see also Fig. 2-59).

*Shaping Keyways.* It is often convenient to use a shaper to cut keyways in the hubs of pulleys, gears, etc. A forged tool for this purpose is not economical and is not much used. Figure 2-58 shows a homemade keyway toolholder that works well. The tool point is held in the bar *b* by a setscrew *a* at the end. The thread on the bar screwing into the holder *h* helps materially in holding. Bars of various lengths may be used. It is much more efficient to set up the work with the layout on *top* and feed *up* because of the tendency otherwise for the tool to chatter and jump.

If the diameter of the work to be keyseated is fairly small, use a short bar *b* and let the tool post travel back and forth over the top of the work. That is, while the toolholder *h* is not caught especially short in the tool post, this lack of rigidity is more than made up for by the shorter bar.

If the diameter of the work is so large that the tool post cannot move back and forth over the blank, the bar will have to project far

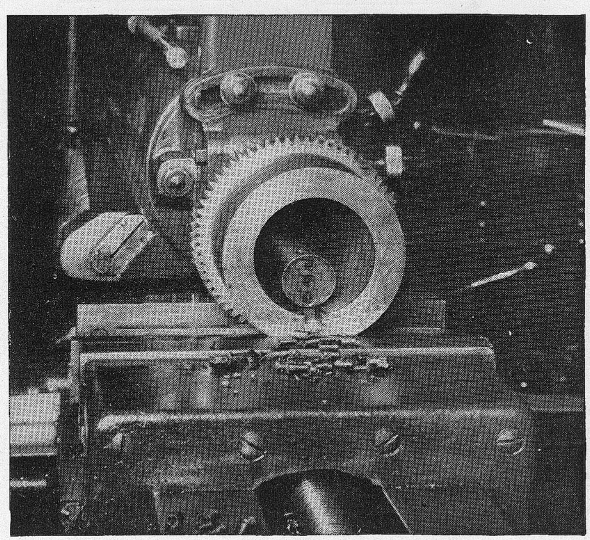


**Fig. 2-59.** Cutting a keyway in a ring gear. (*The Cincinnati Shaper Company*)

enough in front of the tool-post screw to go through the hole. In this case catch the tool short.

Notice that the diameter of the bar in Fig. 2-58 is smaller than the diameter of the hole. This is because the tool bit projects and both the bar and projecting bit must go through the hole.

It frequently happens, when work is held in the shaper vise as shown in Fig. 2-58, that the ram will not go back far enough. In such a case do one of three things: (1) Put one or more parallels, as high as the vise jaw, back of the work and clamp the work between the parallels and the movable vise jaw. (2) Clamp the work in a plain vise (milling or drill press) and then clamp this vise in the desired position in the shaper vise. (3) Remove the shaper vise and clamp the work to an angle plate.

The tool bit for keyseating is quite small and must be very carefully ground. The cutting edge must have clearance in order to cut, and the sides also must be backed off a little; otherwise the tool will rub. Only a very little clearance is needed. To avoid any tendency to have the top of the keyseat wider than the bottom, the cutting edge of the tool should be the widest part, but only a trifle wider; use a micrometer and be careful. Give the tool very little, if any, front rake, and no side rake.

The layout for a keyseat takes only a few moments and is usually done at the time the keyseat is made by merely scribing a radial line, using the center square. If advisable, in addition, to lay out the full width of the keyseat, it may be done quickly in the shaper vise, using a flat square and a scale. However, if several pieces are to be scribed, the full width of the keyseat, it will be best to do the layout work before setting up the machine.

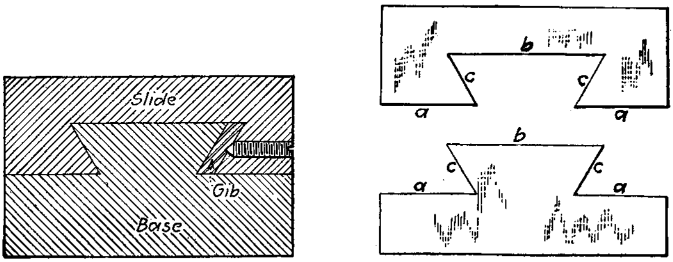
Most keyseats that are machined in a shaper may be made with a full-width tool bit. Draw a radial line indicating the center position of the keyseat. Grip the work lightly at first, and, with a square, set the radial line perpendicular. Then tighten the work securely.

After properly setting the tool, adjust the worktable until the radial line is central with the tool. Take one stroke of the shaper by hand to be sure that there is no interference. When the tool touches the work, set the graduations at zero and feed the required depth. On account of the springy nature of the tool, use a fairly slow speed and do not feed over 0.010 in. per stroke.

If the keyseat is too wide for one cut, it is best to lay out the sides of the slot to be cut, and cut to the lines. Possibly two cuts will make a satisfactory job, but often it is advisable to make three cuts—a full-depth "stocking" cut near the middle, and a finish cut to each line.

**Shaping Dovetails.** A dovetail slide bearing is illustrated in Fig. 2-60. To shape a dovetail calls for operations which are very similar to cutting a tongue and groove. If the student has not already cut a tongue and groove, his attention is called to information in detail given in the paragraphs covering the shaping of tongue and groove, the horizontal surfaces of tongue and groove, the vertical surfaces, and shaping the groove, pages 67 to 70.

Refer to the drawing to note if other surfaces than the dovetail are to be machined, and if so, machine them first. Finish the base surface, for the reason that it is easier to lay out and work from a finished base or working surface. Then machine the other surfaces to within, say, $\frac{1}{32}$ in. of the finished size. It is best to leave about $\frac{1}{32}$ in. on both sides until after the dovetail is cut; then the sides of both the base and the slide may be easily finished in exact relation to the dovetail.




*a* *b*

**Fig. 2-60.** Dovetail slide bearing.

Whether it may be advisable to *rough* one angle and then turn the piece end for end and rough the other angle will depend upon the job. It will save time of changing the tool and the setover of the apron, if practicable. It is not considered good practice to *finish* a dovetail by cutting one angle and then turning end for end; when one part of the dovetail is finished, the work should not be disturbed until the whole dovetail is finished.

The chief difference in shaping the tongue and groove and the dovetail is in the shaping of the surfaces *c* (Fig. 2-60). These surfaces, being angular, call for the setover of the *swivel head* of the shaper, also the setover of the *apron,* and for at least one pair of undercutting tools. The shape of the tools is illustrated in Fig. 2-61. A setup is shown in Fig. 2-61. If considerable metal must be removed, it will probably need a roughing and finishing tool for each side.

*CAUTION:* Do not run the ram back into the column with the slide at an angle.

Assuming the base and sides are already finished, or at least squared up, the surfaces *b* in Fig. 2-60 and the greater part of the

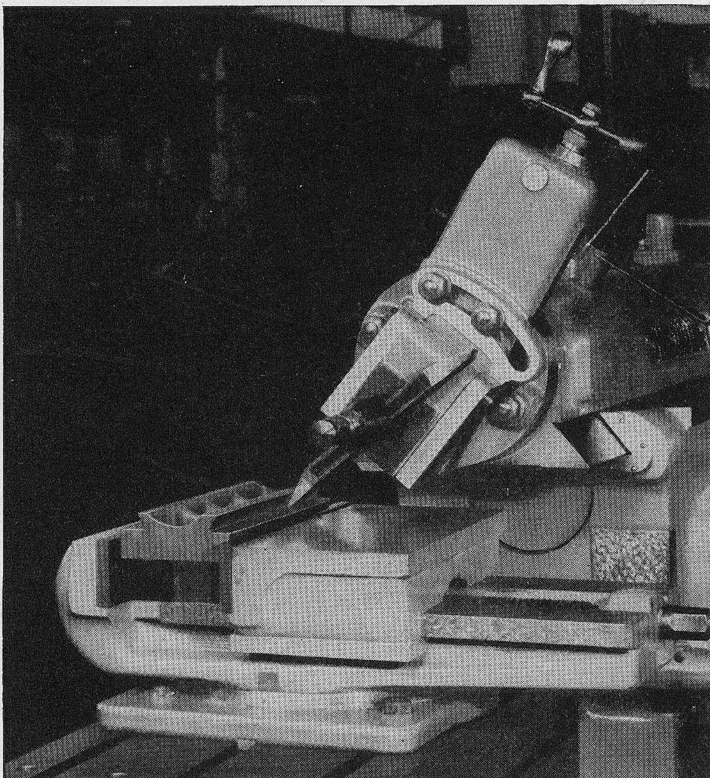


**Fig. 2-61.** Shaping a dovetail. (*The Cincinnati Shaper Company*)

surfaces *a* should be roughed practically as for tongue and groove, and, without the setting of the work's being disturbed, the surfaces *c* and the portion of the surfaces *a* under the overhang may then be roughed. The finishing operations are made in the same order. The tool that is used for finishing surface *c* may be properly used for surfaces *a* and *b*, provided that the tool is not too slender and the surface is not too large. The beginner should pay particular attention to the swivel of the apron. Remember that when *not* properly set it may appear all right, and therefore be *very sure* that the *top* of the apron

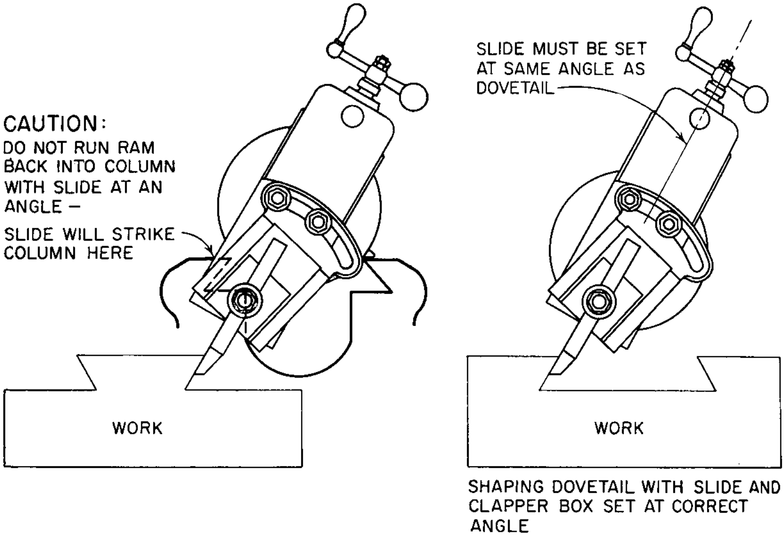



**Fig. 2-62.** Position of the slide when cutting a dovetail. (*The Cincinnati Shaper Company*)

is set in a direction *away* from the surface being finished. Study Fig. 2-62 very carefully for setting the slide.

**Measuring Dovetails.** Probably the greatest difficulty in producing dovetails is in measuring them. When a gib[3] is used between two of the sliding surfaces as great a degree of accuracy is not re-

[3] *Gib.* In machine construction a piece of metal arranged to provide an adjustment for a bearing. In *a*, Fig. 2-60, is shown a cross section of a *straight gib* between two bearing surfaces and adjusted by a series of screws. Frequently *taper gibs* are used, and the dovetail in the base is made correspondingly wider at one end. Such a gib is adjusted lengthwise to take up the wear in the bearing surfaces.

quired in machining as when the two pieces fit together. In either case, however, a smooth cut is necessary, a thousandth or two should be left for scraping, and care must be taken not to "leave too much," and certainly not to "take off too much." It is good practice to lay out the dovetail and, if possible, it should be scribed on a surface that has been finished. If several pieces are to be machined, it will be advisable to make a template of sheet metal ⅛ to ¼ in. thick to use for laying out and possibly as a gage.

The table given below should prove helpful in making accurate measurements of dovetails to find how much more it may be necessary to shape an angular surface and also to check the finished product. It consists of a series of fixed values for determining the measurements for various angles of dovetails when using various sizes of drill rod.

At first glance the table may appear rather difficult, but its use involves only addition, subtraction, and multiplication of decimals. In principle it is similar to the three-wire method of measuring threads and its application is just as easy.

**Measuring Dovetails with Pieces of Drill Rod**

In the table, $R$ is the diameter of the drill rod and the values of $D$ and $F$ have been calculated as follows:

$$D = R\left(\cot \frac{\angle a}{2}\right) + R. \qquad F = 2 \cot \angle a.$$

| Various diameters of drill rod, in. | | Various values of angle $a$ | | | |
|---|---|---|---|---|---|
| | | 45° | 50° | 55° | 60° |
| $R$ = ¼ | $D$ = | 0.853 | 0.786 | 0.730 | 0.683 |
| $R$ = ⅜ | $D$ = | 1.280 | 1.179 | 1.095 | 1.024 |
| $R$ = ½ | $D$ = | 1.707 | 1.572 | 1.460 | 1.366 |
| $R$ = ¾ | $D$ = | 2.562 | 2.358 | 2.190 | 2.049 |
| | $F$ = | 2.000 | 1.678 | 1.400 | 1.155 |

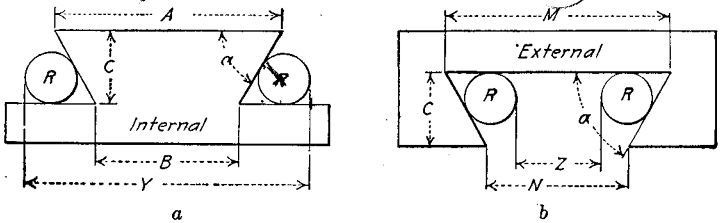



**Fig. 2-63.** Measuring internal and external dovetails.

## RULES

### Internal

CASE 1: When the dimension $B$ is given on the drawing.

$$Y = B + D$$

EXAMPLE: Angle $a = 60°$, $B = 2\frac{1}{2}''$. Using $\frac{1}{2}''$ drill rod, what should $Y$ measure?

SOLUTION: $Y = 2.5'' + 1.366'' = 3.866''$.

CASE 2: When the dimension $A$ is given on the drawing it is necessary to find dimension $B$ before proceeding further.

$$B = A - CF$$

EXAMPLE: Angle $a = 60°$, $A = 3''$, $c = \frac{3}{4}''$. Using $\frac{1}{2}''$ drill rod, what should $Y$ measure?

SOLUTION: First find dimension $B$ thus: $3'' - 0.750'' \times 1.155 = 3'' - 0.866'' = 2.134''$.

Then $Y = 2.134 + 1.366 = 3.5''$.

### External

CASE 1: When the dimension $M$ is given on the drawing.

$$Z = M - D$$

EXAMPLE: Angle $a = 60°$, $M = 3''$. Using $\frac{1}{2}''$ drill rod what should $Z$ measure?

SOLUTION: $Z = 3'' - 1.366'' = 1.634''$.

CASE 2: When $N$ is the dimension given on the drawing it is necessary to find dimension $M$ before proceeding further.

$$M = N + CF$$

EXAMPLE: Angle $a = 60°$, $N = 2''$, $c = \frac{3}{4}''$. Using $\frac{1}{2}''$ drill rod what should $Z$ measure?

SOLUTION: First find dimension $M$ thus: $2'' + 0.750'' \times 1.155 = 2'' + 0.866'' = 2.866''$.

Then $Z = 2.866'' - 1.366'' = 1.5$

*NOTE:* The measurement of $Z$ may be made with adjustable taper parallels placed between the rods (see Fig. 2-56).

**Vertical Shaper.** The vertical shaper (Fig. 2-64) is much used in general machine shops and toolmaking departments. The worktable with longitudinal, transverse, and rotary feeds, both hand and power, gives certain advantages; for example, a variety of power-fed cuts, straight and curved, with the layout lines always in plain sight of the operator.

The construction of the vertical shaper has many points in common with the standard (horizontal) shaper. The operation of the two

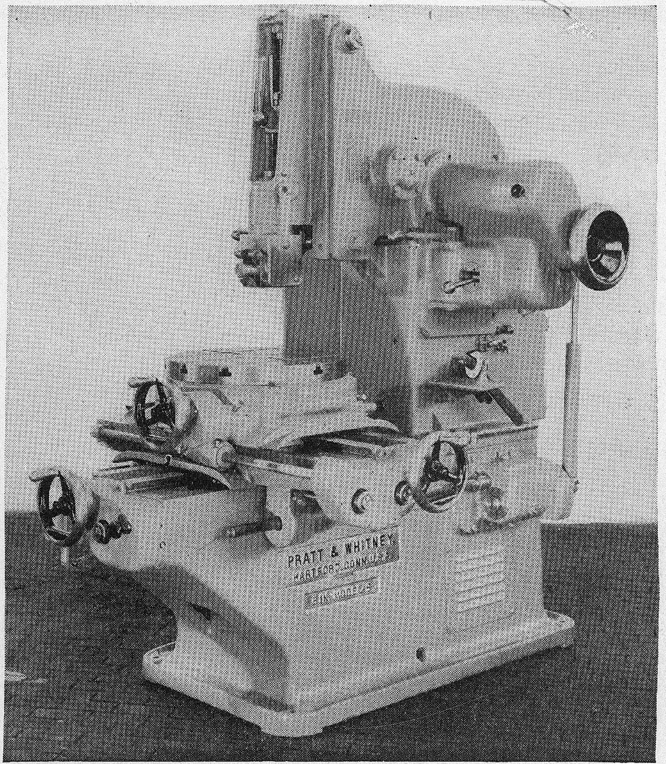



**Fig. 2-64.** The vertical shaper. (*Pratt & Whitney Company*)

types is very similar as to tools used, feeds, speeds, layout, and measurements.

## QUESTIONS ON SHAPER WORK II

1. Given a rectangular block, say, 2 by 6 in. and 1 in. thick, which side should you machine first? Which side will you machine next?
2. After machining the first surface, what objection is there to machining the opposite surface next?
3. Why is the surface that has been machined placed against the solid jaw of the vise?
4. Why is a strip placed between the movable jaw and the work?

5. When machining the third (and fourth) surfaces the use of two parallels, as far apart as convenient, is advisable. Why?
6. How does the arrangement suggested in the preceding question aid in making accurate measurement?
7. How are the ends machined when the piece is short? When the piece is over 6 or 8 in. long?
8. How is short work adjusted in the vise to make sure it is square when machining the ends?
9. Is a coarse feed or a fine feed used in finishing a cast-iron surface in a shaper? For finishing steel?
10. What kind of tool is used for finishing cast iron? For steel?
11. In machining cast metals, what precautions should be taken to prevent the metal breaking out at the end of the cut?
12. Why is it necessary to get under the scale when cutting cast iron?
13. What is a forming tool? Why is a spring toolholder excellent for holding a forming tool?
14. How is a comparatively narrow irregular surface planed?
15. After a wide irregular surface has been roughed out how is the edge beveled? What is the object of the bevel?
16. What kind of tool is used for finishing the sides of a tongue? The horizontal surfaces?
17. How may the groove be accurately and quickly measured?
18. Strange as it may seem, the cutter for keyseats works better up than down. How do you account for this?
19. When required to cut a keyway in a shaft a certain distance why do you first drill a hole at the end of the keyway?
20. What tool is used for finishing the angular surface of a dovetail?
21. State two distinct operating advantages of the vertical shaper.