BASIC UNIMAT

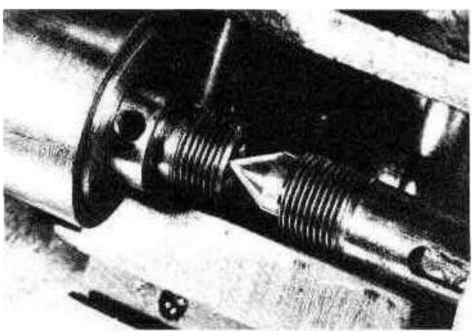
LATHE cont'd

headstock alignment, precision drilling and boring on the lathe...

HEADSTOCK SPINDLE ALIGNMENT

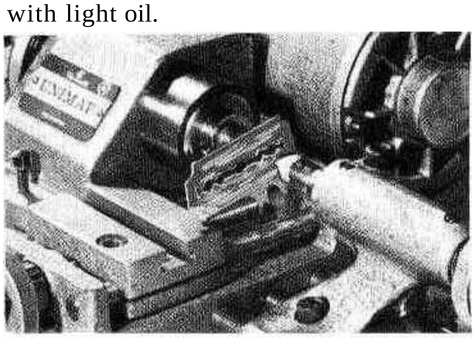
So far, the photos have shown Unimat cutting operations where only rough alignment, of the spindle was needed. At the factory, Unimat headstocks and tailstocks are mounted on the bed, and then bored in one operation. An alignment pin feeds through the lip on the front of the headstock casting down into the lathe bed. It gives you a quick way to center the headstock when setting up rough cuts, installing attachments, etc. However it is accurate to only about one degree of are.

Precise small work requires that the lathe run dead true. To make this setup, loosen the tapered bed clamping screw



10A Centering headstock spindle.

(see Photo 7B) and remove alignment pin. Never force any parts of the machine. Sometimes, if your Unimat is stored in a cold or dry place, and not used for months, tight fitting parts may stick in place. Use a penetrant such as "Liquid Wrench", a solvent available in small cans at hardware stores, and follow up with light oil.



10B Razor blade checks alignment.

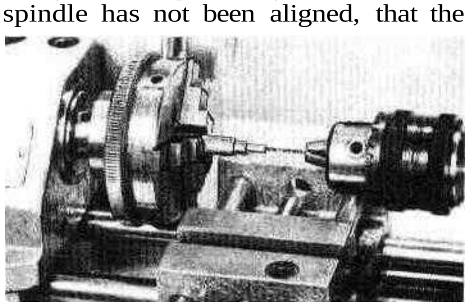
Now insert the feed pinion handlever in the headstock and move the headstock spindle as far as it will go to the right. Lock in place with the headstock cap screws. Move the cross slide, either all the way left until it almost touches the headstock, or far to the right out of the way. Then clean the tailstock bore with kerosene and cloth, and insert for centering.

Next move the tailstock over, lock it to the bed and gently feed it into the headstock spindle as in Photo 10A. As you do this, be sure that there are no chips or cuttings between head and tailstock castings and bed, or on any parts involved. As the dead center mates with the headstock spindle, it will line up the headstock to dead accuracy. Tighten the bed tapered clamping screw, but do not use the aligning pin.

One quick way to check alignment is to use a razor blade as in Photo 10B. If the blade will hang vertically to the bed axis between the two dead centers, your headstock is perfectly aligned. If you find that there is error, you may need to loosen the tapered bed screw slightly and tap on one corner of the headstock casting—(use a wood block or soft rubber mallet) to bring the center into alignment.

PRECISION DRILLING ON THE LATHE

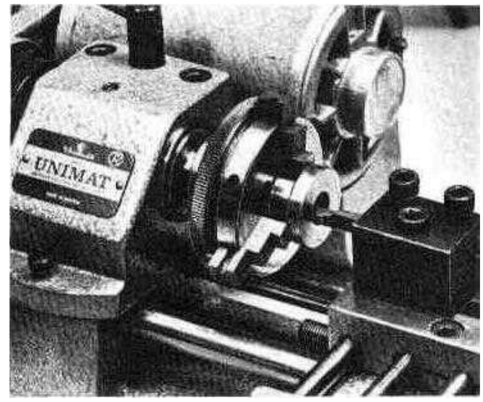
For ordinary rough work, you can drill holes in workpieces held in the 3-jaw lathe chuck even though the headstock spindle is not perfectly aligned. There will be no trouble with drills down to 1/32". However, if you want to drill perfectly true deep holes, you'll find if the spindle has not been aligned, that the



10C Drilling size 80 hole.

drill may produce a slightly conical hole, or the workpiece may even break drill.

With the headstock properly aligned, and chucks and bed cleaned and oiled, you should have no difficulty in drilling (Photo 10C) as small as an 80 size drill (.0135"). This type of work is impossible on most; lathes without, special tools. Whenever drilling below 1/32-inch, remember that it is easy to overfeed the drill. Small drills will tend to bend out of line slightly while they are making the hole, causing inaccurate bore. It is a good idea to watch the work with a large magnifying glass mounted over the lathe.



10D Boring tool mounts perpendicular toworkpiece.

PRECISION LATHE BORING

Purpose of the boring bar (DB 1105) is to turn internal holes. The end of the boring tool is ground at an angle similar to that on the side of the facing tools. Drill the workpiece out to 1/4 or 5/16-inch. Then set the boring bar in the tool holder, centering the edge as with the other lathe tools. Be sure to mount the tool in the holder parallel to the lathe ways. Do not try to cut all the way in one pass. Because chips tend to gather in the workpiece, it is necessary to run boring cuts slower and at lighter feeds than for external work.

As the bore approaches the desired diameter measure with vernier calipers, or by checking against a fitting part.

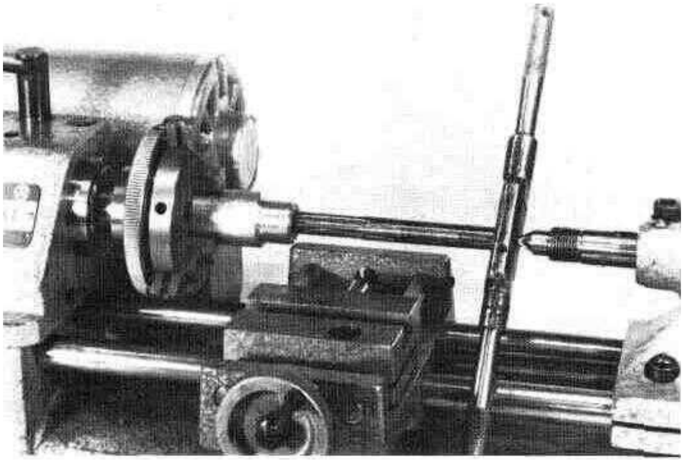
As an example, let's say that the bored ring in Photo 10D is a bearing that has to fit smoothly on a rotating shaft. As we make successive cuts, we can measure with various types of machinist's gauges. But these gauges are costly and you may not have one in the exact size desired.

The easiest method is to machine the shaft first. Make it about one-half inch longer than finished size, and turn off the extra material for use as a diameter gauge.

Use a fine jewelers file or garnet paper and finishing cloth to polish down one end of your plug so it measures about 1/1000" less than the desired shaft diameter. Then as you bore the finish hole in the bearing, use this plug to check the inside diameter. Mark the plug gauge and save for future use. A set of such diameter gauges can be made up as you go along and will have many uses in the Unimat shop.

When boring large holes with the 3-jaw chuck, it is important to improvise a stop on the bed to prevent the tool from cutting into the chuck itself. You can clamp a wood block on the bed, or make bed stops by sawing a ring like the one shown in Photo 10D in half. It is also a good idea to chamfer the end of the workpiece facing the chuck, to avoid having to bore all the way through.

LATHE cont'd



11A Tap handle holds reamer. Use tailstock center to guide look Same setup is used for precision tapping.

precision reaming, mandrel turning.taper turning and lathe threading...

PRECISION REAMING

Though the Unimat runs more accurately than most common lathes, twist drills will generally not cut dead accurate. Boring tools below 1/4-inch are not satisfactory on deep holes because the tool may tend to bend against the workpiece. Reaming is a standard technique to use whenever you want small holes accurate to l/1000-inch.

Just like drills, reamers are available in fractional, number, letter and metric sizes. The reamer is a stiff fluted tool and should never be used to remove more than a few thousandths stock. To make the precise-fitting camera post (Photo 11A) the workpiece was first turned down to diameter and then drilled one size under that of the reamer.

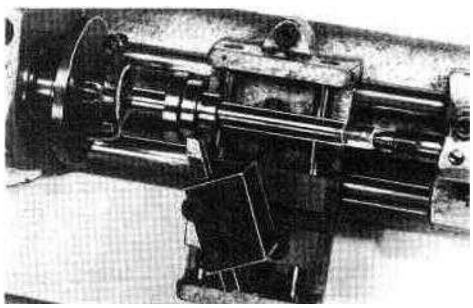
The reamer should not be driven by the drill chuck in the tailstock, since this can cause inaccuracy. Instead hold the reamer in a tap wrench. The back end of most reamers has a center hole. Engage this on the tailstock center, and either with the work rotating at very low speed, or turning the headstock spindle pulley by hand, slowly feed the reamer into the hole. Never force a reamer. Use plenty of lubricant and on deep holes, remove the reamer frequently and clean the chips. Never turn a reamer backward in the work since this can dull the cutting edges,

MANDREL TURNING

Many kinds of work are difficult to mount in the regular chucks. An example is a three step pulley. Photo 11B. The workpiece is mounted between centers on a special hardened dead straight bar called a mandrel. Mandrels are available through machinist's supply houses and are made by drill manufacturers. One end of the mandrel is slightly smaller than the rated diameter, while the other end is larger.

Thus to turn the pulley, your first step la to rough turn the stock to approximate diameter and length in the 3-jaw chuck. Then center drill one size beneath the required shaft diameter. Next the workpiece is reamed, in this example wiih a 1/4-inch reamer. The smaller end of the 1/4-inch mandrel is inserted in the hole and the work tightened by tapping the other end of the mandrel with a soft mallet until the work locks on the shaft.

Both ends of the mandrel have hardened centers. It mounts between lathe



11B Turning pulley on mandrel.

centers with the faceplate driving the lathe dog. As the work revolves you may find that you want to use scrap electrical wire to tie the lathe dog solidly to one faceplate slot, to prevent vibration in the cut. Be sure to oil the tailstock center every few moments as the machine runs. The tailstock should not be set so tightly that the center overheats. If much of this kind of work is done, you will need a DB 1220(a) ball bearing center.

TAPER TURNING

A taper is a cone-shaped cut on a workpiece. Taper-cut pins, shafts, and fittings are widely used in instruments and on machine tools because a tapered rod docs not wear into a hole to cause inaccuracy. Taper pins, for example, tend to seat in their sockets with a wedging action that self corrects for wear.

On most lathes, you cut a taper by offsetting the tailstock. On the Unimat. it is done by operating the headstock at an angle away from dead center. To machine a tapered bar or pin. *punch the ends of your work and mount between centers.* If the headstock is angled toward the rear of the lathe, the taper will cut smaller at the tailstock end. If the headstock is angled toward the lathe front, the taper will cut small at the left.

To establish the taper angle, you will need a *test bar.* This bar will have many other uses later on. Select a perfect piece of free-machining steel about 3/8-inch diameter. Cut it exactly 5 inches long, true the ends and mark the centers. Punch carefully. Align the lathe centers perfectly and use the roughing tool to cut the bar down to about .260". Then use the left hand finishing tool to turn the bar down to an exact 1/4-inch diameter. If your lathe was properly setup, the diameter should check exactly at .25" from one end to the other.

Now to set up a typical taper cut. Let's say that we want to machine some bars that have a taper of 1/4" per foot. This is equal to a taper of .1042 per the 5-inch length of our test bar. Place the test bar between centers and adjust the headstock angle until you can read a difference of .104 between the bar and the tip of a tool mounted exactly on center in the tool holder. Use shim stock (or a dial gauge mounted in the tool holder) and check at each end of the bar.

Some inaccuracy may be introduced when the centers drive the work at an angle. Therefore make a trial cut on another 5-inch bar. Exact diameter is not required since the taper per foot ratio will remain the same. Use a micrometer at each end to measure the diameters. Difference is the taper. A slight adjustment of the headstock may be required.

Tapers can also be bored in short workpieces held in the 3-jaw chuck. If you plan to do considerable taper turning, it would be best to make a protractor bar which can be chucked in the tailstock for checking headstock offset against the faceplate.

LATHE THREADING

There are three ways of cutting threads on the basic Unimat: with taps, dies, or the Thread Chasing Attachment (DB 1270). To cut internal threads, you first must prepare the right size hole for the thread desired.

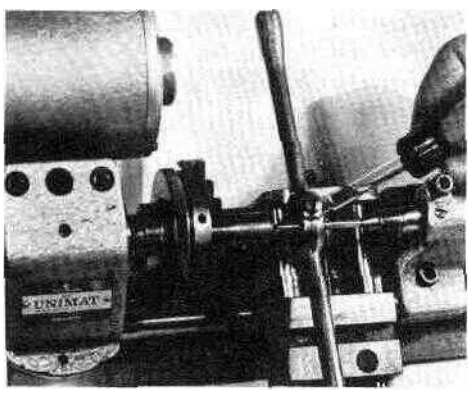
BASIC UNIMAT

LATHE cont'd

die threading, making special chucks,

See a drill rnanulacturers chart of drill sizes. For example, the standard camera socket thread is 1/4-inch diameter by 20 threads-per-inch. For this thread, drill your workpiece with a #7 drill. Then with the lathe properly aligned, drive the tap like the reamer (Photo 11B), and slowly feed the tap into the work by hand. Often, it will be convenient to start the tapping in the lathe and then finish the thread with the workpiece held in a vise.

External threads are cut with a threading die (Photo 12A). It is helpful



12A Use plenty of cutting oil when threading with a die.

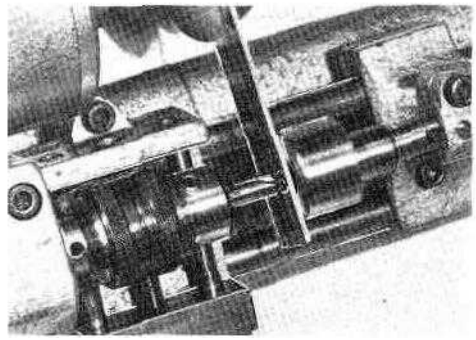
to start, the die against, the work by temporarily placing the faceplate on the tailstock. This guarantees a straight feed. Do not use power. Gently turn the work by hand feeding plenty of oil to the die. If the tension on the die increases, back off, brush chips away and repeat.

FACEPLATE TURNING

The Unimat faceplate has three slots which fit the standard cap screws used on the machine. Flat work up to the full swing of the lathe can be screwed directly to the faceplate. Extra holes can be drilled in the faceplate to clamp odd shaped workpieces that will not fit into the regular chucks. If you mount such work off center, be sure to balance the faceplate with additional bolts if cuts are to be made at high speed. For special setups, extra faceplates are available, (DB 64).

SPECIAL CHUCKS AND SPINDLE FITTINGS

Some kinds of work cannot be held satisfactorily in any ordinary way. The chuck shown in (Photo 12B) was made to hold 1/2-inch round balls for center drilling. To make the chuck, a piece of 1 1/4-inch round steel bar stock was cut to 1 1/2-inch length, and mounted in the 3-jaw chuck. Next it was center drilled



12B Ball chuck doubles as handy tailstock drill pad.

and bored to about 3/8-inch diameter and then drilled with a 13/32-inch drill. This produced a good fit when threaded with the Unimat TM 12 headstock tap (12mm dia. x 1mm pitch). The back end of the thread was cleared to fit the headstock spindle, and the front end bored on a slight taper to fit the balls. During the drilling the balls were held in place with a few drops of sealing wax.

This example illustrates how you can make chucks to fit any kind of special part. Such chucks can also be threaded to hold screws during machining operations. The chuck does double duty as a handy tailstock plate for drilling on the lathe. (Photo 12B)

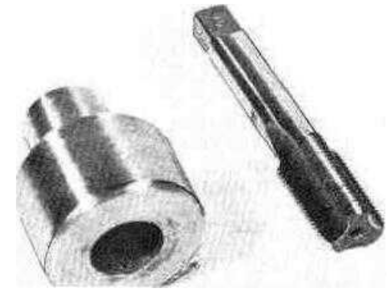
DRILL PRESS

setting up your drill press

The changeover from lathe to vertical drill press can be accomplished in less than a minute. Remove the tapered screw which holds the headstock to the lathe bed. Pull out the headstock alignment pin and lift out the complete headstock unit. Substitute the drill press column, replacing and tightening the tapered screw. *Note that this is a one-way fit.*

Next remove the tapered screw in the adapter on the vertical column and use it to mount the headstock. Place the faceplate on the spindle and with the headstock tapered screw partially tightened, lower the assembly until it touches the top of the cross slide as in Photo 12D. Now tighten the headstock tapered screw and your drill press is vertically aligned ready to use. (Etch a mark in both castings for ready reference).

A slotted screw threaded to fit chucks and faceplate is supplied with the basic Unimat. Use the screw (Photo 13A) to



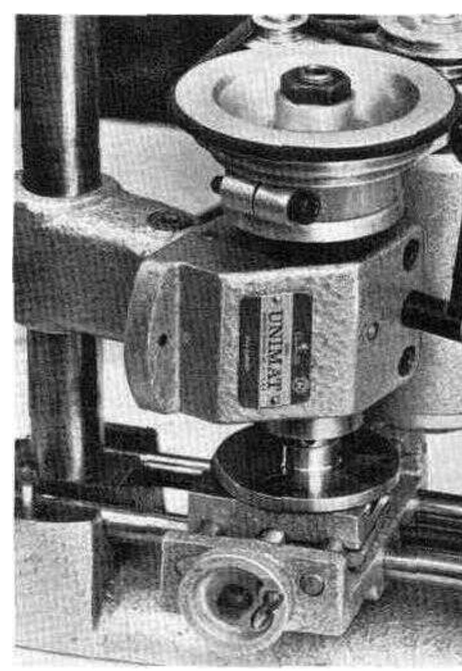
12C Ball chuck and TM 12 Unimat spindle thread lap.

mount the faceplate to the cross slide. Be sure the surface of the. cross slide is free of chips and you'll get a perfectly vertical hole every time. Adjust the cross slide so the hole in the center of the faceplate is beneath the drill.

When drilling in metal, use a starting drill or center drill such as DB 1135 (supplied as part of DB 1130 Tool Box) after you center punch the workpiece. This type of drill is more rigid than a plain drill point and guarantees more perfect centering of the hole. Center drills should be used at slow speed, with plenty of lubricant.

A footswitch makes drilling of unclamped work a lot easier. Heavy-duty sewing machine motor footswitches can be used, or you can improvise your own by using an automobile floor switch mounted on a suitable piece of wood.

The Unimat handlever pinion give you a vertical movement of about 5/8-inch,



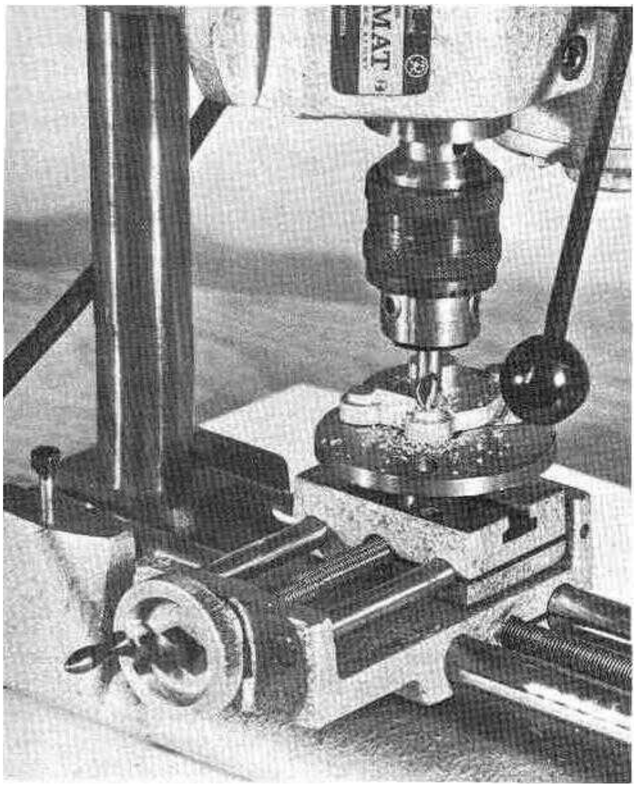
12D Simple setup quickly aligns vertical drill press.

enough for most kinds of precision model work. If you need more movement, you can remove the spindle return spring, increasing spindle movement to 1 inch.

Two basic kinds of drills are available: carbon and high speed steel. The less expensive carbon drills are intended for work in wood, plastics and soft metals, while the high speed drills are harder and will handle work in cast iron and

DRILL PRESS cont'd

drilling technique, types of drills, drilling speeds, miniature work..



13A Faceplate doubles as drill press table.

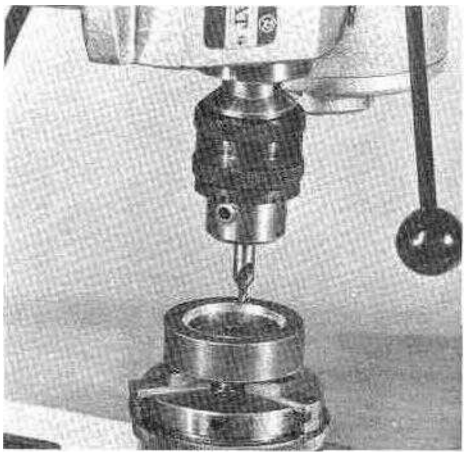
steel. Drills are sold in sets and singly. Sets are offered in fractional inch sizes, numbered wire sizes from 1 to 60. and 61 to 80, in letter and metric sizes. Usually sets of drills are supplied in drill cases with accurately sized holes that enable the user to keep the bits in order.

Drills commonly sold by hardware stores and tool supply houses are called jobbers length straight-shank twist drills. It is suggested that the Unimat owner order shorter length drill called "short set" or screw machine drills (DB 1140). Only about half as long as the regular jobbers length drills, this length gives you more tip rigidity and offers convenience in working. It's a good idea to contact the local office of the larger drill manufacturers. Request their latest catalog. Also, excellent manuals on drilling are usually given away free to drill users. Charts included give you complete information on all drill and tap sizes, cutting speeds, and decimal equivalents.

The correct speed for drilling is given by the formula: S.P.M. = .26 x R.P.M. x drill diameter in inches. S.F.M. refers to surface feet per minute, or the cutting rate of the drill tips against the work. Typical recommended S.F.M. rates for high speed steel drills are as follows:

alloy or stainless steel	20- 40
mild machinery steel	80-110
medium hard cast iron	70-100
brass and aluminum	200-300
bakelite	100-150
wood	300-400

If you use carbon steel drills, reduce cutting speeds by one-half.



13B 3-jaw chuck grips flywheel during drilling.

PROPER DRILLING TECHNIQUE

If a drill dulls rapidly at the edges or splits up the web, it usually means your speed was too high. Use a lubricant in all metals except cast iron. Chips should not be long and coiled since this tends to clog the point. A common mistake is to regard drilling as the easiest operation. The tendency is to rush through drilling jobs overfeeding the point and causing inaccurate holes and dulled cutting edges. Properly used, a quality high-speed drill should last for hundreds of holes in anything but the hardest metals.

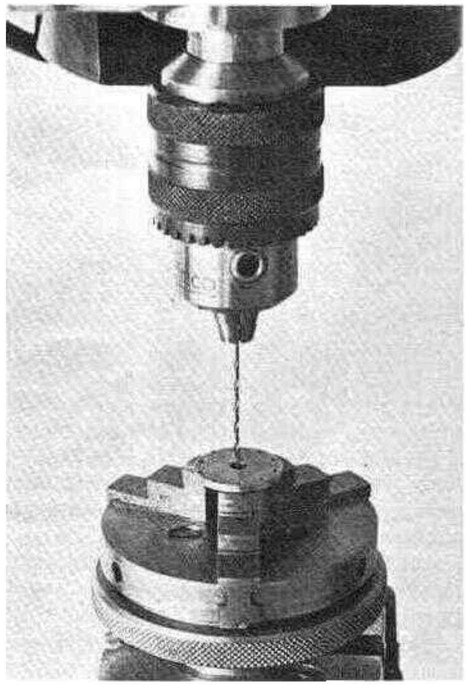
Sometimes the heat of a drill working in steel will "work harden" the hole. The drill will stop cutting and dull rapidly if allowed to run in the hole. The only way to avoid this trouble is to use turpentine, or water soluble oil when cutting hard steel.

Soft materials are drilled at higher speeds and so the tendency is to overheat the work often enlarging the hole. When you drill soft plastics, cut your speed and bring the tool in and out as quickly as possible in avoid melting the material with drill heat.

Drilling sheet metal is often a problem since the pressure of the point tends to distort the metal downward. Also, as the drill breaks through, it may grab on the work and break, or the workpiece can be pulled out of the hands with considerable danger. ALWAYS CLAMP SHEET METAL WORK SOLIDLY TO THE DRILL TABLE. Also, sheet, metal work should always be backed with a piece of hardwood or Masonite.

DEEP HOLE DRILLING

Model work often requires holes deeper than 5 times the drill diameter. This is called *deep-hole drilling* and requires special technique. Ordinarily the drill



13C Drilling with tiny #75 size drill bit.

flutes will pull the chips upward out of the hole, but when the hole is unusually deep, chips will tend to fall back in and clog the tip. The. remedy is to use ample lubricant which will help float the chips out. Abo back the drill out of the hole frequently during drilling. Especially in cast iron, a drill will tend to overheat in deep holes.

BASIC UNIMAT

DRILL PRESS cont'd

miniature hole drilling, drill sharpening, angle drilling...

Space age miniaturization requires new shop technique that just a few years ago were practiced only by dentists, jewelers, and instrument makers. In the ordinary machine shop a size 60 drill is considered a limit in smallness. Until the introduction of the Unimat, smaller drilling could be handled only on very expensive special machines.

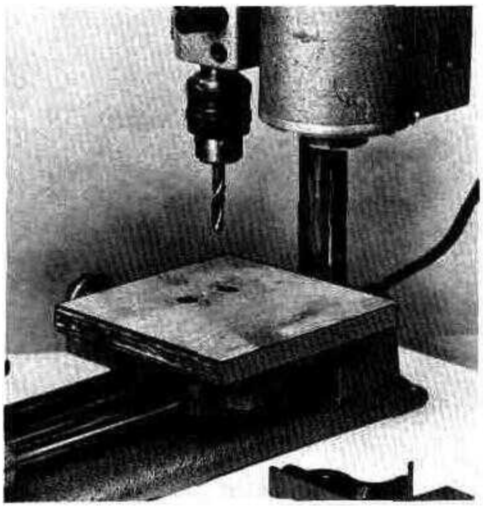
Practically all work with small drills below size 60 (.040-inch) should be handled like deep hole drilling. In proportion to its larger brothers, the miniature drill is very long. The ratio of length to diameter can be as much as 40 to 65 times, and yet the tiny drill must produce a hole in most jobs that is also proportionately many times deeper. Therefore a jeweler's touch is required or drill breakage is certain.

If a mechanic's stethoscope, or any kind of electronic amplifier is handy, use it, to listen to the sound of the tip cutting into the metal. Back the drill out frequently and use the cutting lubricant and an air hose or handy syringe to flush out chips.

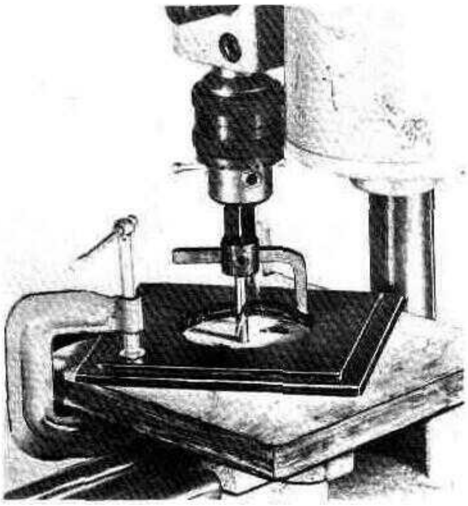
Always feed slowly, and if the drill tip dulls the least bit, immediately stop work and touch up the cutting edges using fine garnet paper under a magnifying glass. Using this technique, #75 size holes, only .021-inch diameter.were run through steel bars 3/8-inch thick. The standard DB 1005 Unimat drill chuck is capable of extreme precision, but it is important that you avoid keeping the chuck tightened on a drill overnight or for long periods of time. This can gradually distort the chuck jaws.

in place. Some workpieces can be held for drilling in the Unimat tool holder. Larger workpieces are best, supported on a work table (Photo 14A) that you can make of plywood or a piece of metal. Such a table supports a 4-inch lens board for drilling with a fly cutter (Photo 14B). Or you can use a pair of steel or plastic bars to support work such as the meter (Photo 14C) for drilling and countersinking. Bore a hole in the surface of the table and you can improvise a rotary sander (Photo 15A) for perfect edging of wood, plastic and metal. Clamp a fence to the table, and you can sand to any accurate width desired. In the same way, you can use your Unimat drill press to rout or carve.

Angle drilling (Photo 15B) is accomplished simply by loosening the headstock tapered clamping screw and setting the spindle axis against a protractor. Before drilling, chuck a test bar or drill blank and check against the work. Both the Unimat and the work should be securely clamped or screwed to the bench to prevent a spoiled hole. Photo 15D shows a method of clamping the Unimat drill press to a large workpiece. Steel bars, 1/4 x 1/2-inch were screwed to the lathe bed and in turn are clamped to the work. The hole is center punched



14A Plywood drilling table is 5 x 5 x 1/2-Inch.



14B Setup for flycutting camera lens board.

14C CountersInking holes in electronic meter case.

On any precise drilling job it is important that the workpiece be securely clamped or chucked as in the photos, page 13. Double check the cross slide and carriage locks. For miniature drilling, center punch the work lightly to avoid creating a hard spot in the metal that, might cause a small drill to deflect from its path.

Clean the drill chuck with kerosene and make certain there are no chips on the jaws. Insert the drill in the chuck as deeply as possible, but. *never grip any drill point, an the flul.es.* If the drill shank is marred by burrs, remove with a fine file and polishing cloth. Use a thinner lubricant than recommended for ordinary drilling, such as very light machine oil, kerosene, etc.

As the drill penetrates the work, watch the tip with a magnifying glass.

Drill Sharpening

Many machinists claim to be able to sharpen drills accurately by hand. While this can be. done on larger drills if one practices the art constantly, it is very difficult to hand grind the proper cutting tip angles and hold an accurate center. Factory-new drills will be accurate to within 1/4-thousandth-inch or better. A drill sharpening fixture (see Photo 16A) will help you to grind drills l/8-inch and larger. However if your work demands maximum precision, It is recommended that you take your drills to a sharpening shop that has the proper grinding equipment.

Work Holding Methods

The 3-jaw chuck also mounts on the cross-slide (Photo 13B) and gives you a means of clamping circular work such as rings, pulleys and camera lens flanges

and started with a center drill for dead true accuracy. Feed any angle drilling job slowly to avoid drift.

Drill Press Tapping

The skilled machinist always starts a threading tap with a drill press or lathe. If a tap enters a hole guided only by hand, it is *very easy* to start as many as 4 or 5 turns of the thread at a slight angle to the true axis of the hole. The result is a broken tap or spoiled work.

Start by drilling the hole the right size. Make sure the hole is clean and free of burrs and chips. Then, without changing the drill press setup, chuck the tap and using the handlever for light pressure, feed into the work, turning the chuck by hand. Do not use motor power.

After two or three threads are cut, lock the spindle, unchuck the tap, back the spindle off and finish with a hand tap