Millwright

Drilling

Machining

First Period
Drilling

Rationale

*Why is it important for you to learn this skill?*

Drilling is fundamental to the millwright trade, but it is not a simple or straight-forward operation. Many things can go wrong and it is up to you to make sure they do not. As well as drilling into a workpiece correctly and at the right feeds and speeds, you need to be safety-conscious at all times.

Outcome

*When you have completed this module, you will be able to:*

Demonstrate procedures for operating drilling machines.

Objectives

1. Describe safety, types, components and applications of drilling machines.
2. Describe workholding and toolholding devices.
3. Describe drilling tools and their applications.
4. Explain operation, speeds and feeds of drilling machines.
5. Describe care and maintenance of drilling tools and machines.

Introduction

This module addresses the drilling process, as well as the machines that are available, the tools that are used and how work is held. It also addresses the limitations of the tools and how to repair them when required.
Objective One

*When you have completed this objective, you will be able to:*

Describe safety, types, components and applications of drilling machines.

Safety First

You are using a rotating cutting tool and injury can happen easily. Follow these safety rules.

- If you are not thoroughly familiar with the operation of drill presses, obtain advice from your instructor or supervisor.
- Do not wear ties, rings, watches or other jewellery. Ensure to roll up your sleeves.
- Always wear safety glasses and a face shield.
- Guards must be in place at all times.
- Make adjustments with the power off.
- Ensure the drill bit or cutting tool is securely locked in the chuck.
- Ensure the chuck key is removed from the chuck before turning on the power.
- Adjust the table or depth stop to avoid drilling into the table.
- Hold the material securely with a vise or clamps.
- Disconnect the drill from the power source when making repairs. Also, put a lock on the machine so someone does not use it while you are away.
- Shut off the power, remove the drill bit or cutting tool and clean the table before leaving the machine.
- Do not leave a drill bit or other cutting tool in the machine when working on the set-up. All tool holding devices should be removed from the drill and put away when the job is done.
- Do not use table vises for anvils.
- If the taper shank of the drill does not seat easily inside the sleeve, the tapers are either dirty or burred. Clean and check them before proceeding.
- Check the drill tang for burrs and file it if necessary.
- Sleeves get nicked when they are dropped on the table, which makes them inaccurate or useless. Treat them like the high precision tools they are.

Types of Drilling Machines

Many different kinds of drilling machines are available because there are so many different kinds of holes. The drill presses commonly found in millwright shops are:

- sensitive drill presses,
- upright drill presses,
- hand drills,
- magnetic base drills and
- radial drilling machines.
**Sensitive Drill Presses**

A sensitive drill press (Figure 1) is meant for small, precise work (holes up to $\frac{1}{2}$ inch in diameter). This type of drill press has no power feed, so the operator must feed the drill into the workpiece by pulling on the handle. This allows the operator to sense the cutting pressure on the drill.

![Figure 1 - Sensitive drill press.](image)

Sensitive drill presses have the following features.

- The spindle on a sensitive drill press is usually driven by a V-belt, permitting the high speeds required by small drills. The speed can be set by moving the belt from one pair of sheaves to another.
- Most sensitive drill presses are equipped with a drill chuck only. Very few sensitive drill presses have a tapered spindle bore to accept a taper shank drill. Some have a chuck which is threaded into the spindle.
- The table on a sensitive drill press can be raised and lowered by hand and clamped in place. Slots or grooves are machined or cast into the table to accept a bolt and are used to secure the workpiece to the table.
- Most sensitive drill presses mount on the edge of a workbench, although some stand on the floor (Figure 1).
- Sensitive drill presses do not have coolant pumps, so the operator must apply coolant.

**NOTE**

When using a sensitive drill press, ease up on the hand feed pressure when the drill breaks through the workpiece; otherwise, the drill tends to grab the workpiece and can break.
Upright Drilling Machines

An upright drilling machine is like a sensitive drill press, but is larger and has power feed (Figure 2).

An upright drill press has the following features.

- An upright drill press is meant for drilling larger holes (½ inch to 4 inches) and usually has a geared head to permit the slow spindle speeds and high torque needed by large drills.
- Large drills require more feed pressure than a millwright can generate by hand, so an upright drill press has a power feed mechanism. The feed in inches per revolution can be set by manipulating the feed rate lever on the head. There are four or five feed rates from which to choose. The drill can still be fed by hand if the operator wishes.
- The spindle on an upright drill press accepts drills with tapered shanks.
- The table on an upright drill press can be raised and lowered on the column by turning the crank. The table has T-slots to accept T-bolts so work can be secured to the table. The T-slots and additional channels in the table are used to collect and return coolant to the coolant reservoir.
- The base of an upright drill press houses a pump and reservoir for coolant.

NOTE

The workpiece must always be clamped to the table when using power feed.
**Electric Hand Drills**

Electric hand drills are used in every millwright shop for drilling small holes (up to $\frac{1}{2}$ inch in diameter) by hand. Many holes do not require the precision or set-up time of drill presses. The hand drill is quicker and more versatile for drilling small non-vertical holes.

Electric hand drills have a drill body that contains the motor, a handle, a three-jaw chuck ranging in size from $\frac{1}{4}$ inch to $\frac{1}{2}$ inch and a power cord. Some are reversible, with a switch (sometimes on the back of the handle) for reversing the drill. Hand drills often have variable speeds. They may have an adjustment screw in the trigger, a variable speed trigger or an adjustment screw in the end of the handle.

Larger drills have larger chucks ranging from $\frac{1}{2}$ inch to $\frac{3}{4}$ inch and removable handles for applying feed pressure and controlling torque. Large hand drills often have two speeds in order to accommodate the slower speeds of large drill bits.

Electric hand drills are limited by the amount of feed pressure that can be applied for large-diameter drills. The feed pressure on small bits must be carefully controlled to avoid breaking them sideways. As well, they do not rotate fast enough for small drill bits. For example, a $\frac{1}{8}$ inch drill bit requires 3200 revolutions per minute (rpm).

Drilling freehand is dangerous on large-diameter holes (over $\frac{1}{2}$ inch). If a large diameter drill catches or binds in a hole, it can take the drill out of your hands. The drill must be very secure. For this reason, do not drill a large hole in stages, other than one pilot hole and the final size. It is difficult to keep your angle of drilling steady. If it is not kept steady, the drill bit may bind or break. Use a magnetic-base drill wherever possible on large diameter holes and when drilling out threads for Helicoils®.

<table>
<thead>
<tr>
<th><strong>NOTE</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>If you are left-handed, change the reversible handle to the other side of the drill for better support.</td>
</tr>
</tbody>
</table>
**Magnetic-Base Drills**

A magnetic-base drill is a heavy portable drill that is secured to the workpiece by means of an electro-magnetic base (Figure 3). The base can be switched on or off to secure the drill machine to any magnetic workpiece, at any angle. This enables you to do necessary drilling operations in place on heavy equipment without disassembling the housing, plate, frame or coupling for transport to the shop. In fact, some large workpieces do not fit into a radial arm drill press. Without a magnetic-base drill, you might require a boring mill to perform simple drilling operations.

Another advantage of the magnetic-base drill is its ability to drill, tap and ream large diameter holes requiring more control of the feed pressure, torque and drilling angle than are possible with a large hand drill.

![Figure 3 - A magnetic-base drill with centre finder.](image)

The drill head attaches to the column either by a dovetail slide with gibbs for adjustment or by clamping screws in the drill head (Figure 3). A gear rack on the column and the hand crank on the drill head are used for setting the height and the down feed pressure of the drill head. Large-diameter drills require more feed pressure than is possible with hand drills. The magnetic-base drill has a three-jaw chuck, usually 1/2 inch or 3/4 inch, for drilling holes from 1/8 inch to 1 1/2 inches in diameter. Some models also have a #3 internal Morse taper in the quill. Some are reversible and have two speeds to accommodate large and small drill bits. Some drills are equipped with centre finders (Figure 3).
The centre finder is flipped down to locate the machine before switching on the magnetic base and then flipped back out of the way for drilling. The magnetic base can vary in shape and size. Most machines have a light to indicate when the magnetic base is in operation.

- When operating a magnetic-base drill for horizontal drilling, ensure that your power source is secure and that the machine has a safety chain to guard against it falling on you. If your machine has gibs, ensure that the gibs are tight on the dovetail. If the drill head is loose on the column, the drill bit will wander off centre when down feed is applied.
- Use the centre finder if your machine is equipped with one.
- Ensure that the base and the workpiece plate are clean.
- Have lots of coolant on hand.
- Do not use too much feed pressure or the magnetic base will come loose.
- When the drill is about to break through the far side of the hole, let up on the feed pressure.

One limitation of magnetic-base drills is their lack of speed control. There may be only one or two speeds available for all sizes of drills, countersinks and reamers.

**Radial Drilling Machines**

Radial drilling machines (often called *radial-arm drill presses*) are used to drill holes in very large workpieces that are difficult to position manually under the spindle of a standard drill press.

The spindle on a radial drilling machine can be positioned anywhere within a certain radius of the column, depending on the size of the machine. The drilling head slides along the radial arm, thus allowing drilling head adjustment to the required distance from the column. Power is used to raise or lower the arm on the column by means of a lead screw. The radial arm pivots around the column to position the drill horizontally. The spindle can be operated by power feed or manually.

There are usually two methods of manual feed on radial-arm drill presses. One method uses a hand feed lever on the drilling head. Alternatively, a hand wheel located on the underside of the drilling head can be engaged to provide mechanical advantage and a fine feed for manual drilling operations.

A large workpiece can be fastened to the table or base or set on the shop floor beside the machine. When the workpiece has been secured, the spindle can be positioned above the location of the hole. Because these drill presses are meant for heavy work they always have geared heads, power feed and coolant pumps.
Sizing a Drill Press

Two ways are available to designate the size of a sensitive or upright drill press. Some drill presses are sized by the distance from the centre of the spindle to the column, while others use the largest diameter of workpiece that can be centred under the spindle axis (Figure 4). For example, one manufacturer may specify that a 10-inch drill press permits a hole to be drilled at the centre of a 10-inch diameter workpiece. Another manufacturer may specify that a 5-inch drill press will accommodate a 10-inch diameter workpiece.

![Diagram of a drill press sizing](image1)

Figure 4 - Sizing a drill press.

Radial arm drilling machine size is designated by the column diameter and by the greatest distance from the spindle axis to the edge of the column when the drilling head is extended to the maximum distance along the radial arm (Figure 5). Typical machines range in size from 3 to 14 ft, with column diameters of 30 inches and more.

![Diagram of a radial drilling machine](image2)

Figure 5 - Radial drilling machine.
Objective Two

When you have completed this objective, you will be able to:

Describe workholding and toolholding devices.

Holding Devices

Tool holding devices secure a cutting tool in the end of a drill press spindle. A good tool holding device has the following characteristics.

- It holds the tool securely.
- It aligns the axis of the tool with the axis of the spindle.
- It is quick and easy to use.

No single tool holding device satisfies all of these requirements. For example, a Morse taper holds the tool securely, but is not as quick to use as a drill chuck. You must choose the tool holding device that best suits the job.

Drill Chucks

Drill chucks have three jaws that may be adjusted in and out to grip onto the shank of a drill, reamer or other cutting tool. These jaws move simultaneously and, when in good condition, hold the tool securely and accurately (Figure 6). Drill chucks are sized according to the largest drill they can hold. For example, a 1/2 inch chuck holds a drill up to 1/2 inch in diameter.

![Figure 6 - Drill chuck.](image)

Drill chucks are the most widely used tool holding devices on drilling machines because they offer the following advantages.

- A drill chuck can hold any number of different cutting tools within its range.
- Drill chucks are used with cutting tools having straight shanks, which are less expensive than cutting tools having tapered shanks.

On the other hand, there are a number of disadvantages to using drill chucks.

- Drill chucks lose their accuracy when their jaws become worn.
- Drill chucks do not hold cutting tools as securely as other methods.
- It takes time to replace one cutting tool with another when using a drill chuck, especially if their shanks are of different diameters.
Key Type Drill Chucks

Key type drill chucks are tightened with a key (Figure 7).

**CAUTION**

Use a key that is properly sized for the chuck; otherwise, you risk damaging the teeth.

Keyless Drill Chucks

*Keyless* drill chucks (Figure 8) are tightened by turning the collar. They do not grip tools as tightly as key type chucks, but hold well enough for light work. These types of chucks are quicker and more convenient to use than key type chucks.

**Figure 7 - Key type drill chuck.**

**Figure 8 - Keyless chuck.**
Drill Sleeves and Sockets

The shanks of drills larger than \( \frac{1}{2} \) inch are tapered, which allows them to mate with the tapered bore of the drill press spindle. These tapers are not all the same size, with larger drills and spindles having larger tapers. The difference in size can be made up with a drill sleeve (Figure 9), which is designated according to the size of the tapers it will bridge. For example, a 2-3 Morse taper sleeve will fit a drill with a #2 taper and a spindle with a #3 taper bore. Several sleeves can be used together to bridge a large difference in size. Sometimes, the tapered shank of a drill is larger than the tapered bore of the drill press. In these cases, a drill socket is used to make up the difference.

Morse tapers are separated using a tapered tool called a drift (Figure 10). Insert the drift in the slot and tap it with a hammer. When separating a tapered tool from the spindle of a drill press, place a piece of wood or soft metal on the table under the spindle to prevent the tool from damaging the table if it drops unexpectedly.
Quick-Change Toolholders
Quick-change toolholders are used when a sequence of operations must be performed repeatedly (Figure 11). Each tool is mounted in its own socket and the sockets are installed and removed rapidly by lifting the collar on the holder.

![Figure 11 - Quick-change toolholder.](image)

Power Tapping Attachments
A power tapping attachment has a number of features that allow a hole to be tapped in a drill press (Figure 12).

- When the down feed pressure is released (that is, when the tap is finished), the tapping attachment reverses the rotation of the tap, which allows it to turn back out of the hole.
- An adjustable clutch slips if the tap binds. This prevents the tap from breaking.
- The tap is held in a floating head so it can align itself accurately with the hole.

![Figure 12 - Power tapping attachment. (Courtesy of DoALL Industries)](image)
**Floating Reamer Holders**

The axis of a reamer must be aligned with the axis of the hole; if it is not, the reamer deflects and the hole loses its accuracy (Figure 13). A floating reamer holder reduces the effect of reamer misalignment because it is self-centring. When entering the hole, the floating reamer holder allows the axis of the reamer to move into alignment with the drill press spindle axis.

![Figure 13 - Reamers, aligned and misaligned.](image-url)
Workholding Devices
A wide range of workholding devices are needed to secure the various workpieces that must be drilled. Table 1 addresses common workholding devices and their uses.

<table>
<thead>
<tr>
<th>Device</th>
<th>Use</th>
</tr>
</thead>
<tbody>
<tr>
<td>vises</td>
<td>Hold work of almost any size or shape and are the most versatile and convenient work holding devices. Vises include standard drill press vises, angle vises and contour vises.</td>
</tr>
<tr>
<td>clamping sets</td>
<td>For large and irregularly shaped workpieces. These types of workpieces are not suitable for holding in a vise and must be clamped directly to the table. A clamping set includes a combination of strap clamps, parallels step blocks and spacer blocks.</td>
</tr>
<tr>
<td>jigs</td>
<td>Special workholding devices used to guide the drill when drilling a large number of identical workpieces.</td>
</tr>
<tr>
<td>other workholding devices</td>
<td>Other types of workholding devices that support, hold and clamp the work in the correct position include jacks, C-clamps, V-blocks and angle plates.</td>
</tr>
</tbody>
</table>

Table 1 - Uses of workholding devices.

Standard Drill Vises
A standard drill press vise holds the workpiece between a movable jaw and a fixed jaw. A screw thread transmits motion to open and close the vise. The handle attaches to the screw thread, which connects on the opposite end to the moveable jaw. Tighten the vise by rotating the handle until the workpiece is held securely between the jaws.

The vise in Figure 14 shows where improper depth of drilling has allowed the drill to damage the vise.

CAUTION

Do not allow the drill to proceed to a depth that will damage the vise.

Figure 14 - Standard drill press vise.

The base is a light casting, which is machined on both the top and bottom. There are usually lugs on either side so it can be bolted to the table of the drill press. The middle of the base is cut away to provide clearance for the drill.
The jaws are made of hardened steel. There are often grooves cut on the face of the jaws to allow them to grip more securely (Figure 15).

![Figure 15 - Step in jaws for gripping thin workpieces.](image)

**NOTE**

Many millwrights cut a small step at the top of the jaws to hold thin workpieces. The steps hold the base of the workpiece parallel to the base of the vise without the use of parallels.

**Angle Vises**

An angle vise is much like a standard drill press vise, except it is hinged to a base so it can be swivelled and set at an angle (Figure 16).

![Figure 16 - The angle vise.](image)

**C-Clamps**

C-clamps are convenient and quick to use. They are especially suited to smaller drill presses that do not have T-slots or grooves in the table. Use two C-clamps because, when only one is used, the workpiece can pivot around the clamping point.
**V-Blocks**

V-blocks are used to hold round work (Figure 17). They are usually sold in matching pairs so shafts can be supported at both ends. Many V-blocks come with a clamp that can be used to secure the workpiece.

![Figure 17 - V-block with clamp.](image)

**Drill Jigs**

A drill jig is a special work holding device used to locate a hole accurately on a workpiece (Figure 18). It consists of a base or a frame, which locates and holds the workpiece. A hardened bushing is used to guide and locate the drill. On small drill jigs, the workpiece is held in place with a thumbscrew. Cams, pneumatic cylinders and toggle clamps are used on larger jigs. Drill jigs are expensive and time consuming to make and can be justified only when many identical parts are to be drilled.

![Figure 18 - Drill jig.](image)
Objective Three

When you have completed this objective, you will be able to:

Describe drilling tools and their applications.

Twist Drills

A twist drill (Figure 19) is divided into three parts, which are the:

- point,
- body and
- shank.

Shank

A twist drill is held by its shank. The two types of shanks are straight shanks and taper shanks (Figure 20).

Straight Shanks

Straight shanks are used on drills up to $\frac{1}{2}$ inch in diameter. A straight shank is less expensive to manufacture, but cannot be held as securely as a taper shank.

Taper Shanks

Taper shanks are found on drills larger than $\frac{1}{2}$ inch in diameter. Taper shanks have a Morse taper, which is a standard taper of about $\frac{5}{8}$ inch taper per foot. This self-locking taper drives the drill. The tang on the end of the shank fits into a slot in the drill spindle. The combination of the tang and taper shank hold the drill securely in the spindle and allow the drill to be changed quickly.

CAUTION

The tang is for removal only, not for driving.
Body
The body of a twist drill has two helical flutes (grooves) cut into it (Figure 21). These grooves carry chips up and out of the hole, provide passage for cutting fluid and coolant and allow the correct cutting geometry at the point. Most drills have a right-hand helix.

<table>
<thead>
<tr>
<th>NOTE</th>
</tr>
</thead>
<tbody>
<tr>
<td>When you look at the side or end view of the drill, the helix falls off to the right.</td>
</tr>
</tbody>
</table>

Figure 21 - Body.

Margin
The margin is a narrow raised section that runs along the flutes for their entire length. The diameter of the drill is measured across the margins. The body of the drill is cut back behind the margins to provide clearance.

Point
The point of the drill is where all the cutting action takes place (Figure 22). The geometry of the point is defined by the cutting lips, chisel edge and lip clearance.

- The *lip or cutting edge* extends from the chisel edge to the margin and is the portion of the point that does the cutting.
- The *chisel edge* is the portion remaining between the flutes. It does not cut and should be as small as possible.
- *Lip clearance* is provided by grinding the *heel* lower than the cutting edge. Without this clearance, the drill would not be able to advance into the workpiece.

Figure 22 - Drill point.
Specialty Drills

Twist drills are not suitable for certain drilling jobs. Other specialized drills must be used.

- **High helix drills** (35° to 40°) are used for deep holes in softer metals like aluminum, copper and zinc (Figure 23). These metals tend to jam in the flutes of a normal drill.

  ![Figure 23 - High helix drill.](image)

- **Core drills** are used to enlarge existing holes. They have three or four flutes and cannot cut at their centre (Figure 24). The greater number of flutes allows higher feed rates and provides a better finish.

  ![Figure 24 - Core drill.](image)

- **Oil hole drills** have one or two small holes that run the length of the drill and exit just behind the cutting edge (Figure 25). Cutting oil, coolant or compressed air is forced through these holes to keep the cutting edges cool and help flush chips back out of the hole. They are especially useful on deep holes where it would otherwise be difficult to deliver cutting fluid to the point of the drill.

  ![Figure 25 - Oil hole drill.](image)

- **Straight fluted drills** are used on soft materials such as brass, bronze and plastic (Figure 26). Normal drills tend to draw themselves (dig) into these materials. The straight flute helps prevent this draw down.

  ![Figure 26 - Straight fluted drill.](image)

- A twist drill can be modified to drill soft materials by grinding a small flat on the cutting edge parallel to the axis of the drill.
- *Step drills* are used to drill and then countersink (or counterbore) a hole all in one operation (Figure 27). The shoulder may be square or angular, depending on the purpose of the drill. Set the depth stop when using a step drill.

![Figure 27 - Step drill.](image)

- A *saw-type hole cutter* consists of a cylindrical cutter with a standard twist drill in the centre (Figure 28). The teeth on the cutter are like teeth on a saw. The purpose of the twist drill is to drill a pilot hole to guide the cutter. These drills are used for drilling thin material because they are much less likely to grab than a regular twist drill and they make a cleaner hole.

![Figure 28 - Saw-type hole cutter.](image)

- A *masonry drill* is used for drilling concrete and stone to install anchors.
- A *centre drill* is a combination drill and countersink used primarily to drill centre holes for lathe work (Figure 29). Centre holes may also be used to spot the location of a hole to be drilled on a drill press.

![Figure 29 - Centre drill.](image)

<table>
<thead>
<tr>
<th>NOTE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Both ends of a centre drill cut, so when one end breaks or becomes dull, the drill may be flipped around and used again. When using a center drill, you must set the correct rpm or you will dull or break the bit.</td>
</tr>
</tbody>
</table>

© 2013, Her Majesty the Queen in right of the Province of Alberta
**Sizing of Drills**

Drills come in standard sizes. You can usually find a drill that is close enough to the desired diameter.

- **Fractional size** drills are available in diameters from 1/16 of an inch to 4 inches. They increase in size by increments of 1/64 of an inch, up to a diameter of 1 3/4 inch. Then, they increase by increments of 1/32 of an inch.
- **Number size** drills range from #97 (0.0059 of an inch in diameter) to #1 (0.228 of an inch in diameter).
- **Letter size** drills range from 0.234 of an inch to 0.413 of an inch in diameter and are identified by a letter ranging from A (smallest) to Z (largest). The letter range picks up where the number range leaves off.
- **Metric size** drills range from 0.04 mm to 80 mm in diameter.

**Use of a Drill Gauge**

Measuring a drill with a micrometer or vernier is time consuming and you must ensure that you measure directly across the margins. A faster and more accurate way of checking the size of a drill is to use a drill gauge (Figure 30). The smallest hole that the drill fits into on the drill gauge is the size of the drill.

![Figure 30 - Using a drill gauge.](image-url)
Objective Four

When you have completed this objective, you will be able to:

Explain operation, speeds and feeds of drilling machines.

Drill Press Operations

A drill press is extremely useful because it can perform so many operations. Besides drilling, it can ream, countersink, counterbore, tap, bore, spotface and saw a hole.

Drilling

Drilling is the process of making a new hole in metal. Most drilling is done with a cutting tool that has two or more cutting edges. The cutting action penetrates the workpiece and the chips escape through the drill flutes. Friction from the cutting action creates heat that is harmful to the operation. The ejection of chips from the flutes makes lubrication and cooling difficult at the point where it is most required. Despite these difficulties, drilling is considered an efficient operation that removes metal in a short time period.

The six approaches to drilling a hole are:

- drilling work held in a vise,
- drilling to an accurate layout,
- drilling large holes,
- drilling holes in mating parts,
- drilling round work and
- deep hole drilling.
Drilling Work Held in a Vise

The most common way to drill a hole in a workpiece is to hold the workpiece in a vise and position the vise under the spindle of a drill press (Figure 31). Large workpieces may clamp directly to the table, but the basic principle of drilling remains the same. This is suitable for rough work, where speed matters more than accuracy.

Figure 31 - Drilling work held in a vise.

To drill a workpiece held in a vise (or clamped directly to the table), follow this procedure.

1. Mark the location of the hole with a centre punch. The centre punch mark should be at least as large as the chisel edge of the drill (Figure 32).

Figure 32 - The centre punch mark size.
2. Clean the vise and table.
3. If you are drilling completely through the workpiece, make sure there is clearance between the workpiece and the vise under the hole (Figure 33). If there is not clearance, use parallels to prevent drilling holes in the vise.

![Figure 33 - Ensure proper clearance under the workpiece.](image)

<table>
<thead>
<tr>
<th>CAUTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>The drill vise in Figure 33 must have C-clamps or bolts to secure it to the table.</td>
</tr>
</tbody>
</table>

4. Place the workpiece in the vise and tighten it. Use a soft-faced hammer to drive the workpiece down flat against the bottom of the vise (Figure 34).

![Figure 34 - Drive the work against the bottom of the vise.](image)
5. Position the workpiece under the centre of the spindle. Turn the spindle in reverse and carefully lower the drill into the centre punch mark. When the drill engages the mark, it will draw the workpiece into alignment (Figure 35). If the work is too heavy to be drawn in by the drill, note which way the drill deflects and push the work into alignment yourself.

![Workpiece Moves into Alignment](image)

**Figure 35 - Drawing the workpiece into alignment.**

6. Raise the drill, being careful not to move the vise. Clamp the vise in place with a C-clamp or hold down bolts.

7. Set the correct spindle speed and feed rate and direct a stream of coolant at the point where the drill enters the work.

8. Drill the hole.

<table>
<thead>
<tr>
<th>NOTE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Do not hold the workpiece or vise in place by hand. Even small drills sometimes grab onto workpieces.</td>
</tr>
</tbody>
</table>

For small holes (under 3/8 inch in diameter), it is acceptable to clamp a stop to the table to prevent the vise from rotating, instead of clamping the vise itself (Figure 36). This has the advantage of allowing the vise to float into position under the point of the drill, but remember that a drill that grabs can still pull the workpiece up and over the stop. Twist drills have a tendency to grab when drilling soft materials. To prevent grabbing, a straight fluted drill is recommended. A conventional drill can be modified by grinding a 1/16 inch flat on both lips of the drill, which provides a zero rake angle.

![Stop and Rotation of Vise](image)

**Figure 36 - Using a stop.**
Drill Large Holes

Large drills need more feed pressure than small drills and are more likely to wander when they start. One way to minimize wandering is to drill a pilot hole (Figure 37), which should be just a little larger than the chisel edge of the drill. If the pilot hole is too large, the drill tends to chatter. Holes over $\frac{1}{4}$ inch in diameter require a pilot hole.

![Figure 37 - Pilot hole.](image)

Feed the drill carefully by hand and engage the power feed only when the drill cuts to the full diameter. A large drill tends to chatter if the power feed is engaged when the drill first starts cutting. Always clamp the workpiece to the table when drilling into a pilot hole; otherwise, the drill can grab the hole and rotate the workpiece.

Drill Holes in Mating Parts

Most holes are drilled to accept fasteners, whether they are screws, dowel pins, rivets or something else. These holes must line up accurately with their counterparts in the mating piece or the fastener will not fit (Figure 38).

![Figure 38 - Holes must align in mating parts.](image)
The best method for drilling holes in mating parts is to clamp the two pieces together and drill them as one (Figure 39). The holes cannot be misaligned that way. However, this works only if neither piece has been drilled yet.

![Figure 39 - Clamping the pieces together and drilling as one.](image)

If one mating part has already been drilled, then the holes must transfer from one piece to the other. The three methods used for hole location transfer are the:

- twist drill method,
- transfer punch method and
- transfer screw method.

**Twist Drill Method**

Clamp the two pieces together and use the existing hole to guide the drill (Figure 40). If the hole to be drilled is smaller or larger than the existing hole (perhaps for tapping or reaming), drill only as deep as the point of the drill. Then, unclamp the workpieces, change the drill and complete the hole.

![Figure 40 - Using existing holes to guide a drill.](image)
**Transfer Punch Method**
Clamp the two pieces together and use a transfer punch to mark the location of the hole on the undrilled workpiece (Figure 41).

![Figure 41 - Using a transfer punch to mark the hole location.](image)

**Transfer Screw Method**
If the hole is a blind tapped hole, use transfer screws to mark the location of the hole on the other workpiece (Figure 42).

![Figure 42 - Using transfer screws to mark hole locations.](image)
Drill Round Work

Round work presents some special challenges.

1. The centre punch mark must be deep; otherwise, the drill tends to slide off the workpiece (Figure 43).

![Figure 43 - Drill sliding off a round workpiece.](image)

2. The work must be clamped so that the centre punch mark is centred above the axis of the shaft. If the centre punch mark is not aligned with the axis of the shaft, the hole will be off centre. Position the centre punch mark using a square and scale (Figure 44).

![Figure 44 - Aligning the centre punch mark with the axis of the shaft.](image)

3. Round work can be held in a vise, provided that the height of the jaws is greater than the radius of the workpiece (Figure 45). Otherwise, the work must be held in a V-block.

![Figure 45 - The workpiece on the left is too large for the vise.](image)
Countersinking

Countersinking is the process of cutting a bevel at the top of the hole so the hole can accept the head of a flat head screw (Figure 46). The point angle of a countersink should match the head of the flat head screw. Most screws have an included angle of 82°.

Figure 46 - Countersinking.

To countersink a hole, follow these steps.

1. Secure the workpiece in a vise. When countersinking small holes, a stop is often fastened to the drill press table to prevent the vise from spinning. Clamp the vise in place if the hole being countersunk is very large (over 3/4 inch). In cases where a stop is used, a better finish is usually obtained because the workpiece is allowed to float.
2. Choose a countersink with the correct included angle and secure it in the spindle of the drill press.
3. Set the spindle speed to approximately one quarter of what you would use for drilling.
4. Apply cutting fluid liberally to the countersink.
5. Start the spindle and firmly feed the countersink into the work. The countersink tends to chatter if the feed pressure is too light.
6. Check the depth of the countersink with the head of the screw. The top of the hole should be slightly larger than the diameter of the head.
7. If several holes are to be countersunk on the same workpiece, set the depth stop so subsequent holes have the same appearance.

NOTE

A countersink can also be used to put a small chamfer at the top of a drilled hole, either to improve its appearance or to make it easier to start a tap.
Counterboring

Counterboring is the process of enlarging the top of an existing hole to allow it to accept the head of a bolt or screw (Figure 47). A pilot guides the counterbore into the hole; the pilot is usually between 0.002 and 0.005 of an inch smaller than the hole diameter. The initial hole is usually drilled to a nominal size. Some counterbores have pilots that are interchangeable. These types of counterbore provide a variety of pilot-to-counterbore diameter combinations. A counterbore diameter is usually $\frac{1}{32}$ of an inch larger than the bolt head diameter.

The counterbored hole has a flat bottom and is made deep enough so the head of the bolt or screw sits below the surface of the work.

![Figure 47 - Counterboring.](image)

To counterbore a hole, follow these steps.
1. Hold the workpiece in a vise, just as you would for drilling.
2. Secure the counterbore in the spindle of the drill press.
3. Position the hole to be counterbored under the counterbore. Turn the spindle by hand while lowering the pilot of the counterbore into the hole. When the hole is properly aligned, raise the counterbore and clamp the vise in place.
4. Set the spindle speed to approximately one quarter of what you would use for drilling.
5. Apply cutting oil generously. When cutting dry with materials such as brass or cast iron, always apply a few drops of lubricating oil to the pilot.
6. Turn on the spindle and feed the tool into the work. Apply firm and consistent pressure.
7. Withdraw the tool periodically to check the depth of the counterbore. When multiple holes must be drilled, set the drilling machine depth stop. The counterbore is deep enough when the head of the bolt or screw lies just below the surface of the work.
8. If several holes are to be counterbored on the same workpiece, set the depth stop so subsequent holes have the same appearance.
Tapping

Tapping is the process of cutting threads in a drilled hole (Figure 48). Tapping can be done by hand, but when done in the drill press, the tap is held square to the work.

Figure 48 - Tapping.

To tap a hole, follow these steps.
1. Hold the work in a vise, just as you would for drilling a hole.
2. Secure the tap in a tapping attachment.
3. Set the spindle speed.
4. Position the workpiece under the tap. Turn the spindle backward by hand while lowering the tap into the drilled hole. When the hole is properly aligned, raise the spindle and clamp the workpiece to the table. The floating head of the tapping attachment can tolerate some misalignment.
5. Apply cutting fluid.
6. Turn on the spindle and feed the tap into the hole.
7. When the tap is deep enough, feed the tap back out of the hole. The tapping attachment reverses the direction of the tap when it is fed upward.

A tap can be turned manually with a tap handle if a tapping attachment is not available. Put a centre in the drill chuck to keep the tap in alignment and maintain steady downward pressure on the spindle to keep the centre engaged with the tap (Figure 49). After drilling the hole to the tap drill size, try to tap the hole immediately without moving the set-up. This saves set-up time and ensures that the tap is accurately aligned.
One of the difficulties encountered in machine tapping is breaking and clearing the chips. Spiral fluted and spiral point (gun) taps clear the chips from the hole. A straight fluted tap does not clear the chips and may jam and break (Figure 50).

Figure 49 - Supporting a tap with a centre.

Figure 50 - Using the correct tap.
**Hole Sawing**

*Hole sawing* is the process of cutting a large hole in thin stock using a pilot drill and a round saw (Figure 51). The pilot drill acts as a guide to keep the saw from wandering as it enters the work.

![Figure 51 - Using a hole saw.](image)

To use a hole saw, follow these steps.

1. Position the work under the hole saw and clamp it in place. Make sure there is clearance under the hole to be cut.
2. Set the spindle speed as you would for drilling.
3. Start the spindle and feed the hole saw into the work by hand. Ease off the feed pressure when the pilot drill exits the work and again when the saw exits the work or the hole saw will grab.

<table>
<thead>
<tr>
<th>NOTE</th>
</tr>
</thead>
<tbody>
<tr>
<td>You can sometimes obtain a better result if you drill the pilot hole separately and use a solid pilot with the saw to replace the drill.</td>
</tr>
</tbody>
</table>
**Speeds and Feeds**

Metal must be drilled and machined at the most profitable speed. If the speed is too slow, the job takes too long. If the speed is too fast, the cutting tools wear prematurely and must be replaced.

**Cutting Speed**

*Cutting speed* is the rate at which the cutting tool moves through metal and is measured in *metres per minute* (m/min) or *surface feet per minute* (sfpm). Imagine a tool being dragged in a straight line across a slab of metal so that it cuts a groove. As the tool moves through the metal, heat is generated at the cutting edge. The faster the tool moves through the metal, the more heat is generated.

If you move the tool too fast, it eventually generates too much heat and loses its hardness. On the other hand, if you move the tool too slowly, it takes a very long time to make the cut. The trick is to move the tool at just the right speed so that it cuts metal as fast and efficiently as possible, without being damaged by heat. The most efficient cutting speed depends on a number of factors (Table 2).

<table>
<thead>
<tr>
<th>Factor</th>
<th>Effect on Cutting Speed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type of metal</td>
<td>More heat is generated when cutting a hard metal, so the tool must move more slowly in order to compensate. For example, a tool used to cut steel will have a lower cutting speed than the same tool used to cut aluminum.</td>
</tr>
<tr>
<td>Type of tool</td>
<td>Certain types of tools withstand heat better than other types and so have a higher cutting speed. For example, carbide tools cut steel at a higher speed than <em>high-speed steel</em> (HSS) tools because they are not as susceptible to heat damage.</td>
</tr>
</tbody>
</table>

**Table 2 - Cutting speed factors.**
Interpret a Cutting Speed Chart

The most efficient cutting speed for any combination of cutting tool and metal can be determined by referring to Table 3. To use this chart, cross-reference the type of material being drilled with the type of tool being used and read the most efficient cutting speed for that particular combination.

<table>
<thead>
<tr>
<th>Type of Material Being Cut</th>
<th>High-Speed Steel</th>
<th>Carbide</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>sfpm</td>
<td>m/min</td>
</tr>
<tr>
<td>Mild steel</td>
<td>100</td>
<td>30</td>
</tr>
<tr>
<td>Alloy steel</td>
<td>60</td>
<td>18</td>
</tr>
<tr>
<td>Cast iron</td>
<td>80</td>
<td>25</td>
</tr>
<tr>
<td>Aluminum</td>
<td>250</td>
<td>75</td>
</tr>
<tr>
<td>Brass</td>
<td>200</td>
<td>60</td>
</tr>
</tbody>
</table>

Table 3 - Approximate cutting speeds for drilling.

Calculate a Spindle Speed

Cutting speed must not be confused with spindle speed or rpm. The cutting speeds in Table 3 are linear. They show how fast the tool moves through the metal in a straight line. The linear value must be converted into a rotational value (spindle speed). The drill press can be set at the proper rpm to obtain the correct cutting speed for each drill size.

The fastest moving part of a rotating object (like a drill or milling cutter) is the outside edge because it has to travel the greatest distance in each revolution (Figure 52). This is often called the peripheral speed.

Figure 52 - Outside edge (periphery) moves the fastest.
The cutting speed must be correct for the outside edge, even though it means that the middle portion of the drill moves too slowly. If the cutting speed is correct for the middle, the outside edge will turn much too fast and burn out.

Use the following formula to calculate precisely how fast you must turn a drill to make its outside periphery move at the desired cutting speed.

\[
\text{rpm (imperial) = \frac{\text{cutting speed} \times 4}{\text{diameter of the drill}}}
\]

or

\[
\text{rpm (metric) = \frac{\text{cutting speed} \times 320}{\text{diameter of the drill}}}
\]

**Use a Chart to Determine Spindle Speed**

A chart like the one in Table 4 can be used to determine spindle speed. Cross-reference the cutting speed (peripheral speed in feet per minute [fpm]) with the diameter of the drill and read the rpm directly off the chart. For example, using the chart, a \(\frac{3}{8}\) -inch diameter drill with a cutting speed of 100 fpm should be rotated at 437 rpm.

<table>
<thead>
<tr>
<th>FT. PER MINUTE (PERIPHERAL)</th>
<th>DIAMETER IN INCHES</th>
</tr>
</thead>
<tbody>
<tr>
<td>1/4</td>
<td>3/8</td>
</tr>
<tr>
<td>REvolutions per minute</td>
<td></td>
</tr>
<tr>
<td>40</td>
<td>611</td>
</tr>
<tr>
<td>50</td>
<td>763</td>
</tr>
<tr>
<td>60</td>
<td>916</td>
</tr>
<tr>
<td>70</td>
<td>1070</td>
</tr>
<tr>
<td>80</td>
<td>1222</td>
</tr>
<tr>
<td>90</td>
<td>1375</td>
</tr>
<tr>
<td>100</td>
<td>1528</td>
</tr>
<tr>
<td>125</td>
<td>1908</td>
</tr>
<tr>
<td>150</td>
<td>2290</td>
</tr>
<tr>
<td>175</td>
<td>2671</td>
</tr>
<tr>
<td>200</td>
<td>3056</td>
</tr>
<tr>
<td>250</td>
<td>3850</td>
</tr>
<tr>
<td>300</td>
<td>4584</td>
</tr>
<tr>
<td>400</td>
<td>6112</td>
</tr>
<tr>
<td>500</td>
<td>7469</td>
</tr>
<tr>
<td>600</td>
<td>9106</td>
</tr>
<tr>
<td>700</td>
<td>10667</td>
</tr>
<tr>
<td>800</td>
<td>12214</td>
</tr>
<tr>
<td>900</td>
<td>13740</td>
</tr>
<tr>
<td>1000</td>
<td>15267</td>
</tr>
<tr>
<td>1200</td>
<td>18321</td>
</tr>
<tr>
<td>1400</td>
<td>21374</td>
</tr>
<tr>
<td>1600</td>
<td>24427</td>
</tr>
<tr>
<td>1800</td>
<td>27481</td>
</tr>
<tr>
<td>2000</td>
<td>30534</td>
</tr>
</tbody>
</table>

*Table 4 - A spindle speed reference chart.*
Feed Rates

*Feed rate* (Figure 53) is the rate at which the drill is fed into the work and is measured in *inches* (or *millimetres* per revolution) (ipr).

![Figure 53 - Feeding a drill into the work.](image)

If the feed rate is too high, the cutting edges of the drill will chip and the drill could break. If the feed rate is too slow, the cutting edges tend to rub rather than cut and production is slowed down. The feed rate for carbide drills is less than that for high-speed steel drills because carbide is not as tough as high-speed steel.

The larger the drill, the higher the feed rate, as indicated in the drill feed table (Table 5). Larger drills are better able to withstand the greater stress of high feed rates.

Interpret a Chart or Table

To determine the correct feed rate, consult a chart such as Table 5.

<table>
<thead>
<tr>
<th>Drill Size</th>
<th>Feed per Revolution</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inches</td>
<td>Millimetres</td>
</tr>
<tr>
<td>1/8 inch and smaller</td>
<td>3 mm and smaller</td>
</tr>
<tr>
<td>1/8 inch to 1/4 inch</td>
<td>3 mm to 6 mm</td>
</tr>
<tr>
<td>1/4 inch to 1/2 inch</td>
<td>6 mm to 13 mm</td>
</tr>
<tr>
<td>1/2 inch to 1 inch</td>
<td>13 mm to 25 mm</td>
</tr>
<tr>
<td>1 inch to 1 1/2 inches</td>
<td>25 mm to 38 mm</td>
</tr>
</tbody>
</table>

*Table 5 - Feed rate.*

The values in this table are for general-purpose work on mild steel. When drilling hard or tough material, like tool steel, a lower feed rate may be used than indicated in the table. When drilling softer material, like aluminum, the feed rate may be increased beyond the specified range.
Objective Five

When you have completed this objective, you will be able to:

Describe care and maintenance of drilling tools and machines.

Grind a Drill

Drills become dull quickly, so sharpening them is a fundamental skill. Until recently, drill sharpening was done by hand at a bench grinder. Now, there are bench grinding attachments and drill sharpening machines that can do the job better.

Grind a Drill by Hand

Small drills can be sharpened well enough by hand, using an off-hand grinder (Figure 54). Although it is not the most accurate method, it is by far the quickest. With practice, you can achieve consistent results.

Figure 54 - Grinding a drill by hand.
You must consider three elements of the point geometry when grinding a drill.

1. The cutting lips must be of the same length. If not, the drill cuts oversize (Figure 55).

![Figure 55 - Cutting lips of unequal length.](image)

2. The point angle must be even. If it is not, the smaller angle does all the cutting (producing a bell mouthed hole) and the drill lasts half as long as it should (Figure 56).

![Figure 56 - Uneven point angle.](image)

3. There must be the correct amount of clearance behind the cutting edge (Figure 57). If there is not enough clearance, the drill cannot be fed into the work efficiently. If there is too much clearance, the cutting edges lack sufficient support and chip easily. The clearance angle for general-purpose work is 8° to 12°.

![Figure 57 - Clearance.](image)
To sharpen a drill by hand, use this procedure.

1. Take all steps necessary to ensure your safety while operating the grinder. Wear eye protection and inspect the grinder to ensure that the wheel is in good condition and the tool rest is adjusted to \( \frac{1}{16} \) inch of the wheel face.
2. Scribe a line on the tool rest at 59° to the axis of the grinding wheel and use it as a visual guide for the angle of the drill bit as you grind.
3. Hold the drill near the point with one hand and near the shank with the other. Adjust the drill so the shank is slightly lower than the point and the axis of the drill is approximately 59° from the face of the wheel (Figure 58).

![Figure 58 - Aligning the drill.](image)

4. Check that the cutting lip is horizontal. Lightly touch the drill to the wheel to see that sparks are coming off horizontal and across the full face of the cutting edge to ensure that it is even and at right angles.
5. Bring the cutting lip into contact with the wheel and slowly lower the shank, thereby creating the proper degree of clearance (Figure 59).

![Figure 59 - Slowly lower the shank.](image)
6. Without moving the position of your arms or hands, rotate the drill 180° and repeat the process for the other cutting lip. Keep the drill cool by dipping it in coolant between grinds. If the drill turns blue, it loses its hardness.

7. Remove the drill and inspect the point with a drill point gauge. The cutting lips must be the same length and the point angle must be the same on both sides (Figure 60).

![Figure 60 - Using a drill point gauge.](image)

8. Repeat steps 2 to 6 until the cutting edges are sharp and the margins are free of wear.
**Modify the Drill Point**

Drills are manufactured with a standard point geometry. You must modify this geometry when:

- the chisel edge of the drill becomes too thick from repeated sharpening or
- the angle of the point must be changed to drill very soft or very tough materials.

**Change the Drill Point Angle**

The drill point angle for a general-purpose twist drill is 118°. Harder materials, such as tool steel, are best drilled with a 150° point angle. The shorter cutting lips produce less friction and heat. Softer materials, like plastic or non-ferrous metal, can be drilled with a point angle of between 60 and 90° (Figure 61). The angle of the drill point can be checked with a drill point gauge.

![Figure 61 - Drill point angles.](image)

**Thinning the Chisel Edge**

The chisel edge on a drill point does not cut efficiently and should be as thin as possible (Figure 62). The thickness of the chisel edge is determined by the width of the web. The web gets thicker as it gets closer to the shank of the drill. Eventually, as the drill is made shorter through repeated sharpening, the chisel edge becomes unacceptably wide and must be ground away.

![Figure 62 - Chisel edge that has been thinned.](image)

Give the grinding wheel a sharp corner by truing it and then thin the web by using the corner to cut away the portion of the web ahead of each cutting edge (Figure 63).

![Figure 63 - Thinning the chisel edge.](image)
Drill Problems
Drills are subjected to many different kinds of abuse in a millwright shop. You must be able to recognize the types of problems that can arise in order to correct them (Table 6).

<table>
<thead>
<tr>
<th>Problem</th>
<th>Cause</th>
<th>Remedy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Discolouration</td>
<td>Cutting speed too high.</td>
<td>Reduce cutting speed.</td>
</tr>
<tr>
<td></td>
<td>Not enough coolant.</td>
<td>Use more coolant.</td>
</tr>
<tr>
<td>Broken or split web</td>
<td>Too much feed.</td>
<td>Decrease feed.</td>
</tr>
<tr>
<td></td>
<td>Not enough clearance.</td>
<td>Grind more clearance.</td>
</tr>
<tr>
<td>Poor tool life</td>
<td>Cutting speed too high.</td>
<td>Reduce cutting speed.</td>
</tr>
<tr>
<td></td>
<td>Feed too high.</td>
<td>Increase feed.</td>
</tr>
<tr>
<td></td>
<td>Chipped cutting edges.</td>
<td>Do not overheat drill when grinding.</td>
</tr>
<tr>
<td></td>
<td>Point angle uneven.</td>
<td>Use drill gauge when grinding.</td>
</tr>
<tr>
<td></td>
<td>Too much clearance.</td>
<td>Grind less clearance.</td>
</tr>
<tr>
<td>Bell-mouth hole</td>
<td>Cutting lips uneven length.</td>
<td>Grind drill using a drill gauge.</td>
</tr>
<tr>
<td></td>
<td>Point angle uneven.</td>
<td>Grind drill using a drill gauge.</td>
</tr>
<tr>
<td>Out-of-round hole</td>
<td>Interrupted cut.</td>
<td>Plug existing hole.</td>
</tr>
<tr>
<td>Excessive drilling pressure</td>
<td>Not enough clearance.</td>
<td>Grind more clearance.</td>
</tr>
<tr>
<td></td>
<td>Chisel edge too wide.</td>
<td>Make the chisel edge thinner.</td>
</tr>
<tr>
<td>Drill chatter</td>
<td>Set-up not rigid enough.</td>
<td>Use larger machine and heavier workholding device.</td>
</tr>
<tr>
<td></td>
<td>Pilot hole too large.</td>
<td>Use a smaller pilot hole.</td>
</tr>
<tr>
<td>Squeals and jams</td>
<td>Worn margins.</td>
<td>Grind drill.</td>
</tr>
<tr>
<td></td>
<td>Excessive heat.</td>
<td>Use coolant.</td>
</tr>
<tr>
<td></td>
<td>Inadequate chip removal.</td>
<td>Withdraw drill to clear chips more often.</td>
</tr>
<tr>
<td>Burr</td>
<td>Rounded corners on drill.</td>
<td>Grind drill.</td>
</tr>
<tr>
<td>Blue chips</td>
<td>Excessive heat.</td>
<td>Use coolant.</td>
</tr>
<tr>
<td>Oversize hole</td>
<td>Cutting lips of unequal length.</td>
<td>Grind drill using a drill gauge.</td>
</tr>
</tbody>
</table>

Table 6 - Drilling problems, causes and remedies.
Care and Maintenance of Drilling Machines

Drilling machines need to be maintained regularly. You must consider the following in order to keep this equipment in good operating condition and to keep yourself safe.

**Sensitive Drill Presses**

If you maintain the following, the sensitive drill press should last for many years.

- The pulley shafts must be lubricated regularly.

<table>
<thead>
<tr>
<th>DANGER</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stop the machine before lubricating it.</td>
</tr>
</tbody>
</table>

- Restart the drill after oiling in order to give the bearings a chance to absorb the oil. Start several times, using all pulley or gear settings.
- All grease points on the drill press should be lubricated.
- Any machined surfaces that could rust must be kept clean and oiled.

<table>
<thead>
<tr>
<th>DANGER</th>
</tr>
</thead>
<tbody>
<tr>
<td>Do not stand with your feet under the worktable. Tables have been known to break off.</td>
</tr>
</tbody>
</table>

**Radial Arm Drill Presses**

The radial arm drill press needs to be monitored closely.

- A radial arm drill press has oil sight glasses. The oil levels must be kept up in the sight glasses to ensure proper operation of the drill.
- The screw shaft, which raises the column, must be greased as required.
- Clean and oil the table at the end of your shift and put all tooling and workpieces back on the shelf or on a bench.
- The T-slots must be free of chips and T-nuts for the next millwright.
- The floor area should also be swept and any oil or coolant on the floor cleaned up. You must prevent slippery floors.
- Excess oil should be wiped up and rags taken off the machine.
- Change coolant if it becomes rancid or if other cutting fluids contaminate it.

**Magnetic Base Drills**

The following are magnetic base drill maintenance items.

- When using a magnetic base drill, keep the magnetic base clean to maintain the strength of the magnet.
- The gibbs on the dovetail slide must be tight enough to keep the drill head steady when under pressure. Adjust them if necessary.

<table>
<thead>
<tr>
<th>DANGER</th>
</tr>
</thead>
<tbody>
<tr>
<td>When you are drilling horizontally ensure the power supply to the drill is secure. These drills can cause injury if dropped.</td>
</tr>
</tbody>
</table>
Drill Care and Maintenance

You need to look after drills with the following procedures.

- When installing drill bits into the spindle bore, ensure that the taper and the bore are both clean. Wipe them with a rag. Be careful not to scrape or mark the internal taper of the spindle bore. Any burrs in the taper throw your drill bit off-centre and prevent a tight fit for the drill.
- Repair any burrs or nicks using a honing stone.
- Never drill into the table. It is a machined surface, meant for precise work. Use spacers to raise the workpiece.
- Never use a metal hammer on the quill, spindle, table or column of a machine. Do not use the table as a surface for pounding the end of a taper sleeve or drift; the table could be marked up.
- Never leave a Jacobs chuck key or a taper drift in the spindle or chuck. These become projectiles if the machine is accidentally started. Do not over-tension the belts on a sensitive drill press as this puts too much stress on the motor bearings and the pulley shaft bearings.
Self-Test

1. Which variable is the most important when determining spindle speed?
   a) rigidity of the set-up
   b) diameter of the drill
   c) type of cutting fluid
   d) condition of the drill press

2. Any machined surfaces on a drill press that could rust must be _________________.

3. When installing a taper shank drill into the spindle bore, you must take several precautions. What are they and why?

4. Cutting speed refers to:
   a) peripheral speed.
   b) revolutions per minute (rpm).
   c) feed rate.
   d) inches per revolution.

5. How fast should a 1/2 inch drill turn when drilling mild steel if the cutting speed (CS) is 90 sfpm?
   a) 180 rpm
   b) 360 rpm
   c) 90 rpm
   d) 720 rpm

6. Small drills require a higher feed per revolution than larger ones.
   a) true
   b) false

7. Which abbreviation is used to express the feed rate on a drill press?
   a) sfm
   b) ipm
   c) ipr
   d) rpm

8. A drill press spindle has a #2 Morse taper. A twist drill with a #3 Morse taper shank must be mounted in the spindle. How should you secure the drill in the spindle?
   a) Use a drill sleeve.
   b) Turn the shank down in a lathe.
   c) Use a drill socket.
   d) Use a Jacobs keyless drill socket.
9. To remove a tapered shank twist drill from the drill press spindle, use a:
   a) drill sleeve.
   b) drill drift.
   c) drill punch (round).
   d) tapered punch.

10. When using a sensitive drill press, what should you do as the drill breaks through the workpiece?
   a) Apply cutting fluid.
   b) Increase the pressure.
   c) Ease up on the pressure.
   d) Switch the machine off.

11. The included point angle of a countersunk hole for most flathead screws is:
    a) 60°.
    b) 70°.
    c) 82°.
    d) 100°.

12. Which type of machine tap will push the chip out ahead of the tap?
    a) helical fluted tap
    b) spiral point (gun) tap
    c) bottoming tap
    d) fluteless tap

13. The lip relief (clearance) angle on a drill should be:
    a) 5° to 7°.
    b) 8° to 12°.
    c) 3° to 5°.
    d) 2° to 3°.

14. A general-purpose drill bit for drilling most metals has an included point angle of:
    a) 150°.
    b) 130°.
    c) 118°.
    d) 59°.

15. The hand drill is quicker and more versatile than any drill press for:
    a) reaming small holes for dowel pins.
    b) tapping vertical holes.
    c) drilling small non-vertical holes.
    d) counterboring.

16. Electric hand drills are limited by:
    a) the amount of feed pressure (and rpm) that can apply to large-diameter drills.
    b) the size of the internal Morse taper in the spindle.
    c) the removable handles are for right-handed workers only.
    d) the depth of hole that can be drilled.
Self-Test Answers

1. b) diameter of the drill
2. Any machined surfaces on a drill press that could rust must be kept clean and oiled.
3. Ensure that the drill taper and the bore are both clean. Wipe them with a rag. Be careful not to scrape or mark the internal taper of the spindle bore. Any burrs in the taper will throw your drill bit off centre and prevent a tight fit for the drill.
4. a) peripheral speed.
5. d) 720 rpm
6. b) false; Small drills require less feed per revolution than larger drills.
7. c) ipr
8. c) Use a drill socket.
9. b) drill drift.
10. c) Ease up on the pressure.
11. c) 82°.
12. b) spiral point (gun) tap
13. b) 8° to 12°.
14. c) 118°.
15. c) drilling small non-vertical holes.
16. a) the amount of feed pressure (and rpm) that can apply to large diameter drills.
This learning material is the result of a collaborative effort between Alberta Enterprise and Advanced Education and the apprenticeship training providers in Alberta.

Grande Prairie Regional College • Keyano College
Lakeland College • Lethbridge College • Medicine Hat College
Northern Alberta Institute of Technology • Northern Lakes College • Olds College
Portage College • Red Deer College • Southern Alberta Institute of Technology
The Individual Learning Modules Initiative is committed to continuous updating and is interested in your comments regarding technical accuracy and clarity of presentation. Please visit our web site to register your comment. To view the ongoing discussion, go to the Bulletin Board.

http://tradesecrets.alberta.ca/ilm/home.html

The Individual Learning Modules Initiative would like to thank the many companies for their contribution and gratefully acknowledges their valuable input. A complete listing of contributors can be found at our web site.