You Know the Drill

Bits used to restore or repair a Model A

By Lynn Sondenaa

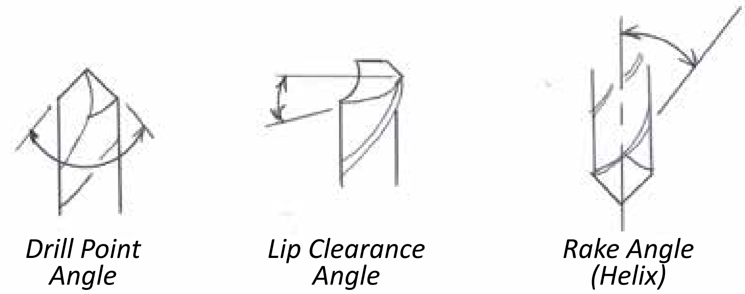

Figure 1

DRILLING IS THE MOST efficient method to make a hole in wood, plastic, or metal. First, it is important to understand the geometry of the twist drill bit. These are the *drill point angle, lip clearance angle,* and *rake angle* — also known as the *helix angle*. (See Figure1.)

Drill Point Angle

(A) 118-degree included angle, 59-degree point angle. These are general-purpose bits, mostly used for soft materials and nonferrous metals.

(B) 135-degree included angle, 67½-degree point angle. These are fast-cutting bits with low kickback, used for harder materials.

Figure 2

Note: When sharpening a drill bit, the drill point angle should be the same length on each side of the cutting lip. Different lengths will create oversized holes and uneven wear on the drill bit.

Lip Clearance Angle • This angle should be between 8 and 12 degrees. The purpose is to provide a cutting edge clearance so there is no drag in the cutting action. If excess feed pressure is needed to make the drill bit cut, that is because of an incorrect clearance angle. In Figure 3, A is the incorrect angle and B is the correct angle.

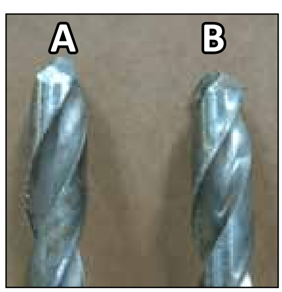

Figure 3

Rake Angle (Helix) • The rake angle is the angle of the flutes that run vertically, parallel to the drill bit's center line. This angle provides keenness to the cutting edge. This edge needs to be protected when not in use, which is why drill bits should be stored in an index and not just thrown into a box or a drawer.

Bits with a small rake angle, also known as a low helix drill or slow spiral, are more rigid. They are less likely to grab when cutting through metal. These styles are used when drilling sheet metal and soft materials. They are poor at removing chips from deep holes. (See Figure 4, A.)

Bits with a large rake angle are also known as high helix or fast spiral drill bits. These bits are designed to remove chips fast from deep holes and are also well suited for aluminum. (See Figure 4, B.)

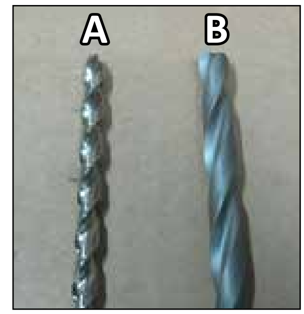

Figure 4

Drill Size Series

Fractional: by 1/16, 1/32, 1/64 of an inch
Number: 1 to 80 (carburetor jet sizes)
Letter: A to Z
Metric

Drill bits need to be able to match the decimal equivalents table so that holes can be tapped with threads. Number, letter, and metric drill bit diameters fall between the fractional inch measurements.

The selection of a drill bit depends on several factors: the type of material being drilled; whether using a hand-

Figure 5

Drill Bit Materials

(*From left*)

Designation	Material	Color
HS	High carbon steel	Black
HSS	High speed steel	Dull chrome
HSSCO	High speed steel with cobalt	Bright chrome Bright chrome
HSSTN	High speed steel titanium nitrate coating	Gold
Carbide	Solid carbide	Blued steel look

held power drill or a drill press; the size of the hole being drilled; and the depth of hole being drilled.

Styles of Drill Bits

Figure 6 shows a bullet head bit (A) and a center drill (B). The bullet head bit has a built-in center drill so it will not wobble or wander when starting. The center drill should be used whenever drilling metal. They range in size from ⅛" to ¾" body diameter. They are sized from 00 to 8.

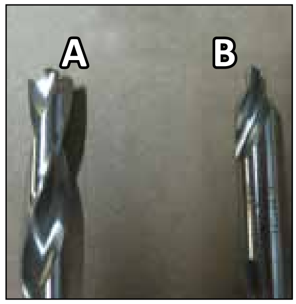

Figure 6

I recommend a Model A owner use a #2 and #3, as these would be the most useful, with #3 having a body diameter of ¼". This drill bit is used to start the hole for the regular twist bit. It provides accuracy as the bit won't wander when starting.

Center drills have a delicate end, so a low feed rate should be used. Drill only to a depth of ⅛" to 3/16". If you do not use a center drill, the hole needs to be marked with a center punch to keep the bit from wandering.

Figure 7 displays a solid carbide bit. While these bits are expensive, they are needed to drill spring steel, bumper brackets, leaf springs, broken twist bits, and broken taps. They work well to drill out broken head studs and manifold studs. While hard steel drill bits are also available to drill hardened steel, they are very expensive. They can easily drill through a mill bastard file.


Figure 7

Figure 8 shows a Rota-broach (A) and a hole saw (B). These two bits are used to cut larger holes in sheet metal. Examples would be cowl light holes or taillight wire holes in the rear fenders. The Rota-broach is very easy to use and is fast cutting, but it will cost about three times more than a hole saw. A hole saw will tend to grab and want to throw the power drill.

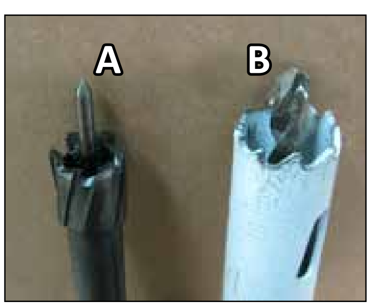

Figure 8

Figure 9 shows a step drill (A) and a countersink (B). The step drill is used to enlarge an existing hole. They usually have between 4 and 12 steps on the bit, each increasing in size. They have almost zero kickback and grabbing, so they are a great choice in drilling sheet metal.

Countersinks are used to place a bevel on a drilled hole. They actually do two operations. First, they will deburr the hole. Second, they will provide easy starting of taps if cutting threads. They are also used for the correct seating angles for countersunk screw heads.

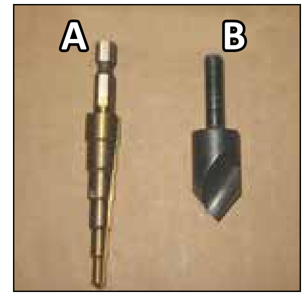

Figure 9

Figure 10 displays two important tools that are needed with twist bits. The drill size gauge (left) is used for a quick reference to the size of the bit. They are manufactured in fractional, number, letter, and metric sizes. To be absolutely sure of the drill bit size, a micrometer should be used. The drill point angle gauge (right) is used during the sharpening process to measure for correct point angle and equal lengths of the lips. Each side of the drill bit needs to be equal to prevent drilling oversized holes and causing uneven wear to the bit.

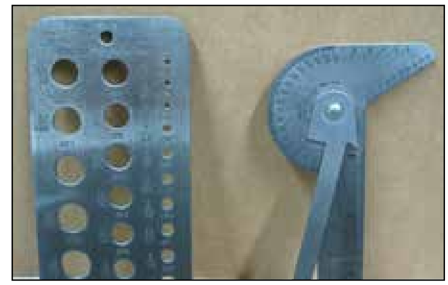
Figure 10

Cutting oils are used to keep the bit cool and help in the removal of chips from the hole being drilled. *Note:* Do not use WD-40, CRC-36, Liquid Wrench, or silicons. These products are designed to lubricate and will greatly hinder cutting. Do not use motor oil, as it will produce smoke. Use these for cutting oils:

Aluminum: Kerosene
Cast iron: Dry. No cutting oil necessary
Steel: Dark sulfur cutting oil

Figure 11 shows a twist bit with torn-up helix cutting lips due to excessive heat and lack of oil. This bit is now useless.


Figure 11

Why Bits Spin or Grab • Though three-jaw universal chucks move all at once, each needs to be tightened to prevent spinning when drilling metal. A dull drill bit or incorrect lip angle will also cause the bit to spin in the chuck, as will a low helix drill or the lack of a pilot hole.

The number-one cause of a drill bit grabbing in the work is too much feed pressure being applied, especially when cutting through the hole. When drilling, remember to interrupt the feed occasionally to break up the chips.

Cutting Speeds • Machinists will use a math formula to determine the exact drilling speed. But a Model A mechanic or restorer can use this rule of thumb:

1/16" to ¼" diameter — 1000 rpm
¼" to ½" diameter — 500 rpm
½" to ¾" diameter — 300 rpm

If the cutting chips are dark blue, the speed is too high. The harder the material, the slower the speed should be.

Drilling Safety

Wear eye protection but never gloves. Clamps should be used to hold work pieces. Do not attempt to grab moving parts, and don't use your fingers to brush chips.

When drilling, heat is generated. So be aware when moving the drilled part or removing the bit from the chuck.

When drilling a hole larger than ⅜", first drill a pilot hole. When using a drill press, place the long side of the work piece to the left side, resting against the column. This will help prevent kickback.

If you apply this information, drilling will become easier, safer, and more efficient. ⊖

What's on Tap?

How to select the correct tap for Model A restoration and repair

By Lynn Sondenaa


Figure 1

Internal threads are made by taps. These taps vary in style and uses. This article will help you identify and select the correct tap for a specific operation on a Model A. Taps cut the internal threads in holes and nuts, while dies cut the external threads on bolts, studs, and rods. While taps are also made to be used in drill presses and milling machines, I will cover only those that are used by hand, as that is what most Model A mechanics own.

Taps are made to cut either right-hand or left-hand threads. The tie rod A-3281 on the Model A front axle has right-hand threads on one end and left-hand threads on the other, as do the tie rod end caps A-3285 and A-3286. These right- and left-hand threads allow for the adjustment of the toe in and toe out on the Model A front end. Figure 1 has the four common styles of taps from left to right: taper, plug, bottoming, and pipe.

There are also Acme screw thread taps. I will not cover these taps as they are primarily used in the machine tool industry. They are a type of thread used to transmit power into motion on a mechanical part.

Taps are made in two-, three-, and four-flute styles (Figure 2). Sometimes small screws are referred to as machine screws. Examples are 4-40, 5-40, and 6-32. Machinists refer to them as National Fine (NF) or National Coarse (NC).


Figure 2

Features of Taps

The flutes of a tap provide space for the chips to collect and the passage for cutting fluids. *Taper taps* are designed with a chamfer at the bottom to help keep the tap at 90 degrees to the hole to make it easier to start cutting the threads. The taper tap will widen to full threads about halfway up the thread length.

Figure 3

Plug taps are general purpose taps. Most tap-and-die sets are plug taps. These taps have a slight chamfer, and full thread starts about one-quarter up the thread length.

Bottoming taps have no chamfer and can be used only after threads have been started. They start with full threads at the bottom of the threads. Bottoming taps are used to produce threads that extend almost to the bottom of a blind hole.

A *blind hole* is one that is not drilled clear through a part. Examples are head studs and manifold studs on the engine block. The area by the shank of the tap is back-tapered to reduce tool contact. Figure 3 shows the chamfer and the back taper on a tap.

Pitch is the distance from the crest of one thread to the crest of the next. A thread is a helical structure used to convert between rotational and linear movement. Classes of fits are 1, 2, 3 (loose to tight) and A and B (external and internal).

For example, the head on the Model A engine is held down by a threaded stud. When the head is removed, it is nice to have the stud remain in the block. This end requires a tighter thread fit than the end of the stud accepting the nut. If the nut end fit is too tight, it will remove the stud from the block.

Taps are made from high-carbon steel or high-speed steel, which is the most common. Special-use taps are also made from cobalt or coated with titanium nitrate (gold color), and solid carbide for tapping very hard metals such as spring steel.

The markings on a tap (Figure 4) provide the following information. Generally the first number is the nominal size, followed by the number of threads per inch and either NC or NF to indicate coarse or fine threads. (Sometimes UNC and UNF are used.) These are known as the unified national thread series, which is the basic American standard for screw threads.

Below that will be the tolerance range and type of material from which the tap is made. Sometimes the tap drill size and country of manufacture will be imprinted. Depending on the size of the tap and the manufacturer, the tap might not show all of this information.

Figure 5 shows, from left to right, the common tapping tools. The screw pitch gage is used to measure threads per inch. The tap wrench is used to hold taps from ¼ to ½ inch. Larger wrenches are used for larger taps. The T-handle is used to hold taps smaller than ¼ inch. The tap wrench and T-handle provide the torque in hand tapping.

Pre-Tapping Information

When tapping, the material should be lubricated to prevent tap breakage and provide smooth, continuous threads. Aluminum and brass call for kerosene or light cutting fluid. Cast iron generally does not require cutting oil. Steel calls for dark, sulfur cutting oil. Do not use WD-40, Liquid Wrench, or motor oil, as these do not aid in the removal of the chips.

Figure 6

Tap Size	Tap Drill
4-40	#43
5-40	#38
5-44	#37
6-32	#36
6-40	#33
8-32	#29
8-36	#29
10-24	#25
10-32	#21
12-24	#16
12-28	#14
¼-20	#7
¼-28	#3
5⁄16-18	F
5⁄16-24	I
⅜-16	5⁄16
⅜-24	Q
7⁄16-14	U
7⁄16-20	25⁄64
½-13	27⁄64
½-20	29⁄64
9⁄16-12	31⁄64
9⁄16-18	33⁄64
⅝-11	17⁄32
⅝-18	37⁄64

Pipe Taps

Tap Size	Tap Drill
⅛-27	R
¼-18	7⁄16
⅜-18	37⁄64
½-14	23⁄32
¾-14	59⁄64

Figure 4

Figure 5


Figure 7

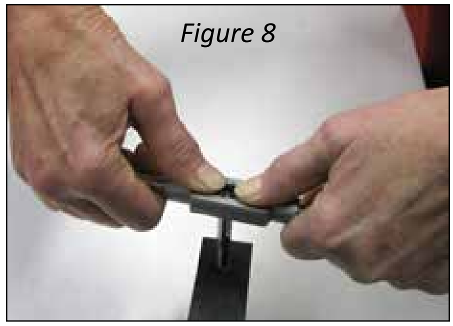
Figure 8

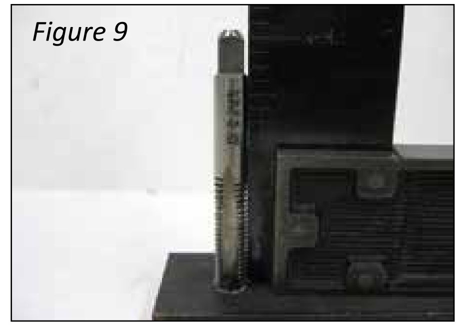
Figure 9

The correct size drill bit must be used so that the cut threads will hold torque values. A tap drill should be sized to produce approximately 75 percent full threads. These drill bits may be fractional, number, or letter so the exact diameter of the hole can be drilled to maintain the 75 percent thread depth. Figure 6 shows common tap drill sizes for 75 percent threads.

Using Hand Taps

After the hole is drilled, it should be countersunk lightly (Figure7), as this helps in starting the tapping process. Apply the proper cutting fluid to the tap. Use either a taper tap or plug tap to start the hole.

Figure 8 shows the correct position of the hands when tapping. Apply even pressure downward while turning the tap. Turn the tap into the hole about one-half turn, then back it out a quarter turn. Repeat over and over again. The idea is to break the chips into small pieces so they go into the flutes.

By using this process, the tap will not break, and the cut threads won't be damaged and become rough or torn. After the tap has cut several threads, it is a good idea to use a square and check that the tap is 90 degrees to the

Specialty Taps and Others

Spark plug holes	⅞-18 (this is not a pipe tap)
Distributor can screw	5/16-32
Tie rod ends	⅝-18 right hand, ⅝-18 left hand
Zenith carburetor fuel inlet threads	½-20
Sediment bowl outlet internal threads	½-20
Zenith carburetor jet threads	.5mm x .75 pitch
Zenith carburetor, throttle, and choke plate screws	5-40
1928 oil pan drain plug hole threads	⅞-16
1929–1934 oil pan drain plug hole threads	¾-24
Front spindle nut	¾-16
Steering shaft nut	⅝-18

Examples of Pipe Taps

Oil pump and intake manifold vacuum hole	⅛-27
Threaded grease fitting holes	⅛-27
Steering box oil filler plug hole	¼-18
Transmission drain and filler plug holes	¾-14
Differential drain and filler plug holes	¾-14

Removing a Broken Tap

If you break a tap, try using pliers to remove the broken piece from the hole. If the tap is broken flush to the surface, a tap extractor can be used (Figure 10). A tap extractor has small fingers that fit into the flutes of the tap to grip the tap inside the hole and remove it.

If a tap extractor is not available, there are a couple of practices I use. (Wear safety glasses as taps are hardened steel, and they can shatter.) I first try using a scratch awl to turn the tap by lightly hitting the awl with a small hammer. If it does not turn, use the awl to break the tap into small pieces. If that does not work, I use a fluted concrete nail (Figure 11). These nails are the same hardness as a tap, and they have a diamond-shaped point. This will break the tap into small pieces.

A solid carbide drill bit at a slow speed can drill out the broken tap. But these bits are expensive and difficult to use for the inexperienced. A machine called an EDM (electro-discharge machining) does a very fast and accurate job of removing broken drill bits and taps. Some modern automotive machine shops and machine tool shops have EDMs.

Causes of Broken Taps

- Poor quality tap
- Wrong style of tap for the material
- Incorrect tap drill size
- Lack of cutting fluid
- Chips packed into the flutes
- Not backing the tap out for a half turn to break the chip
- Excessive force used to turn the tap
- Dull tap
- Tap hitting the bottom of the hole

Figure 10

Figure11

hole, as shown in Figure 9. If the hole is deep, the tap should be withdrawn and the flutes cleaned.

Blind holes are ones that do not go completely through the piece. They have a solid bottom. A bottoming tap is used after the threads have been started. When tapping if you feel the tap bind or get really hard to turn, stop! The tap needs to be backed up or taken out, and the chips cleaned from both the tap and hole. Excessive pressure will lead to a broken tap.

You now have a wealth of information about taps and tapping. Use the tap drill chart to use the correct drill bit as you work on your Model A. Taps can also be used to chase existing threads and to help remove dirt and rust from internal holes. ⊖

Casting the Die

What you need to cut threads on a rod or bolt

By Lynn Sondenaa

Dies are used to cut external threads on the surface of a rod or bolt. Many parts on the Model A are put together with these threaded fasteners.

Dies of good quality are made of high speed steel. Dies are identified by the markings on their face (Figure 1a). These markings tell the size of the threads, number of threads per inch, and if they are National coarse (NC), or National fine (NF).

On American-made dies, the back side (Figure 1b) will usually say either, "start from this side" or "start from other side." This is so the operator will get the die's chamfer in the correct position to cut threads.

As with taps, dies are made in all standard thread sizes: in English sizes, metric sizes, and pipe sizes (Figure 2). Notice the die on the right has the letters NPT, which stand for National Pipe Tap.

Dies are also made with right-hand and left-hand threads. Left-hand threads can be found on one end of the Model A tie rod and lug studs of AA trucks, front left AA-1108-B, rear left AA-1119, and the new hubs

Figure 1a

Figure 1b

Figure 2

Figure 3


Figure 4

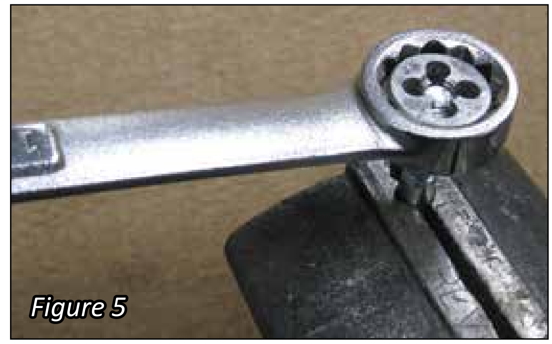
Figure 5


Figure 6


Figure 7


Figure 8

AA-1114-F and AA-1119-E. Figure 3 shows a left-hand die with the abbreviation LH for left-hand.

There are two common die shapes, as shown in Figure 4, round and hexagon. Hexagon dies are more expensive but have a positive grip in the diestock. Round dies have a slight tendency to slip. Hexagon dies also have the advantage that a wrench or socket can be used to turn them (Figure 5). Round, split, adjustable dies are also made, which allow minor adjustments to make a loose or tight fit on a threaded part (Figure 6).

Figure 7 shows the two common types of diestocks, which are the tools used to hold dies. There are also rethreading dies, which are used to recut threads that are slightly damaged or rusted. These do not have the chamfer on one side, so they can't be used to cut new threads. Rethreading dies are commonly used to chase threads on lug studs and brake rods.


Figure 9


Figure 10

When Cutting New Threads

1. The outside diameter of the material must be the same size of the die or .002" to .005" undersized.
2. Use the chamfer side of the die facing down when starting. (Remember to wear eye protection.)
3. Use cutting oils that are sulfurized. They do not smoke and reduce the friction.
4. Chamfer the rod before using the die. The chamfer should be 45 degrees and half the pitch of the threads to be cut (Figure 8).
5. Start the die with one hand, pressing down firmly while turning (Figure 9). This helps keep the die at 90 degrees.
6. After threads are started, use both hands to turn the diestock (Figure 10), using a slight downward pressure. If the die is crooked, more pressure can be placed on one of the diestock handles to bring the die back to 90 degrees.
7. After one full turn, back off the die a half turn. This will break the chip and keep the threads clean. Follow this process to the bottom of the threads.

Finally, do not use a regular die to cut new threads on grade 8 bolts, as these are hardened steel and will damage the die. A rethreading die can be used to chase the threads on grade 8 bolts, but not to cut new threads.

Dies are useful on the Model A to cut new threads or repair slightly damaged threads. ϴ

Lynn Sondenaa of Sandy, Oregon, purchased his first Model A while in the 7th grade. He and his wife, Patty, are members of the Beaver Model A Club of Portland, Oregon and own a 1929 Roadster Pickup and a November 1930 Victoria.