1

MACHINING

DRILLING

Drilling holes in materials seems easy, but obtaining a clean, smooth hole without breaking the drill bit takes only a bit of knowledge about drilling. Here are some quick rules to make every drilling job better.

The tip of a drill is *not* a small point but a horizontal lip. A center punch must be used to make sure the drill starts at the desired position on the workpiece. If a large drill is used, it is very important to drill a pilot hole first in order to achieve proper centering.

The *smaller* the drill, the faster the speed at which it needs to turn, and the slower the feed.

The *larger* the drill, the slower the speed at which it needs to turn, and the faster the feed.

Quick Guide to Drill Press Speed (RPM)

Drill Size	Steel	Brass	Aluminum*	Plastic (Acrylic)	Hard Wood	Soft Wood	Cast Iron
1/8″–3/16″	3000	3000	3000	2500	3000	3000	1500
1/4″–3/8″	1000	1200	2500	2000	1500	3000	700
7/16″–5/8″	600	750	1500	1500	750	1500	500
11/16″–1″	350	400	1000	1000	500	750	400

* When drilling aluminum, use a "pecking" technique, bringing the drill down and back up frequently, to remove chips and keep flutes from clogging. If this is not done, drill breakage can result.

DRILL PRESS TROUBLESHOOTING

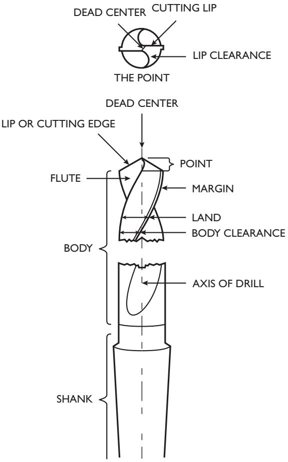
Symptoms	Cause	Fix
Drill breaks or snaps.	Speed is too low in proportion to the feed. Drill is dull. Lip clearance is too small.	Increase speed or decrease feed. Sharpen drill. Regrind properly.
Outer corners of cutting edges are breaking down.	Material being drilled has hard spots. Speed is excessive. Improper cutting oil is being used. Lubricant is not reaching point of drill.	Reduce speed. Use different lubricant and apply more often. Remove drill and spray hole.
Drill breaks when drilling wood or brass.	Chips are clogging flutes.	Increase speed. Use drills designed for these materials.
Hole is too large.	Angle or length of cutting edges (or both) is unequal, Spindle on drill press is loose.	Regrind drill properly. Tighten or adjust spindle.
Drill fails to cut stainless steel, although initially it worked okay.	Stainless steel has work-hardened because of slow feed and drill no longer cuts because cutting edges are dulled.	Resharpen drill and use heavy feed to cut through work-hardened surface, then use increased feed to continue.
Drill overheats on deep hole.	Heat cannot be removed because drill is going too fast.	Reduce speed and remove drill from hole frequently.

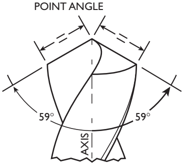
DRILL SHARPENING

There are a great many gadgets on the market to sharpen drills, but freehand sharpening is a skill that is easy to master and can be used to make drilling certain materials a lot easier. Sharpening a drill differently for steel or brass or aluminum can make a hard job easy and give a clean, burr-free hole.

The most common parts of a twist drill are shown here.

A drill is really a rotating chisel called the *lip* (or cutting edge), which skives off material as it turns. The *flutes* allow the material a place to go so that more material can be cut.





Drill Tip Geometry	
Material	**Point Angle**
Aluminum	90–135°
Brass	90–118°
Cast Iron	90–118°
Mild Steel	118–135°
Stainless Steel	118–135°
Plastics	60–90°

The drill-point angle can be changed to better accommodate drilling holes in various materials. The standard drill bit has a 118-degree included angle and will work in most materials.

There are generally three essential requirements in twist drill sharpening:

1. Equal drill-point angles, which are usually 59 degrees each, for a total of 118 degrees
2. Cutting lips of equal length
3. Correct clearance behind the cutting lips, which is approximately 8 to 12 degrees

Grind all twist drills without overheating them. Keep their points cool enough so that you can touch them with your bare fingers. Making very light passes over the wheel can do this. If a drill is under 1/8″ in diameter, the rule of thumb is to just get a new one rather than sharpening.

When sharpening, always make sure that there is relief behind the cutting edge. If the trailing edge of the bit is higher than the cutting edge, it will not cut.

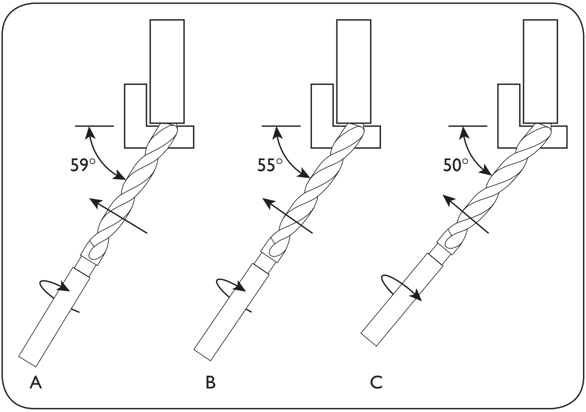
If the bit is used to drill hard material, the end should be flatter so it won't dig in and dull fast. *Always* center-punch the work to prevent the drill from "walking." The 59-degree angle used on many drills is a standard angle for mild steel; for softer materials, make the point more like a pencil. When learning, start with a large bit, and remember that if the steel gets hot enough to turn black at the cutting edge, carbon has been removed from the metal and the edge will dull *very* soon. Grind slowly, and dip in water often. Get a new bit and compare the shape to the ground one—there should be very little difference. When thinning the web, remember that the thinner the web, the more easily it will penetrate the work, but also the more easily it can grab and split the drill down the center, ruining the drill immediately. If the sides (lands) are worn on the drill bit, it will grab and break.

After grinding, hold the bit up with a bright background behind it and with the cutting edges going left and right. Make sure that both the tips are the same height and that the point is in the center before attempting to thin the web (if needed). If there is a pilot hole with a greater diameter than the thickness of the web, no thinning is required.

When grinding by hand, always hold the cutting lip level against the wheel at the center height of the wheel and rotate the drill upward to make the relief. Grind slowly and don't try it with a wheel that is out of round—you will get hurt badly. Dress the wheel to be true before starting any grinding process.

Begin by holding the bit on your forefinger, with its cutting lip horizontal and the axis of the drill at an angle of about 59 degrees. The actual grinding process involves three distinct motions of the shank while the bit is held lightly against the wheel. The three motions occur simultaneously:

- To the left
- In a clockwise rotation
- Downward



DRILL SIZES AND DECIMAL EQUIVALENTS

Drill	mm	Decimal	Drill	mm	Decimal
–	**0.10**	.0039	**62**	0.97	.0370
–	**0.20**	.0079	**61**	0.99	.0380
–	**0.25**	.0098	–	**1.00**	.0390
–	**0.30**	.0118	**60**	1.02	.0400
80	0.34	.0135	**59**	1.04	.0410
79	0.37	.0145	**58**	1.07	.0420
1/64	0.40	.0156	**57**	1.09	.0430
78	0.41	.0160	**56**	1.18	.0465
77	0.46	.0180	**3/64**	1.19	.0469
–	**0.50**	.0197	**55**	1.32	.0520
76	0.51	.0200	**54**	1.40	.0550
75	0.53	.0210	**53**	1.51	.0595
74	0.57	.0225	**1/16**	1.59	.0625
–	**0.60**	.0236	**52**	1.61	.0635
73	0.61	.0240	**51**	1.70	.0670
72	0.64	.0250	**50**	1.78	.0700
71	0.66	.0260	**49**	1.85	.0730
–	**0.70**	.0276	**48**	1.93	.0760
70	0.71	.0280	**5/64**	1.98	.0781
69	0.74	.0292	**47**	1.99	.0785
–	0.75	.0295	–	**2.0**	.0787
68	0.79	.0310	**46**	2.06	.0810
1/32	0.79	.0313	**45**	2.08	.0820
–	**0.80**	.0315	**44**	2.18	.0860
67	0.81	.0320	**43**	2.26	.0890
66	0.84	.0330	42	2.37	.0935
65	0.89	.0350	3/32	2.38	.0938
–	**0.90**	.0350	41	2.44	.0960
64	0.91	.0354	40	2,50	.0980

Drill	mm	Decimal	Drill	mm	Decimal
63	0.94	.0360	39	2.53	.0995
38	2.50	.1015	**15**	4.57	,1800
37	2.64	.1040	**14**	4.62	.1820
36	2.71	.1065	**13**	4.70	.1850
7/64	2.78	.1094	**3/16**	4.76	.1875
35	2.79	.1100	**12**	4.80	.1890
34	2.82	.1100	**11**	4.85	.1910
33	2.87	.1130	**10**	4.91	.1935
32	2.95	.1160	**9**	4.98	.1960
–	**3.00**	.1181	–	**5.00**	.1968
31	3.05	.1200	**8**	5.05	.1990
1/8	3.18	.1250	**7**	5.11	.2010
30	3.26	.1285	**13/64**	5.16	.2031
29	3.45	.1360	**6**	5.18	.2040
28	3.57	.1405	**5**	5.22	.2055
9/64	3.57	.1406	**4**	5.31	.2090
27	3.66	.1440	**3**	5.41	.2130
26	3.73	.1470	**7/32**	5.56	.2188
25	3.80	.1495	**2**	5.61	.2210
24	3.86	.1520	**1**	5.79	.2280
23	3.91	.1540	**A**	5.94	.2340
5/32	3.97	.1562	**15/64**	5.95	.2344
22	3.98	.1570	–	**6.00**	.2362
–	**4.00**	.1575	**B**	6.05	.2380
21	4.04	.1590	**C**	6.15	.2420
20	4.09	.1610	**D**	6.25	.2460
19	4.22	.1660	**1/4**	6.35	.2500
18	4.31	.1695	**E**	6.35	.2500
11/64	4.37	.1719	**F**	6.53	.2570
17	4.39	.1730	**G**	6.63	.2610
16	4.50	.1770	**17/64**	6.75	.2656

Drill	mm	Decimal	Drill	mm	Decimal
H	6.76	.2660	**Z**	10.49	.4130
I	6.91	.2720	**27/64**	10.72	.4219
–	**7.00**	.2756	–	**11.00**	.4331
J	7.04	.2770	**7/16**	11.11	.4375
K	7.14	.2810	**29/64**	11.51	.4531
9/32	7.14	.2812	**15/32**	11.91	.4688
L	7.37	.2900	–	**12.00**	.4724
M	7.49	.2950	**31/64**	12.30	.4844
19/64	7.54	.2969	**1/2**	12.70	.5000
N	7.67	.3020	–	**13.00**	.5118
5/16	7.94	.3125	**33/64**	13.10	.5156
–	**8.00**	.3150	**17/32**	13.49	.5312
O	8.03	.3160	**35/64**	13.89	.5469
P	8.20	.3230	–	**14.00**	.5512
21/64	8.33	.3281	**9/16**	14.29	.5625
Q	8.43	.3320	**37/64**	14.68	.5781
R	8.61	.3390	–	**15.00**	.5906
11/32	8.73	.3438	**19/32**	15.08	.5938
S	8.84	.3480	**39/64**	15.48	.6094
–	**9.00**	.3543	**5/8**	15.88	.6250
T	9.09	.3580	–	**16.00**	.6299
23/64	9.13	.3594	**41/64**	16.27	.6406
U	9.35	.3680	**21/32**	16.67	.6562
3/8	9.53	.3750	–	**17.00**	.6693
V	9.56	.3770	**43/64**	17.07	.6719
W	9.80	.3860	**11/16**	17.46	.6875
25/64	9.92	.3906	**45/64**	17.86	.7031
–	**10.00**	.3937	–	**18.00**	.7087
X	10.08	.3970	**23/32**	18.26	.7188
Y	10.26	.4040	**47/64**	18.65	.7344
–	**19.00**	.7480	**7/8**	22.23	.8750
3/4	19.05	.7500	**57/64**	22.62	.8906

Drill	mm	Decimal	Drill	mm	Decimal
49/64	19.45	.7656	–	**23.00**	.9055
25/32	19.84	.7812	**29/32**	23.02	.9062
–	**20.00**	.7874	**59/64**	23.42	.9219
51/64	20.24	.7969	**15/16**	23.81	.9375
13/16	20.64	.8125	–	**24.00**	.9449
–	**21.00**	.8268	**61/64**	24.21	.9531
53/64	21.03	.8281	**31/32**	24.61	.9688
27/32	21.43	.8438	–	**25.00**	.9842
55/64	21.84	.8594	**63/64**	25.00	.9844
–	**22.00**	.8661	**1″**	25.40	1.0000

Type of Drill	Characteristics
Low-carbon steel	Cheap. Used for wood. Requires frequent sharpening.
High-carbon steel	Can be used on metal or wood. Loses temper if overheated.
High-speed steel	A form of tool steel. Resistant to heat. Good for wood and various metals. General purpose.
Cobalt steel	Holds hardness at higher temperatures. Works with stainless steel.
Black oxide and titanium nitride	Coatings used on drills to improve hardness. Unfortunately, removed by sharpening.

SCREW THREADS

Recorded history shows the screw thread was developed by Archimedes in the third century BC. He used a pipe wrapped around a shaft in a helical pattern to make a crude bilge pump for ships. Archimedes took the basic inclined plane and wrapped it into a spiral shape. Early screws were made by wrapping wire around plain bar. Nuts were made of softer material (copper, for example) by forging them around the wire-wrapped rod. Later, screws were cut from solid bar using single-point cutting tools or chasers. Because of its many advantages, thread rolling is the preferred method of manufacture today. Early screw manufacturing suffered from the absence of accurate and powerful machinery capable of holding minimally accurate tolerances. This was compounded by the lack of accurate inspection methods. For many years screws and nuts were manufactured and used in matched sets and, as a result, were not interchangeable.

Most mechanical screw thread has three different pitches (threads per inch) available for a given diameter. The terms used to refer to them are *coarse, fine,* and *extra-fine.*

Coarse thread (UNC) is used for bolts, screws, nuts, and other applications where rapid assembly or disassembly is needed or corrosion or other slight damage might occur.

Fine thread (UNF) is used for bolts, screws, nuts and other applications where a finer thread is needed. It is used where the length of engagement is short, where a smaller lead angle is desired, or when the wall thickness demands a fine pitch.

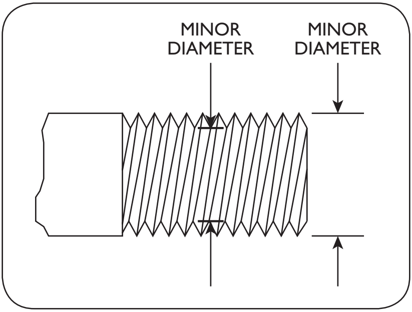
Extra-fine thread is used where even finer pitches are desirable for a short length of engagement and for thin-walled tubes, nuts, ferrules, or couplings.

Machine screws with a body diameter less than 1/4″ use a number designation ranging from #0 to #12. The major diameter of #0 is 0.0600″ and each higher number adds 0.0130″ to the previous diameter. The major diameter of #3 screw thread is 0.0600″ + 0.0130″ + 0.0130″ + 0.0130″, or 0.0990″.

Fractional screw sizes are generally limited to ¼ through 5/8 (by 1/16ths), ¾ through 1-¼ (by 1/8ths), and 1-½ through 2 (by ¼ths). They are commonly available in UNC and UNF pitches. UNEF pitches are available, but they are not commonly stocked.

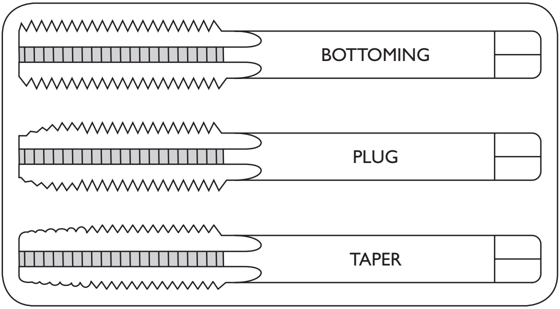
> *You don't have a thread chart available and you need to know the major diameter of a #8 machine screw? Remember that the #0 screw is 0.0600″, and add 0.0130″ for each number designation. Starting with the #0 screw and a major diameter of 0.0600″, add 0.0130″ 8 = 0.1040″. 0.0600″ + 0.1040″ = 0.1640″. How about a #6 screw? Answer: 0.1380″*

Thread standards for pipe include National Pipe Tapered (NPT) and National Pipe Straight (NPS). The threads for tapered pipe are cut on a taper of 1/16″ per inch. This taper allows them to form a seal when torqued as the flanks of the threads compress against each other, as opposed to NPS fittings in which the threads merely hold the pieces together and do not provide the seal. However, a clearance remains between the crests and the roots of the threads, resulting in leakage around this spiral. This means that NPT fittings must be made leakfree with the aid of thread seal tape or a thread sealant.

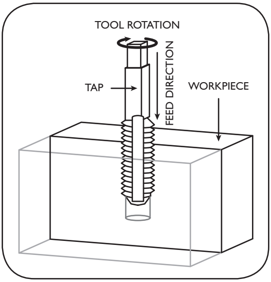


DRILLING AND TAPPING A HOLE

Drilling and tapping a hole with threads can be a tricky operation. Good taps are made of hardened high-speed steel (HSS) to allow sharp cutting of the threads, but they also are susceptible to breaking or snapping if care is not used. The hole drilled for a tapped thread is usually large enough to allow about 75 percent threading. This amount is adequate to keep the threads from stripping out while tightening a bolt or fastener to a recommended torque. One of the most common reasons taps break is that the drilled hole is too small and the threading requirement is above 75 percent. This causes excessive torque on the tap handle. The tap is very likely to snap. Since a tap is harder than most drills, a broken tap is difficult to drill out using conventional drills. Another reason for tap breakage is that, of all the machining operations, tapping requires the most lubricant. Too little lubricant causes the tap to bind and snap. You cannot use enough lubricant when you are tapping.



There are three types of taps in common usage: the plug tap, the bottoming tap, and the taper tap. The plug tap is used for production tapping with a tapping machine. Most maintenance work involves the use of the bottoming and taper types of taps. The taper tap is easy to start and keep straight to make sure the threads are at right angles to the axis of the drilled hole. The bottoming tap is used when a hole is drilled into a piece of material but not all the way through. Usually a hole must be drilled a



little farther than the length of threads needed with a bottoming tap. If a plug or taper tap is used in a blind hole, there is a good chance it will lock up or break.

Bottoming taps should not be used to start a new thread. Use a taper tap to begin the threading, and then remove and use the bottoming tap to finish a blind hole.

Taps are usually broken by bending, not twisting. A wrench that pushes on only one side puts an asymmetrical stress on the tap and is very likely to break it. Instead, a "T" handle should always be used, which makes it easier to turn the tap without bending it.

Sometimes when tapping a hole, commercially produced lubricants aren't available. In that case, the following lubricants may be substituted:

Material	Substitute Lubricant
Carbon steel	Sulfurized cutting oil used for pipe threading and found in hardware stores
Aluminum	Kerosene or fuel oil
Cast iron	Can be dry-tapped or add some WD-40
Plastics	Dry-tap or use liquid soap
Copper	Crisco shortening

The steps involved in tapping a hole are as follows:

1. Determine the size and depth of the thread desired; usually, this is the same size as the bolt or fastener being screwed into the hole.
2. Determine the tap drill size from a chart or calculation.
3. Drill the hole to the correct depth. Remember that taps cannot thread all the way to the bottom of a blind hole, so a blind hole must be deeper than the thread depth.

4. Blow out the hole to remove any drill shavings.
5. Make sure the workpiece is locked down or braced to prevent it from moving.
6. Apply enough lubricant to the hole and the tap.
7. Insert the tap into a "T" handle or similar holder.
8. Hold the tap in a straight line with the hole and turn it clockwise. The sharp cutters on the tap will bite into the material and begin forming the threads.
9. Every few turns, back the tap up (turn it counterclockwise) to clear away chips of loose material. With some materials, the tap may need to be backed up even after making a half turn. If the pressure increases on the handle, back up to clear the tool and start again. Add more lubricant.
10. When the tap reaches the proper depth or creates sufficient threads, back it all the way out, blow the chips out, and make sure to brush any chips out of the tap itself before putting it away.

Small taps are extremely easy to break. It takes only about 5 lb of force (off center) to snap a #6-32 tap. Most better sets of taps are necked down at the top to confine breakage to this area, allowing the tap to be removed with a vise-grip pliers, since part of it protrudes above the workpiece. If the workpiece is extremely valuable, a broken tap can be removed with electrical discharge machining (EDM) techniques, usually available at any precision machine shop. The price for removing the tap is much less than it would cost to scrap an expensive part.

CALCULATING A TAP DRILL SIZE WITHOUT A TABLE

If you are without a table for selecting the proper tap drill size, it can be calculated quickly.

Subtract 1/pitch (threads per inch) from the major diameter of the thread. A few examples show how to do this:

Drill and tap for 1/4-20 NC threads

1/4″ = major diameter of the thread

20 = pitch, or threads per inch

1/4″ (0.250″) – 1/20″ (0.050″) = 0.200″

Either a #7 (0.201″) or a 13/64″ (0.203″) drill will work.

Drill and tap for 1/4-28 NF threads

1/4″ = major diameter of the thread

28 = pitch, or threads per inch

1/4″ (0.250″) – 1/24″ (0.035″) = 0.215″

Either a #3 (0.213″) drill or a 7/32″ (0.218″) drill will work.

You must drill and tap a 1/2-13 NC threaded hole. What tap drill size do you need? 1/2″ = 0.500″. 1/13 = 0.076″. 0.500″ – 0.076″ = 0.424″. 27/64″ (0.421″) is the tap drill size needed.

Calculate the tap drill size for a 3/8-16 NC drilled and tapped hole.

3/8" = 0.375" and 1/16" = 0.063"
0.375" – 0.063" = 0.312", or a 5/16" tap drill should be used

What about a 5/8-11 NC drilled and tapped hole?

5/8" = 0.625" and 1/11" = 0.090"
0.625" – 0.090" = 0.535", or a 17/32" tap drill should be used.

SCREW THREAD—DRILL AND TAP CHART (UNC AND UNF)

Machine Screw Size		Threads per Inch	Minor Diameter	Tap Drills				Clearance Hole Drills			
				75% Thread		50% Thread		Close Fit		Free Fit	
# or Diam.	Major Diam.			Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.	Drill Size	Decimal Equiv.
0	.0600	80	.0447	3/64	.0469	55	.0520	52	.0635	50	.0700
1	.0730	64	.0538	53	.0595	1/16	.0625	48	.0760	46	.0810
		72	.0560	53	.0595	52	.0635				
2	.0860	56	.0641	50	.0700	49	.0730	43	.0890	41	.0960
		64	.0668	50	.0700	48	.0760				
3	.0990	48	.0734	45	.0785	44	.0860	37	.1040	35	.1100
		56	.0771	43	.0820	43	.0890				
4	.1120	40	.0813	43	.0890	41	.0960	32	.1160	30	.1285
		48	.0864	42	.0935	40	.0980				
5	.1250	40	.0943	38	.1015	7/64	.1094	30	.1285	29	.1360
		44	.0971	37	.1040	35	.1100				
6	.1380	32	.0997	36	.1065	32	.1160	27	.1440	25	.1495
		40	.1073	33	.1130	31	.1200				
8	.1640	32	.1257	29	.1360	27	.1440	18	.1695	16	.1770
		36	.1299	29	.1360	26	.1470				
10	.1900	24	.1389	25	.1495	20	.1610	9	.1960	7	.2010
		32	.1517	21	.1590	18	.1695				
12	.2160	24	.1649	16	.1770	12	.1890	2	.2210	1	.2280
		28	.1722	14	.1820	10	.1935				
1/4	.2500	20	.1887	7	.2010	7/32	.2188	F	.2570	H	.2660
		28	.2062	3	.2130	1	.2280				
5/16	.3125	18	.2443	F	.2570	J	.2770	P	.3230	Q	.3320
		24	.2614	I	.2720	9/32	.2812				
3/8	.3750	16	.2983	5/16	.3125	Q	.3320	W	.3860	X	.3970
		24	.3239	Q	.3320	S	.3480				
7/16	.4375	14	.3499	U	.3680	25/64	.3906	29/6	.4531	15/32	.4687
		20	.3762	25/64	.3906	13/32	.4062				
1/2	.5000	13	.4056	27/64	.4219	29/64	.4531	33/6	.5156	17/32	.5312
		20	.4387	29/64	.4531	15/32	.4688				
9/16	.5625	12	.4603	31/64	.4844	33/64	.5156	37/6	.5781	19/32	.5938
		18	.4943	33/64	.5156	17/32	.5312				
5/8	.6250	11	.5135	17/32	.5312	9/16	.5625	41/64	.6406	21/32	.6562
		18	.5568	37/64	.5781	16/32	.5938				
3/4	.7500	10	.6273	21/32	.6562	11/16	.6875	49/6	.7656	25/32	.7812
		16	.6733	11/16	.6875	45/64	.7031				
7/8	.8750	9	.7387	49/64	.7656	51/64	.7969	57/6	.8906	29/32	.9062
		14	.7874	13/16	.8125	53/64	.8281				
1	1.000	8	.8466	7/8	.8750	59/64	.9219	1-	1.0156	1-1/32	1.0313
		12	.8978	15/16	.9375	61/64	.9531				

TAPER PIPE—DRILL AND TAP CHART (NPT)

Tap Size	Threads per Inch	Tap Drill
1/8	27	11/32
1/4	18	7/16
3/8	18	37/64
1/2	14	23/32
3/4	14	59/64
1	11-1/2	1-5/32
1-1/4	11-1/2	1-1/2
1-1/2	11-1/2	1-3/4
2	11-1/2	2-7/32
2-1/2	8	2-21/32
3	8	3-1/4
3-1/2	8	3-3/4
4	8	4-1/4

SCREW THREAD—DRILL AND TAP CHART (UNEF)

Machine Screw Size				Tap Drills				Clearance Hole Drills			
				75% Thread		50% Thread		Close Fit		Free Fit	
# or Diameter	Major Diameter			Drill Size	Decimal Equivalent	Drill Size	Decimal Equivalent	Drill Size	Decimal Equivalent	Drill Size	Decimal Equivalent
12	.2160	32	.1777	13	.1850	9	.1960	2	.2210	1	.2280
1/4	.2500	32	.2117	7/32	.2188	1	.2280	F	.2570	H	.2660
5/16	.3125	32	.2742	9/32	.2812	L	.2900	P	.3230	Q	.3320
3/8	.3750	32	.3367	11/32	.3438	T	.3580	W	.3860	X	.3970
7/16	.4375	28	.3937	Y	.4040	Z	.4130	29/64	.4531	15/32	.4687
1/2	.5000	28	.4562	15/32	.4688	31/64	.4844	33/64	.5156	17/32	.5312
9/16	.5625	24	.5114	33/64	.5156	17/32	.5312	37/64	.5781	19/32	.5938
5/8	.6250	24	.5739	37/64	.5781	19/32	.5938	41/64	.6406	21/32	.6562
11/16	.6825	24	.6364	41/64	.6406	21/32	.6562	45/64	.7031	23/32	.6562
3/4	.7500	20	.6887	45/64	.7031	23/32	.7188	49/64	.7656	25/32	.7812
13/16	.8125	20	.7512	49/64	.7656	25/32	.7812	53/64	.8281	27/32	.8438
7/8	.8750	20	.8137	53/64	.8281	27/32	.8438	57/64	.8906	29/32	.9062
15/16	.9375	20	.8762	57/64	.8906	29/32	.9062	61/64	.9531	31/32	.9688
1	1.000	20	.9387	61/64	.9531	31/32	.9688	1-1/64	1.0156	1-1/32	1.0313

METALWORKING LUBRICANTS

Material	Type of Operation		
	Drilling	**Threading**	**Milling or Turning**
Aluminum	Soluble oil Kerosene	Soluble oil Kerosene	Soluble oil
Brass	Dry Soluble oil Kerosene	Soluble oil	Soluble oil
Bronze	Dry Soluble oil	Soluble oil	Soluble oil
Cast Iron	Dry Air jet Soluble oil	Dry Sulfurized oil	Dry Soluble oil
Copper	Dry Soluble oil Kerosene	Lard oil	Soluble oil
Machine Steel	Soluble oil Sulfurized oil Mineral lard oil	Soluble oil Mineral lard oil	Soluble oil
Malleable Iron	Dry Soda water	Lard oil Soda water	Soluble oil Soda water
Monel	Soluble oil	Soluble oil	Lard oil
Steel Alloys	Soluble oil Sulfurized oil	Sulfurized oil	Soluble oil
Tool Steel	Soluble oil Sulfurized oil	Sulfurized oil	Soluble oil