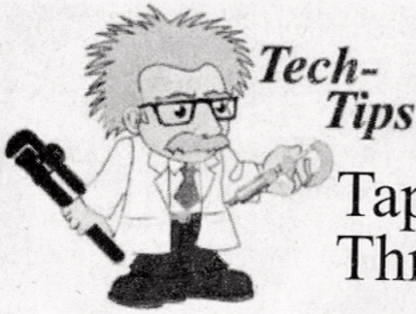


Tapping & Threading

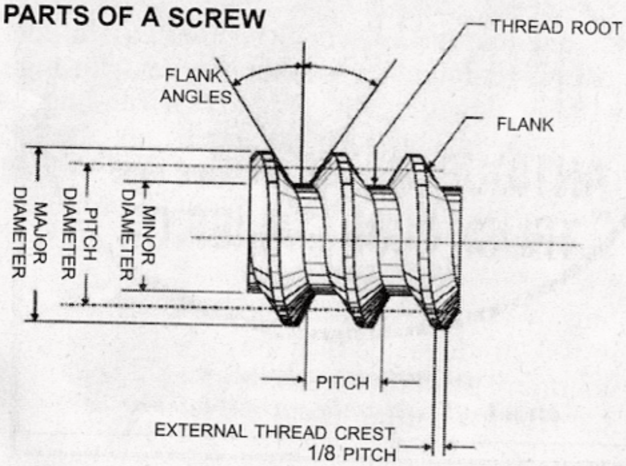
One of the topics that we are regularly questioned about are the correct procedures for using taps and dies, cutting new threads, putting threads into a hole and restoring damaged threads.

Quite a few years ago we ran an article on the use of taps and dies. It looks like it is time to look at the subject again. We will call on some of the material from the earlier article in this look at threaded fasteners.

I guess we have to start with basics. The thread of a bolt or hole is not merely a spiral cut into the metal. It is a very complex system of geometry designed to yield the strongest holding power. Some of the factors that go into figuring a thread are: Leading-flank angle, Trailing-flank angle, Pitch-cylinder diameter, Major diameter, Minor diameter, Lead angle, Helix angle, Pitch, Threads per inch, Length of engagement and Centerline or thread axis.

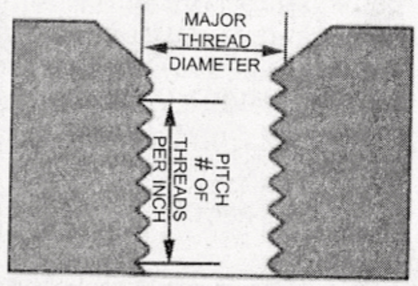
Don't panic - here's the good news: you don't really have to understand all of that technical talk. When you buy a bolt or a nut, two primary measurements are what you need to know: major diameter and pitch.

PARTS OF A SCREW



When you buy a quarter-twenty bolt, what you are actually looking for is a bolt with a major diameter of 0.250" or 1/4", and 20 threads to the inch. You pick up your bolt, pay your quarter and never think about what actually went into designing your 25¢ purchase. Fortunately, you don't have to; someone else did it for you. But you should know just a little more about that bolt.

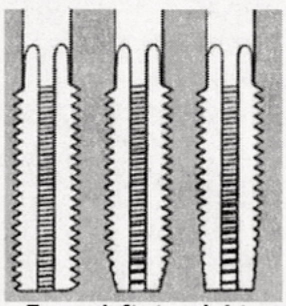
That's about the briefest description of a bolt that you are ever going to get. But there are two other very important factors: first, the threads within the hole must match the threads of the bolt. Your 1/4-20 bolt will not fit into a hole that is 1/4-28. Your bolt has 20 threads per inch, and the hole has 28. They don't fit together. The second factor is that the threads (flanks) are not worn to the point that their geometry has changed - that is, that the bolt is too loose in the hole. The hole, by the way, could be a threaded hole in a chunk of metal, or it could be in the form of a fastener called a nut.



Enough basics; let's get to the topic at hand. When you cut threads onto a shaft of metal you will use a specially designed cutter - a large nut, for want of another description - which is hardened and incorporates all of the necessary factors in thread geometry. It is called a 'DIE'. To cut threads in a hole, we use a shaft of metal, also hardened with the cutting surfaces engineered to produce a hole with the necessary factors to match a threaded bolt. It is called a 'TAP'. Both the shaft of metal onto which the threads are being cut, and the hole into which the threads are cut must be of a predetermined diameter so that there is adequate metal to provide threads that will grip and hold tight to the mating fastener.

We will only discuss hand-tapping; using a tapping machine is a specialized area requiring special and often expensive equipment. For the occasional hole that needs threads, hand tapping will suffice.

Although there are many variations and special application taps, for normal hand-tapping operations there are three types of taps: the plug tap, often called a 'general purpose tap', a bottoming tap, specifically designed to cut threads as close as possible to the bottom of a blind hole, and a tapered tap, a tap designed for starting a tapped hole and for ease of cutting. The tapered tap generally has about seven to ten chamfered threads at the beginning to ease the tap into the cutting job. Because of the large number of chamfered threads, the tapered tap is normally used for open holes and coarse threads (NCT). It is often used on harder material like stainless or carbon steel. The plug tap has about three to five chamfered threads and is normally adequate for cutting threads in most materials. It is often used as a starter tap for a blind hole and works well on softer materials such as brass, cast iron, aluminum and most plastics. A bottoming thread has only one-and-a-half chamfered threads. It is not the right tool for starting a thread, but once the initial threads are cut it can be used to go down to the bottom (except for that 1 ½ threads) of a blind hole.



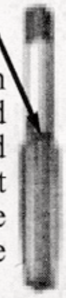
From left to right: a bottoming tap, a plug tap and a tapered tap.

This is probably as good a place as any to discuss terminology.

GLOSSARY:

Die - A cutting tool designed to cut threads on the outside of a rod or shaft such as on a bolt or machine screw. It can be used to chase existing threads on a bolt or threaded rod.

Tap - A cutting tool designed to cut threads inside a hole. It can be used to chase threads which may have been damaged, slightly stripped or are filled with residue. A tap (of the same size and thread as the existing hole) will not repair badly stripped threads. When the metal is gone a new, larger tapped hole will have to replace it.



Reduced Shaft Tap - The shaft is designed to break before the thread portion of the tap to allow ease of removal.

On a reduced shaft tap, the shaft has a 'waist' intended to break before the threaded portion of the tap does.

Drill bit - A cutting tool designed to cut a hole in material. A drill will leave flat, smooth sides on a hole. Threads have to be cut into this hole to secure a bolt or screw.

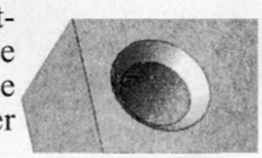
Drill bit sizes - Drill bits are size-designated in one of three categories: small bits, under ¼" diameter, which are measured by a number: 1 (0.2280" diameter) is the largest going down to (normally) 60 (0.0400" diameter), but often in miniature bits much smaller than that; fractional sizes and finally letter-designated: A (0.2340" diameter) to Z (0.4130").

Drill jig - A tool designed to assure that the angle of a hole is exact. Hand drilling is not accurate enough to ensure the proper angle. An adjustable drilling jig can set the angle from ninety degrees to the surface (perpendicular) to a specific number of degrees off perpendicular when required.

Basic O.D. of threads - The outside diameter of a given thread. The threads of a tap measure slightly over basic and the threads of a screw measure slightly under.

Clearance drill - A drill bit that cuts a hole just big enough for a screw to pass through without engaging the threads or binding in the hole. In making a jig for a tap, the tap must just pass through without binding and with a minimum of free play to eliminate wobble.

Chamfer - Refers to cutting of the edges of a hole or rod. For taps and dies the chamfer offers an easier start for the cutting tool.



CHAMFERED HOLE

Tap drill - A bit that cuts a hole of the best size for tapping as illustrated on the accompanying charts. Often it is 75% of the thread, but may be fractionally more on harder materials.

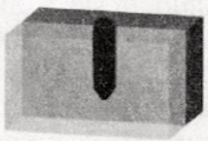
Tapping - Cutting threads inside a hole with a tap.

Threading - Cutting outside threads with a die.

Chasing threads - Refers to running a die over existing threads to straighten bent threads and reform threads that are damaged or filled with residue.

Open Hole - A hole that is open from both sides. May or may not be accessible from both sides.

Blind Hole - A hole that bottoms out in the work, where the other side of the hole is not accessible.



Tapping fluids - A lubricant specially formulated to help a tap (or die) cut through the metal. Tapping fluid is not the same as (although often interchanged for) cutting fluid which is designed to cool a part being cut on a lathe.

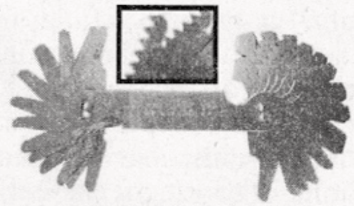
Thread-repair inserts - are individual "threaded holes" that can be placed inside a previously tapped hole which has been stripped of usable threads. Various systems are utilized by manufacturers to insure that their inserts will not back out and will provide bolt-holding strength. One of the most popular brands of thread-repair inserts is Heli-Coil.

Heli-Coils - are formed screw thread coils of stainless steel wire having a diamond shaped cross section. When properly installed, they provide permanent screw threads that accommodate any standard bolt or screw.

Thread diameter - The nominal width of a thread from outside to outside. A ¼" screw refers to the nominal diameter. (See Basic O.D. of threads above.) A ¼" tap will be fractionally larger than ¼" and a ¼" screw will be fractionally smaller.

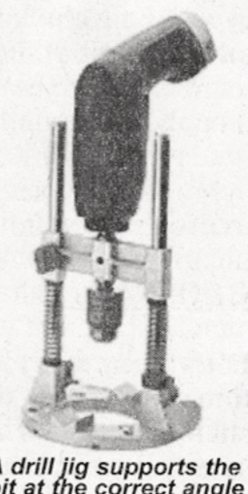
Thread pitch - (For U.S. threads) - The number of threads per inch. The higher the number the more threads (finer pitch) per inch. The lower the number, the fewer threads (coarser pitch) per inch. Most U.S. screws and bolts are available in both fine (NFT) or coarse (NCT), and often in extra fine, meaning even more threads per inch (NEF). An example would be ¼-20 (NCT), ¼-28 (NFT) and ¼-32 (NEF). Both the diameter and the thread pitch of a tap must match the intended bolt/screw size. Metric threads are referenced as the 'thread pitch' not 'thread count'. The metric figure indicates the distance in mm between the pitch of one thread and the next.

Thread gauge - or pitch gauge - A tool for measuring the number of threads per inch.



Each blade has a certain number of notches that will match the bolt or screw. Each blade is marked with the pitch number.

Let's discuss some precautions before we start discussing procedures. First, always wear eye protection. Metal chips or shards are very sharp and often tiny and can easily embed themselves in your eye, requiring a trip to the doctor or hospital. Those same chips can find their way under your skin providing painful metal splinters. Next, cleanliness. When working on a machine, such as a car engine, those chips can migrate into moving parts or into oil galleries where they will do damage later on. Drilling accuracy: this is where that drilling jig comes in handy - actually it is indispensable. Most often a hole will have to be drilled at 90 degrees to the surface (perpendicular), but sometimes the hole has to be angled in. The jig will hold the drill or bit at the exact angle to assure a properly drilled hole. An example of what can happen if the hole is not perfect might be with drilling, tapping or heli-coiling cylinder head studs. The studs, if not per-



A drill jig supports the bit at the correct angle to the work.

pendicular to the surface, will prevent the head from being removed in the future, requiring removal of the studs - often a major job - to lift the head off. Heli-coils, if installed in a non-perpendicular hole, will not grip the threads properly and not give adequate holding power. Using a tap at an angle, even slight, to the drilled hole will often result in a broken tap. Taps are made of carbon or hardened steel and will require special machine work, sometimes utilizing diamond-tipped drill bits or electrostatic procedures to remove them. This can get to be expensive. Taps will also break, if the chips are not cleaned or 'broken'. A good, safe rule is to never turn a tap more than one full turn without backing it out and cleaning the threads, or at least backing it out half a turn to break the chip. Forcing a tap will generally result in a broken tool (and that expensive extraction noted above). Some taps are made with a reduced shank size. This is to encourage the tap to break, if it is going to break, above the hole making removal of the tap easier.

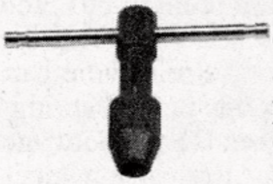
Make sure the tap hole is of the correct size, that the tap is lubricated and that the tap is sharp. Taps are tools, and their cutting edges do get dull. If a tap is not working smoothly, don't force it. Try a new tap. If it works well, put the old one aside. It cannot be sharpened, but it can be used for chasing.

Okay, let's get to the mechanics of drilling and tapping a hole. More often than not, the size of the bolt to be used in a tapped hole will be pre-determined, either by the type of application or by the similar bolts doing the same type of work. There are complicated formulas for determining what size tap drill is necessary for a given size bolt. The general rule of thumb is that the hole diameter should be 75% of the outside thread diameter of the intended bolt, but it is easier to refer to a chart for virtually every size screw/bolt, either U.S. or metric. Such a chart is shown on pages 42 and 43. By referring to the chart, a bolt of a given size will require a drill bit of the proper size to drill the pilot hole. With softer material, 75% will allow for adequate thread to provide holding power; harder materials require just a little less thread, and the pilot hole can be fractionally larger. As indicated above, the pilot hole must be drilled precisely. The work must be held securely in place in a vise. A drill jig will keep the bit at the correct angle to the work. A commercially available drill jig will do the job, or you can make one by drilling a hole in a cube of metal (this hole must be done on a drill press to assure accuracy. A second cube can be made with a hole - or a second hole in the first cube - to assure that the tap is in the same alignment. This hole should be just large enough to allow for clearance of the tap).

It is imperative, especially when drilling or redrilling a blind hole, that the depth be determined beforehand and a depth stop placed on the drill bit to keep from drilling too deep. This could be a serious problem, for example, when redrilling a hole in the cylinder block. You could easily inadvertently drill into a water or oil passage or into the combustion chamber. Slightly chamfer the very top of the hole using a countersink bit to allow the tap an easy first bite into the metal.

Once the pilot hole is drilled, all of the metal chips must be collected and discarded to prevent them from getting into the moving parts of the engine or impeding the tapping operation. We recommend against using compressed air to clean the chips; that could blow them into areas from which they cannot be retrieved.

Insert the correct size tap into a T-handle, and dip the point into the tapping fluid. Being sure that the tap is held perfectly straight, begin to rotate it in a clockwise direction using just light pressure. The chamfer on the tap and the chamfer in the hole will allow the tap to begin cutting into the metal. Do not try to get more than one complete turn out of the tap before removing it. This will break the chip and allow you to clean the chip away



A T-handle is made to hold the tap.

from the tool. As you tap deeper into the hole, stop at least once each complete turn and reverse direction of the T-handle for half a turn to break the chip. Every two complete rotations or so remove the tap from the hole, clean up metal chips, relubricate the tap and begin again.

If tapping an open hole, allow the tap to pass through the hole far enough so that the chamfered threads of the tap are completely clear of the bottom of the hole. If tapping a blind hole, proceed only until the tip of the tap bottoms out; do not force the tap past this point. Remove the tool and replace the tapered or plug tap with a bottoming tap. With this tool you should be able to get another couple of threads beyond what the first tap could provide. The threads will extend to within one or two threads of the bottom of the hole. Remove the tap, clean up the metal chips and use a solvent to clean the lubricant and any oil or grease from the hole. If not cleaned, the lubricant or other contaminants will prevent a sealant or thread-locker from doing its job.

Tapping a hole is not difficult, but the hole must be straight, the tap sharp and of the right type, and the work kept clean. If done properly, a newly tapped hole or a retapped hole will be as strong as the original from the factory.

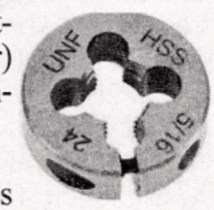
Hand tapping with a tap wrench.

Cutting a thread on a rod or shaft with a die is also a rather simple procedure. Many of the warnings that apply to taps also apply to dies. Be sure that the rod or shaft size is not larger than the designated bolt size. If the shaft is too large, it can be turned down on a lathe. Chamfer the end of the rod, too, to allow the die to get a start. Most dies are marked with a 'start side'. If it is not marked, careful examination will indicate that one or two threads in the die are chamfered to allow an easier start. Before you get to the end of the desired thread length, reverse the die in the holder so that you get clean threads right up to the end.

Use the tapping fluid on your work, and again, for at least each complete revolution, back off the die half a turn to break the chip. Every two full revolutions remove the die and clean the metal shards from the die and the work.

A die handle will hold the tool and afford necessary leverage.

Most dies have a minor adjustment. A split die allows an adjusting screw to be tightened (increasing die diameter) or loosened (reducing die diameter).



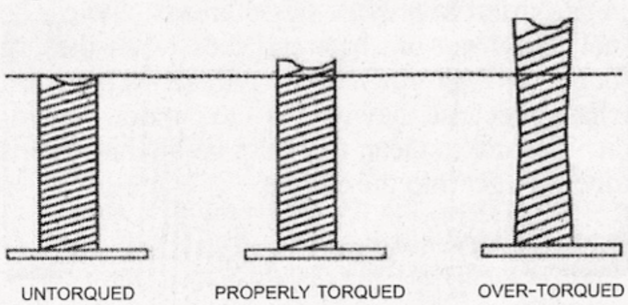
A special procedure is known as 'Chasing a Thread'. Chasing entails a mechanical system of cleaning, straightening and reforming otherwise distorted threads without removing additional metal. Normally the tap or die is designed to cut through metal forming new and sharp threads to provide maximum hold for a bolt. The chase - also known as a die nut or rethreading die - will remove debris from the threads, making installation of the part easier. A chase is essentially an undersized tap (or an oversized die) that will not cut away new material but will reform what remains. If the threads have previously been stripped out, that is physically removed from a bolt or hole, chasing will not restore the threads nor will it help provide the necessary strength to hold the two parts together. In that case one of two options is open to you: retap the hole to a larger size, or use a thread-repair insert to provide fresh threads. A trick: do not throw out old taps and dies when they've begun to wear down. They will act as excellent chases because they tend not to cut new material. Be sure to clean the work so that the debris does not get into the engine.

When working with nuts and bolts, it is important that the working parts are tightened just enough to provide holding power, but not enough so that the stress actually 'rips' the threads apart, destroying the mechanical bond. This degree of 'enough holding power' is pre-determined by the size of the bolt and the material of which it is constructed. This degree of force is called 'torque', and although we have discussed this topic a number of times in the past, it is important enough to include a basic torque chart (p. 41) indicating the correct foot pounds for various sized bolts of common and hardened material.

When properly torqued, a bolt will stretch (I know, it's steel. It still stretches). This stretching exerts a pressure against the female threads in the hole or the nut. It's that pressure that keeps the two parts tight. You want to stretch the bolt gently and in at least three increments. That's why it is recommended that on the first go-round, you torque to $^1/_3$ of the desired foot pounds, on the next application of pressure, up to $^2/_3$ and then finally to the full torque value.

The amount of 'stretch' is dependent on the type of material, the diameter of the material and the hardness of the material. When you refer to a Motor's Manual or a Chilton's for torque values, they are given in OEM hardnesses. Generally, a Grade 5 bolt was used. Using a Grade 2/3 will over-torque the bolt; using a Grade 8 will under-torque it.

More is not better. Over-torquing will stretch the bolt beyond its optimum recovery ability. It can also put undo strain on the female threads in the block, which can result in stripped threads (and then some serious repair work).



The second important point to be noted is that provision must be made to prevent the accidental loosening of a bolt due to vibration while in use. Most often we put a 'spring' adjacent to the nut or bolt head to keep it from working loose. Commonly this spring takes the form of an offset ring of metal which exerts a force against the nut or bolt head. This 'spring' is known as a lock washer. Another option is that of a chemical bond between the two parts.

In actuality, only about 15% of the threads of a bolt make contact with the nut/threaded hole. The remaining 85% is air gap. A liquid threadlocker eliminates the need of a lock washer and actually fills the air gap with a material that binds the bolt to the nut/hole. As the threadlocker cures, it fills that 85% with a plastic-type material that keeps the bolt from vibrating out, while providing much greater holding power than the bolt alone, or a bolt with a lock washer.

An Anomoly - The Exception to the Rule

Most taps are measured by the maximum diameter of the cutting shaft. A 1/4" tap will measure (approximately) 0.250". The exception is special taps made for National Pipe Thread (NPT). Here it gets confusing because the size called out has no relationship to actual measurements. In addition, pipe thread is tapered in order to provide a good seal. The taper is 1" per 16" of length (0.750" per foot).

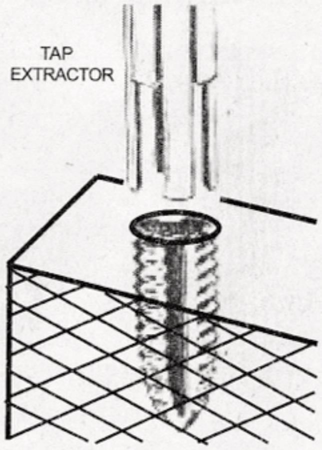
American Standard Pipe Thread

Pipe Size	TPI	Nominal Diameter
1/16"	27	0.313"
1/8"	27	0.405"
1/4"	18	0.540"
3/8"	18	0.675"
1/2"	14	0.840"
3/4"	14	1.050"
1"	11-1/2	1.315"
1-1/4"	11-1/2	1.66"
1-1/2"	11-1/2	1.900"
2"	11-1/2	2.375"

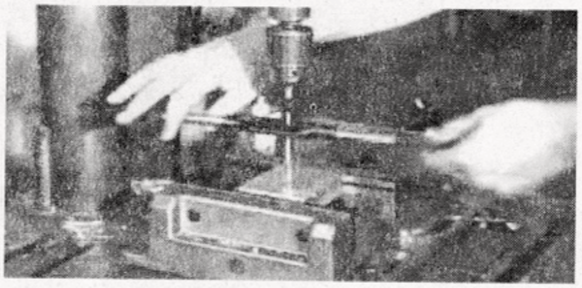
TRICKS AND HINTS FOR SUCCESSFUL TAPPING

A broken tap is generally difficult to remove and requires special tools and often the technical expertise of a machine shop. If a reduced shaft tap is used, there is a good chance that enough of the tap will remain above the hole so that you can grasp it with a tap extractor or a vise grip. Even if the tap is broken flush with or slightly below the work surface, an extractor may be able to grip it.

It is imperative that the extractor or vise grip be kept perfectly straight. Attempting to remove the tap at an angle is almost certainly going to result in further damage to the broken part, possibly making it impossible to remove it with hand tools.



A trick for keeping the hand-tap perfectly straight is to use a drill press as a guide. DO NOT TURN THE MOTOR ON. It is used merely to hold and guide the hand tap into the work which must be held securely in a vise. Most hand taps have a square end which fits into a tap wrench or a T-handle. A normal drill chuck will not hold the tap by the square end. The chuck will have to be fastened over the round segment of the shaft above the threads. Be sure to lubricate the tap with tapping fluid. A flux brush used in soldering is the perfect size to apply the fluid.



TORQUING A BOLT

Providing adequate holding power without ripping out threads

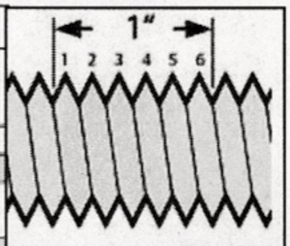
A properly tightened bolt is one that is stretched so that it acts like a very rigid spring pulling mating surfaces together. The rotation of a bolt (torque) at some point causes it to stretch (tension). Several factors affect how much tension occurs when a given amount of tightening torque is applied. The first factor is the bolt's diameter. It takes more force to tighten a 3/4-10 bolt than to tighten a 3/8-16 bolt because it is larger in diameter. The second factor is the bolt's grade. It takes more force to stretch an SAE Grade 8 bolt than it does to stretch an SAE Grade 5 bolt because of the greater material strength. The third factor is the coefficient of friction, frequently referred to as the "nut factor." The value of this factor indicates that harder, smoother, and/or slicker bolting surfaces, such as threads and bearing surfaces, require less rotational force (torque) to stretch (tension) a bolt than do softer, rougher, and stickier surfaces. These charts apply to clean and dry parts. A lubricated bolt requires less torque to attain the same clamping force as a non-lubricated bolt. The highly recognized engineering formula for determining torque is $T = K \times D \times P$ where 'T' = target tighten torque (the result of this formula is in inch pounds, dividing by 12 yields foot pounds); 'K'= coefficient of friction (nut factor), always an estimation in this formula; 'D'= bolt's nominal diameter in inches; 'P' = bolt's desired tensile load in pounds (generally 75% of yield strength). Figures at the right are for steel bolts.

Bolt size-Thread pitch	Grade 2	Grade 5	Grade 8
1/4-20	6	10	12
1/4-28	7	12	15
5/16-18	13	20	24
5/16-24	14	22	27
3/8-16	23	36	44
3/8-24	26	40	48
7/16-14	37	52	63
7/16-20	41	57	70
1/2-13	57	80	98
1/2-20	64	90	110
9/16-12	82	120	145
9/16-18	91	135	165
5/8-11	111	165	210
5/8-18	128	200	245
3/4-10	200	285	335
3/4-16	223	315	370

Figures above in ft lbs. for steel or iron. Certain bolt materials have more or less 'stretch' and require special torque charts.

Explanation of torque credited to:
http://www.zerofast.com/torque.htm

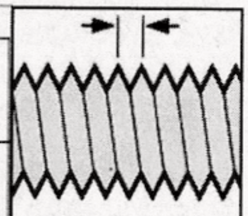
Tap & Clearance Drill Sizes				Tap Drill				Clearance Drill	
Screw Size	Major Diameter	Threads Per Inch	Minor Diameter	Drill Size for Aluminum, Brass, & Plastics		Drill Size for Steel, Stainless, & Iron		Free Fit	
				Drill Size	Dec. Eq.	Drill Size	Dec. Eq.	Drill Size	Dec. Eq.
0	.0600	80	.0447	3/64	.0469	55	.0520	50	.0700
1	.0730	64	.0538	53	.0595	1/16	.0625	46	.0810
		72	.0560	53	.0595	52	.0635		
2	.0860	56	.0641	50	.0700	49	.0730	41	.0960
		64	.0668	50	.0700	48	.0760		
3	.0990	48	.0734	47	.0785	44	.0860	35	.1100
		56	.0771	45	.0820	43	.0890		
4	.1120	40	.0813	43	.0890	41	.0960	30	.1285
		48	.0864	42	.0935	40	.0980		
5	.125	40	.0943	38	.1015	7/64	.1094	29	.1360
		44	.0971	37	.1040	35	.1100		
6	.138	32	.0997	36	.1065	32	.1160	25	.1495
		40	.1073	33	.1130	31	.1200		
8	.1640	32	.1257	29	.1360	27	.1440	16	.1770
		36	.1299	29	.1360	26	.1470		
10	.1900	24	.1389	25	.1495	20	.1610	7	.2010
		32	.1517	21	.1590	18	.1695		
12	.2160	24	.1649	16	.1770	12	.1890	1	.2280
		28	.1722	14	.1820	10	.1935		
		32	.1777	13	.1850	9	.1960		
1/4	.2500	20	.1887	7	.2010	7/32	.2188	H	.2660
		24	.1975	4	.2090	2	.2210		
		28	.2062	3	.2130	1	.2280		
		32	.2117	7/32	.2188	1	.2280		
5/16	.3125	18	.2443	F	.2570	J	.2770	Q	.3320
		24	.2614	I	.2720	9/32	.2812		
		32	.2742	9/32	.2812	L	.2900		
3/8	.3750	16	.2983	5/16	.3125	Q	.3320	X	.3970
		24	.3239	Q	.3320	S	.3480		
		32	.3367	11/32	.3438	T	.3580		
7/16	.4375	14	.3499	U	.3680	25/64	.3906	15/32	.4687
		20	.3762	25/64	.3906	13/32	.4062		
		28	.3937	Y	.4040	Z	.4130		
1/2	.5000	13	.4056	27/64	.4219	29/64	.4531	17/32	.5312
		20	.4387	29/64	.4531	15/32	.4688		
		28	.4562	15/32	.4688	15/32	.4688		
9/16	.5625	12	.4603	31/64	.4844	33/64	.5156	19/32	.5938
		18	.4943	33/64	.5156	17/32	.5312		
		24	.5114	33/64	.5156	17/32	.5312		
5/8	.6250	11	.5135	17/32	.5312	9/16	.5625	21/32	.6562
		18	.5568	37/64	.5781	19/32	.5938		
		24	.5739	37/64	.5781	19/32	.5938		
11/16	.6875	24	.6364	41/64	.6406	21/32	.6562	23/32	.6562
3/4	.7500	10	.6273	21/32	.6562	11/16	.6875	25/32	.7812
		16	.6733	11/16	.6875	45/64	.7031		
		20	.6887	45/64	.7031	23/32	.7188		
13/16	.8125	20	.7512	49/64	.7656	25/32	.7812	27/32	.8438
7/8	.8750	9	.7387	49/64	.7656	51/64	.7969	29/32	.9062
		14	.7874	13/16	.8125	53/64	.8281		
		20	.8137	53/64	.8281	27/32	.8438		
15/16	.9375	20	.8762	57/64	.8906	29/32	.9062	31/32	.9688
1	1.000	8	.8466	7/8	.8750	59/64	.9219	1-1/32	1.0313
		12	.8978	15/16	.9375	61/64	.9531		
		20	.9387	61/64	.9531	31/32	.9688		



US - Threads Per Inch (TPI)

Metric Tap & Clearance Drill Sizes

		Tap Drill				Clearance Drill	
		Drill Size for Aluminum, Brass, & Plastics		Drill Size for Steel, Stainless, & Iron		Free Fit	
Screw Size (mm)	Thread Pitch (mm)	Drill Size (mm)	Closest American Drill	Drill Size (mm)	Closest American Drill	Drill Size (mm)	Closest American Drill
M1.5	0.35	1.15	56	1.25	55	1.65	52
M1.6	0.35	1.25	55	1.35	54	1.75	50
M 1.8	0.35	1.45	53	1.55	1/16	2.00	5/64
M 2	0.45	1.55	1/16	1.70	51	2.20	44
	0.40	1.60	52	1.75	50		
M 2.2	0.45	1.75	50	1.90	48	2.40	41
M 2.5	0.45	2.05	46	2.20	44	2.75	7/64
M 3	0.60	2.40	41	2.60	37	3.30	30
	0.50	2.50	39	2.70	36		
M 3.5	0.60	2.90	32	3.10	31	3.85	24
M 4	0.75	3.25	30	3.50	28	4.40	17
	0.70	3.30	30	3.50	28		
M 4.5	0.75	3.75	25	4.00	22	5.00	9
M 5	1.00	4.00	21	4.40	11/64	5.50	7/32
	0.90	4.10	20	4.40	17		
	0.80	4.20	19	4.50	16		
M 5.5	0.90	4.60	14	4.90	10	6.10	B
M 6	1.00	5.00	8	5.40	4	6.60	G
	0.75	5.25	4	5.50	7/32		
M 7	1.00	6.00	B	6.40	E	7.70	N
	0.75	6.25	D	6.50	F		
M 8	1.25	6.80	H	7.20	J	8.80	S
	1.00	7.00	J	7.40	L		
M 9	1.25	7.80	N	8.20	P	9.90	25/64
	1.00	8.00	O	8.40	21/64		
M 10	1.50	8.50	R	9.00	T	11.00	7/16
	1.25	8.80	11/32	9.20	23/64		
	1.00	9.00	T	9.40	U		
M 11	1.50	9.50	3/8	10.00	X	12.10	15/32
M 12	1.75	10.30	13/32	10.90	27/64	13.20	33/64
	1.50	10.50	Z	11.00	7/16		
	1.25	10.80	27/64	11.20	7/16		
M 14	2.00	12.10	15/32	12.70	1/2	15.50	39/64
	1.50	12.50	1/2	13.00	33/64		
	1.25	12.80	1/2	13.20	33/64		
M 15	1.50	13.50	17/32	14.00	35/64	16.50	21/32
M 16	2.00	14.00	35/64	14.75	37/64	17.50	11/16
	1.50	14.50	37/64	15.00	19/32		
M 17	1.50	15.50	39/64	16.00	5/8	18.50	47/64
M 18	2.50	15.50	39/64	16.50	41/64	20.00	25/32
	2.00	16.00	5/8	16.75	21/32		
	1.50	16.50	21/32	17.00	43/64		
M 19	2.50	16.50	21/32	17.50	11/16	21.00	53/64
M 20	2.50	17.50	11/16	18.50	23/32	22.00	55/64
	2.00	18.00	45/64	18.50	47/64		
	1.50	18.50	47/64	19.00	3/4		



Metric - Thread pitch

S.K.