1.4 METAL CUTTING BAND SAWS:

Metals can be bought from suppliers in standardized forms and sizes, such as round, rectangular or square bar stock or in the form of large sheets (plates). Bar stock normally is available in lengths of up to 4 [m], sheets in dimensions up to 1.2 [m] x 2.4 [m]. This means that for most “jobs” the bar stock needs to be cut to length; this is normally done using band saws. Sheets are pre-cut using flame or plasma cutting machines or shearing machines and can then be further cut using band saws.

.

There are two types of band saws used for cutting metal: the vertical band saw and the horizontal cut-off saw.

1.4.1 Vertical band saw:

It is used to cut metal plate to a required approximate size, the so-called “blank”, which is then converted into the final product using machines such as the Mill, the Drill Press or the Lathe. Bar stock can be cut on a vertical band saw, but the preferred machine is the horizontal band saw.

A continuous blade runs in a vertical plane, in a clock-wise direction. Most of it is hidden inside the housing of the machine and only a small part (slightly more than the thickness of the material to be cut) is exposed between the table and the bottom end of the blade guard. The blade guard can be raised or lowered by loosening and tightening the guard adjustment knob.

The work-piece must be resting on, and be fully supported by the table; do not cut round bar stock in this type of saw, unless you are using a machining vise to hold the bar.

Cutting speeds are always a function of the type of blade being used (number of teeth per inch [tpi]) and the material being cut. The most critical parameter is the rate at which the operator pushes (feeds) the material into the blade. Although slow feed rates dull the blade more quickly, they are preferred over high feed rates which can lead to a blade breaking (generally not dangerous, but it can be a bit scary).

Some vertical metal cutting band saws are equipped with a blade welder; saw blades used to be bought in 100 feet length and then cut to the required length and welded together. Today blades are shipped by the supplier in the required lengths.

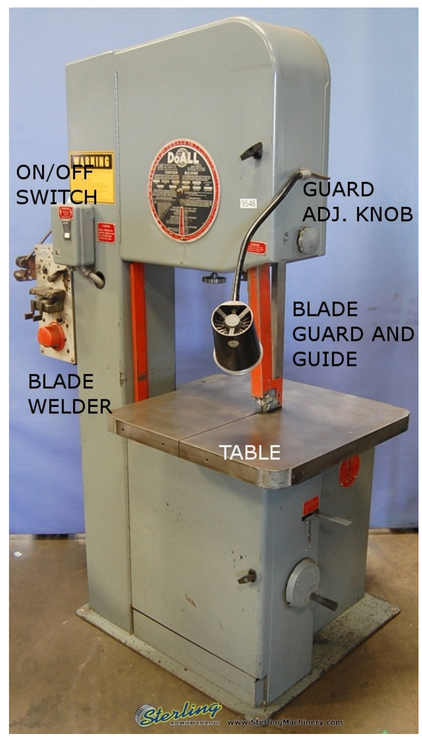


Figure 1.4.1: Vertical Band Saw

1.4.2 Horizontal Band Saw:

This type of saw is used to cut blank pieces from bar stock (round, square, rectangular, L-shaped etc. “extruded” profiles).

Figure 1.4.2: Horizontal Band Saw (Cut-Off Saw)

The motor, drive unit and blade are in the tilting upper part of the machine. While the saw is in this “open” position, the work-piece can be clamped, after having been positioned correctly between the vise jaws. Then the upper half of the machine is slowly lowered towards the work-piece. The blade is started before contact between the blade and the work-piece occurs. The weight of the upper saw assembly provides the necessary feed force; do not push down on the housing.
As before, only expose as much of the blade as is necessary. For that purpose the blade guard can be moved, exposing more or less of the blade. If the machine is equipped with a cutting fluid nozzle, start the cutting fluid before starting the blade; on some machines the start button turns on both, the cutting fluid and the machine.

1.4.3 Band saw blade types and cutting speeds:

Band saw blades are defined by the "number of teeth per inch" (TPI) and the width of the blade. The width of the blade determines the minimum radius of a curve that can be cut, whereas the type of material that is being cut determines the number of teeth per inch required to produce a good cut at the correct cutting speed.

In general, most band saws are set to an "average" speed and equipped with an "average" TPI blade of "average" width.

TPI:	-for soft and/or thin materials use band saw blades with high TPI numbers, such as 24 - 32 TPI -for aluminum use 6 - 10 TPI -for steels use 10 - 14 TPI
Cutting speeds:	for softer materials use higher cutting speeds -aluminum: 200 and higher feet per minute [fpm] -brass and bronze: 150 - 250 [fpm] -cast iron and mild steel: 50 - 100 [fpm]

1.5 HAND TOOLS:

This refers to tools which may be used following machining operations as outlined in the previous chapters.

1.5.1 Deburring Tool:

The tools described here are used to deburr (= dull) the edges of newly machined parts. These edges would otherwise represent a serious safety hazard when handling the part.

Figure 1.5.1: Deburring Tool

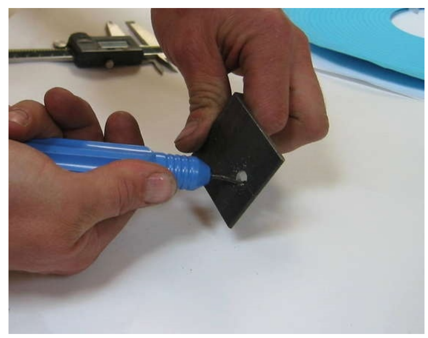


Figure 1.5.2: Deburring a Hole

The hardened tip of the deburring tool shown in Figure 1.5.1 is, at an angle of approximately 45 degrees, moved over the just machined edges. This process removes any burrs which may be present following a machining process. To deburr holes, a tool similar in appearance to a counter-sink tool and shown in Figure 1.5.3 can also be used.

Figure 1.5.3: Hole Deburring Tool

1.5.2 Files:

Files, for the purpose of this document, are another deburring tool. They are available in different serrations and profiles; the most common serration patterns is referred to as a bastard pattern. The file card, shown in Figure 1.5.4 is used to clean built up metal from the file.

Flat files can be used to deburr straight edges, half-round files are suitable for curved edges or large holes and round files can be used on holes. To deburr an edge, move the file forth and back in the direction of the long axis of the tool (as shown in Figure 1.5.4), putting light pressure on the file during the forward-stroke, but no pressure on the back-stroke.

Note: if you value your tools (or those in the shop), do not use files on Aluminum; it has a tendency to permanently clog files; use a deburring tool instead.

Figure 1.5.4: Flat Mill Bastard File

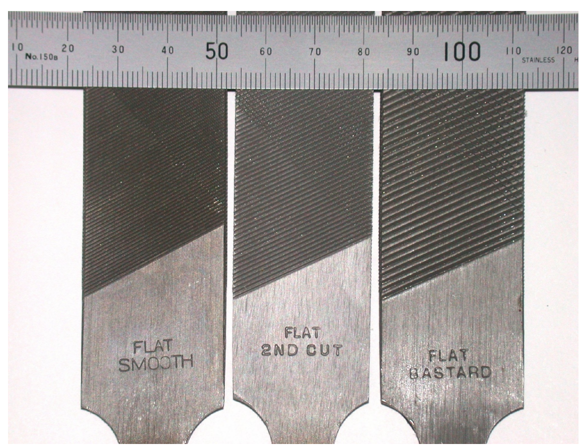


Figure 1.5.5: Flat File Profiles

As can be seen in Figure 1.5.5, bastard files are medium-coarse files; 2nd cut and smooth files have finer profiles.

1.5.3 Tapping tools:

Tapping is the process of producing internal or external threads. For small diameter threads taps and dies are used to cut the threads; large or odd diameter threads, internal or external, can be produced on the lathe.

Figure 1.5.6: Tap and Die Set

Threads are classified by their profile, their diameter and the number of threads per inch or the pitch (in the case of metric threads).

Tap and die sets are normally cutting Unified, American National or the Metric thread profiles; taps and dies for other thread profiles are also available (but not easy to find).

Some Thread Designations:

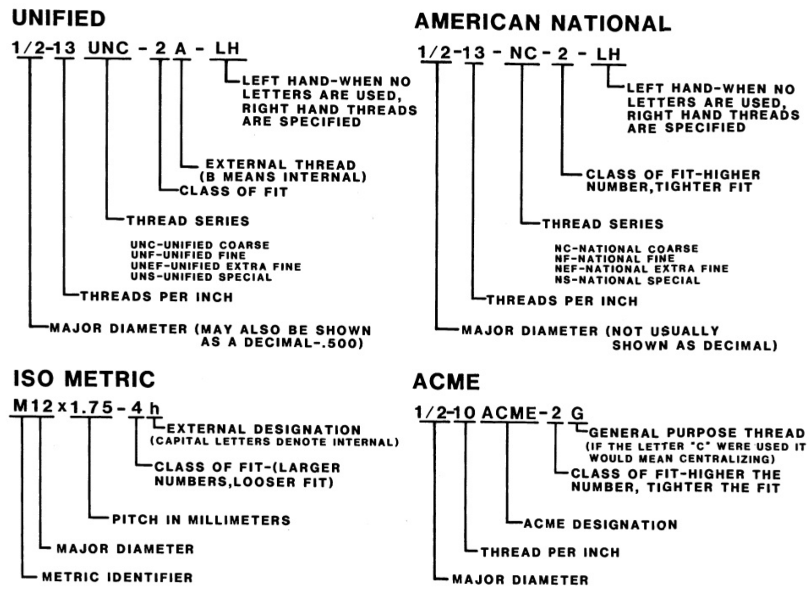


Figure 1.5.7: Thread Designations

Some Thread Profiles:

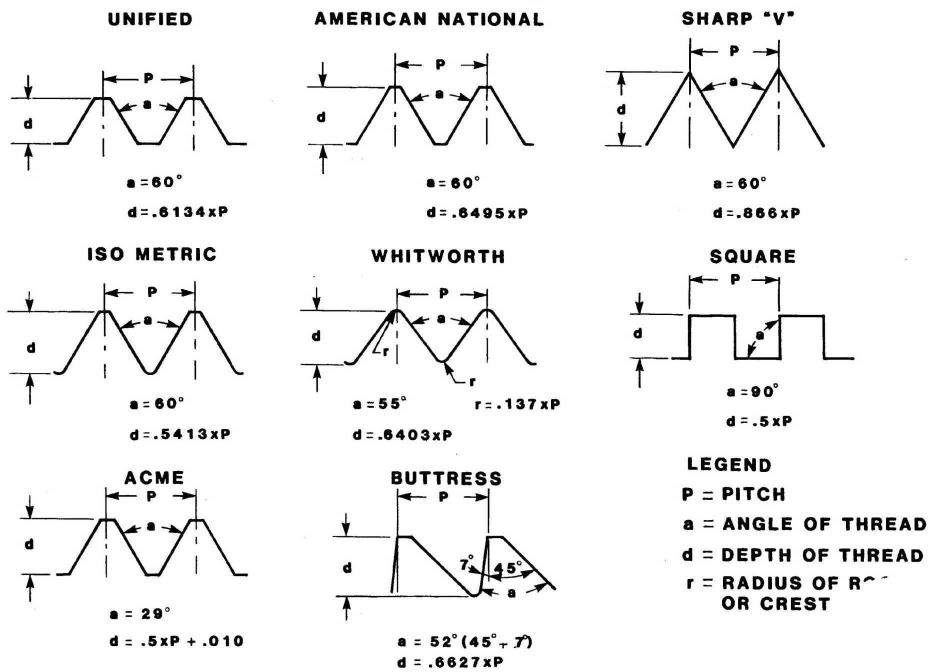


Figure 1.5.8: Thread Profiles

American National and Unified, as well as Sharp V, profiles are profiles used in the U.S. and Canada and are now identified as Unified National (UN) thread profiles; they are used for all types of "inch" based fasteners. The ISO Metric, or simply "M" thread profile is used for metric fasteners. The Whitworth profile is used in Britain for fasteners and is almost identical to the UN profile. Square, Acme and Buttress profiles are used for power screws: those are screws used to convert torque into large linear forces for lifting or moving objects (such as moving the carriage on a lathe).

1.5.4 Creating internal threads:

Before creating an internal thread, a hole with a diameter smaller than the nominal tap (thread) size must be drilled. The required hole diameters are critical and can be found from tables. The following Tap Drill Size Tables are taken from the "Starrett Tool Company, Reference Tables".

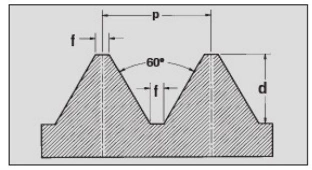
American National and Unified Coarse and Fine Thread Dimensions and Tap Drill Sizes

$p = \text{pitch} = \dfrac{1}{\text{No. thread per inch}}$

$d = \text{depth} = p \times .649519$

$f = \text{flat} = \dfrac{p}{8}$

$\text{pitch diameter} = D - \dfrac{.6495}{N}$



Size	Threads per inch NC UNC	Threads per inch NF UNF	Outside Diameter Inches	Pitch Diameter Inches	Root Diameter Inches	Tap Drill Approx. 75% Full Thread	Decimal Equiv. of Tap Drill
0	–	80	.0600	.0519	.0438	3/64″	.0469
1	64	–	.0730	.0629	.0527	53	.0595
1	–	72	.0730	.0640	.0550	53	.0595
2	56	–	.0860	.0744	.0628	50	.0700
2	–	64	.0860	.0759	.0657	50	.0700
3	48	–	.0990	.0855	.0719	47	.0785
3	–	56	.0990	.0874	.0758	46	.0810
4	40	–	.1120	.0958	.0795	43	.0890
4	–	48	.1120	.0985	.0849	42	.0935
5	40	–	.1250	.1088	.0925	38	.1015
5	–	44	.1250	.1102	.0955	37	.1040
6	32	–	.1380	.1177	.0974	36	.1065
6	–	40	.1380	.1218	.1055	33	.1130
8	32	–	.1640	.1437	.1234	29	.1360
8	–	36	.1640	.1460	.1279	29	.1360
10	24	–	.1900	.1629	.1359	26	.1470
10	–	32	.1900	.1697	.1494	21	.1590
12	24	–	.2160	.1889	.1619	16	.1770
12	–	28	.2160	.1928	.1696	15	.1800
1/4″	20	–	.2500	.2175	.1850	7	.2010
1/4″	–	28	.2500	.2268	.2036	3	.2130
5/16″	18	–	.3125	.2764	.2403	F	.2570
5/16″	–	24	.3125	.2854	.2584	I	.2720
3/8″	16	–	.3750	.3344	.2938	5/16″	.3125
3/8″	–	24	.3750	.3479	.3209	Q	.3320
7/16″	14	–	.4375	.3911	.3447	U	.3680
7/16″	–	20	.4375	.4050	.3726	25/64″	.3906
1/2″	13	–	.5000	.4500	.4001	27/64″	.4219
1/2″	–	20	.5000	.4675	.4351	29/64″	.4531
9/16″	12	–	.5625	.5084	.4542	31/64″	.4844
9/16″	–	18	.5625	.5264	.4903	33/64″	.5156
5/8″	11	–	.6250	.5660	.5069	17/32″	.5312
5/8″	–	18	.6250	.5889	.5528	37/64″	.5781
3/4″	10	–	.7500	.6850	.6201	21/32″	.6562
3/4″	–	16	.7500	.7094	.6688	11/16″	.6875
7/8″	9	–	.8750	.8028	.7307	49/64″	.7656
7/8″	–	14	.8750	.8286	.7822	13/16″	.8125

Size	Threads per inch NC UNC	Threads per inch NF UNF	Outside Diameter Inches	Pitch Diameter Inches	Root Diameter Inches	Tap Drill Approx. 75% Full Thread	Decimal Equiv. of Tap Drill
1″	8	–	1.0000	.9188	.8376	7/8″	.8750
1″	–	12	1.0000	.9459	.8917	59/64″	.9219
1 1/8″	7	–	1.1250	1.0322	.9394	63/64″	.9844
1 1/8″	–	12	1.1250	1.0709	1.0168	1 3/64″	1.0469
1 1/4″	7	–	1.2500	1.1572	1.0644	1 7/64″	1.1094
1 1/4″	–	12	1.2500	1.1959	1.1418	1 11/64″	1.1719
1 3/8″	6	–	1.3750	1.2667	1.1585	1 7/32″	1.2187
1 3/8″	–	12	1.3750	1.3209	1.2668	1 19/64″	1.2969
1 1/2″	6	–	1.5000	1.3917	1.2835	1 11/32″	1.3437
1 1/2″	–	12	1.5000	1.4459	1.3918	1 27/64″	1.4219
1 3/4″	5	–	1.7500	1.6201	1.4902	1 9/16″	1.5625
2″	4 1/2	–	2.0000	1.8557	1.7113	1 25/32″	1.7812
2 1/4″	4 1/2	–	2.2500	2.1057	1.9613	2 1/32″	2.0313
2 1/2″	4 1/2	–	2.5000	2.3376	2.1752	2 1/4″	2.2500
2 3/4″	4	–	2.7500	2.5876	2.4252	2 1/2″	2.5000
3″	4	–	3.0000	2.8376	2.6752	2 3/4″	2.7500
3 1/4″	4	–	3.2500	3.0876	2.9252	3″	3.0000
3 1/2″	4	–	3.5000	3.3376	3.1752	3 1/4″	3.2500
3 3/4″	4	–	3.7500	3.5876	3.4252	3 1/2″	3.5000
4″	4	–	4.0000	3.3786	3.6752	3 3/4″	3.7500

Figure 1.5.9: UNC, UNF, NC and NF Tap Drill Sizes

Millimeter Tap Drill Sizes

Metric Tap	Tap Drill mm	Decimal Equiv. Inches	Metric Tap	Tap Drill mm	Decimal Equiv. Inches	Metric Tap	Tap Drill mm	Decimal Equiv. Inches	Metric Tap	Tap Drill mm	Decimal Equiv. Inches
M1.6 x 0.35	1.25	.0492	M6 x 1	5.00	.1968	M16 x 2	14.00	.5512	M27 x 3	24.00	.9449
M1.8 x 0.35	1.45	.0571	M7 x 1	6.00	.2362	M16 x 1.5	14.50	.5709	M27 x 2	25.00	.9843
M2 x 0.4	1.60	.0630	M8 x 1.25	6.70	.2638	M18 x 2.5	15.50	.6102	M30 x 3.5	26.50	1.0433
M2.2 x 0.45	1.75	.0689	M8 x 1	7.00	.2756	M18 x 1.5	16.50	.6496	M30 x 2	28.00	1.1024
M2.5 x 0.45	2.05	.0807	M10 x 1.5	8.50	.3346	M20 x 2.5	17.50	.6890	M33 x 3.5	29.50	1.1614
M3 x 0.5	2.50	.0984	M10 x 1.25	8.70	.3425	M20 x 1.5	18.50	.7283	M33 x 2	31.00	1.2205
M3.5 x 0.6	2.90	.1142	M12 x 1.75	10.20	.4016	M22 x 2.5	19.50	.7677	M36 x 4	32.00	1.2598
M4 x 0.7	3.30	.1299	M12 x 1.25	10.80	.4252	M22 x 1.5	20.50	.8071	M36 x 3	33.00	1.2992
M4.5 x 0.75	3.70	.1457	M14 x 2	12.00	.4724	M24 x 3	21.00	.8268	M39 x 4	35.00	1.3780
M5 x 0.8	4.20	.1654	M14 x 1.5	12.50	.4921	M24 x 2	22.00	.8661	M39 x 3	36.00	1.4173

Figure 1.5.10: Metric Tap Drill Sizes

Example: *A production drawing calls for 4 threaded holes ½ -13 UNC 2B on a part. Select the appropriate tap drill size.*

From Figure 1.5.9: tap drill size for a ½ - 13 UNC hole is 27/64"

Another production drawing calls for 4 x M12 x 1.25 holes. You only have inch-drills available, but have the required metric tap. What inch size drill bit would you select for the production of those 4 metric holes?

From Figure 1.5.10: tap drill size for a M12 x 1.25 hole is 10.80 [mm] or 0.4252"
The closest inch-size drill bits would be 27/64" or 7/16" (0.4219" or 0.4375").

Always select the slightly larger drill bit; this might produce a slightly looser fit between the part and the mating screw, but it will not break the tap (something that might happen with an undersized hole).

1.6 MACHINE SHOP SAFETY BASICS:

Before being allowed to operate any of the machinery in the Machine Shop MC 78, you will receive safety instructions from the Machine Shop Staff.
The following “rules” are common to the operation of any machine described in these notes and are intended as a frame-work for shop safety.

1. Shoes with leather uppers, covering the whole foot, must be worn. Laces must be tied neatly to avoid tripping. No sneakers or open-toed shoes.

2. No loose clothing, especially no loose belts and no wide, long-sleeved garments. Loose sleeves must be rolled up securely; do not slide them up your arm. Also, do not wear shorts.
   Although not required, an apron or work-coat (similar to a lab-coat) could be used.

3. Jewellery must not be worn in the Machine Shop (exception small ear rings).

4. Long hair must be safely tied back.

5. Safety Glasses must be worn at all times.

6. Do not attempt to communicate with another person, while that person is in the middle of a machining operation.

7. Never leave an operating machine unattended.

8. After pressing the Stop button, wait until all moving parts have come to rest, before taking measurements, turning away or leaving the machine.

9. If you notice an unsafe situation at one of the other machines, shut your machine down safely and then alert the other operator.

10. Before starting any machine, make sure that all cutting tools are safely mounted and any chuck keys, wrenches (for tool changing), measuring tools, rags etc. have been removed from the immediate working area.

11. If not sure about something, ASK!!

12. Until properly deburred, handle newly machined parts with care to avoid cutting your hands and fingers.