Tap Drill Sizes

75% thread engagement — the standard for most work

The formula, if the chart isn't handy: Inch: tap drill = major diameter – (1 ÷ TPI). Metric: tap drill = major diameter – pitch. Both give roughly 75% thread.

Size (UNC)	Drill	Dec.
#4-40	#43	.089
#6-32	#36	.1065
#8-32	#29	.136
#10-24	#25	.1495
#12-24	#16	.177
1/4-20	#7	.201
5/16-18	F	.257
3/8-16	5/16	.3125
7/16-14	U	.368
1/2-13	27/64	.4219
9/16-12	31/64	.4844
5/8-11	17/32	.5312
3/4-10	21/32	.6562
7/8-9	49/64	.7656
1-8	7/8	.875

Size (UNF)	Drill	Dec.
#4-48	#42	.0935
#6-40	#33	.113
#8-36	#29	.136
#10-32	#21	.159
#12-28	#14	.182
1/4-28	#3	.213
5/16-24	I	.272
3/8-24	Q	.332
7/16-20	25/64	.3906
1/2-20	29/64	.4531
9/16-18	33/64	.5156
5/8-18	37/64	.5781
3/4-16	11/16	.6875
7/8-14	13/16	.8125
1-12	59/64	.9219

Metric (coarse)	Drill
M3 × 0.5	2.5 mm
M4 × 0.7	3.3 mm
M5 × 0.8	4.2 mm
M6 × 1.0	5.0 mm
M8 × 1.25	6.8 mm
M10 × 1.5	8.5 mm
M12 × 1.75	10.2 mm
M14 × 2.0	12.0 mm
M16 × 2.0	14.0 mm
M20 × 2.5	17.5 mm

Going for more thread engagement is usually a mistake. 75% is the practical sweet spot — beyond that you gain very little holding strength and dramatically increase the chance of snapping the tap. In tough or gummy material, a slightly larger drill (60–65% thread) will save you more taps than it costs you in strength.

Bolt Grades & Head Markings

What the marks on the head are telling you

SAE / Inch

Grade	Head marking	Material	Tensile	Yield
Grade 2	No marks	Low / medium carbon	74 ksi	57 ksi
Grade 5	3 radial lines	Medium carbon, Q&T	120 ksi	92 ksi
Grade 8	6 radial lines	Medium carbon alloy, Q&T	150 ksi	130 ksi
Socket head	Usually unmarked	Alloy steel (ASTM A574)	180 ksi	153 ksi

Metric

Class	Head marking	Material	Tensile	Yield
8.8	“8.8” on head	Medium carbon, Q&T	830 MPa	660 MPa
10.9	“10.9” on head	Alloy steel, Q&T	1040 MPa	940 MPa
12.9	“12.9” on head	Alloy steel, Q&T	1220 MPa	1100 MPa

Reading a metric number. The first figure × 100 gives tensile strength in MPa. Multiply the two figures together and × 10 gives yield. So 10.9 is 1000 MPa tensile, 900 MPa yield — close enough for the shop.

Two things worth knowing. A higher grade is not automatically better: harder bolts are more brittle and more notch-sensitive, and in some joints a Grade 5 outlives a Grade 8. And stainless is *not* a strength grade — 18-8 stainless is roughly Grade 2 in strength, not Grade 8.

Torque Reference

Coarse thread, plain finish — general guidance only

Read this first. These are general values for clean, plain-finish fasteners in a standard joint. **An engineering specification always overrides this card.** If the print, the WPS, the service manual or the customer gives you a torque value, use theirs. Never use a general chart on anything structural, pressurised, or safety-critical.

Size	Gr 5 dry (ft-lb)	Gr 5 lubed	Gr 8 dry (ft-lb)	Gr 8 lubed
1/4-20	8	6	12	9
5/16-18	17	13	24	18
3/8-16	31	23	44	33
7/16-14	49	37	70	52
1/2-13	75	57	106	80
9/16-12	109	82	153	115
5/8-11	150	112	212	159
3/4-10	266	200	376	282
7/8-9	429	322	606	455
1-8	644	483	909	682

Torque is not clamp load. You are measuring friction, and roughly 90% of the torque you apply is spent overcoming it. That is why lubrication changes the number so much — a lubed fastener at dry torque values can be stretched past yield.

Practical notes. Torque in a star or crossing pattern on anything with a gasket or flange. Bring everything up in two or three passes rather than one. Do not reuse a torque-to-yield (stretch) bolt. And store a click wrench set to its lowest setting, not wound up.

Fraction · Decimal · Millimetre

The conversions worth knowing cold

Fraction	Decimal	mm
1/32	0.0312	0.79
1/16	0.0625	1.59
3/32	0.0938	2.38
1/8	0.1250	3.17
5/32	0.1562	3.97
3/16	0.1875	4.76
7/32	0.2188	5.56
1/4	0.2500	6.35
9/32	0.2812	7.14
5/16	0.3125	7.94
11/32	0.3438	8.73
3/8	0.3750	9.52
13/32	0.4062	10.32
7/16	0.4375	11.11
15/32	0.4688	11.91
1/2	0.5000	12.70

Fraction	Decimal	mm
17/32	0.5312	13.49
9/16	0.5625	14.29
19/32	0.5938	15.08
5/8	0.6250	15.88
21/32	0.6562	16.67
11/16	0.6875	17.46
23/32	0.7188	18.26
3/4	0.7500	19.05
25/32	0.7812	19.84
13/16	0.8125	20.64
27/32	0.8438	21.43
7/8	0.8750	22.22
29/32	0.9062	23.02
15/16	0.9375	23.81
31/32	0.9688	24.61
1	1.0000	25.40

Quick conversions. Inches to mm: multiply by 25.4. Millimetres to inches: divide by 25.4, or multiply by 0.03937.

Worth memorising: 1/16 = .0625 · 1/8 = .125 · 1/4 = .250 · 3/8 = .375 · 1/2 = .500 · 5/8 = .625 · 3/4 = .750 · 7/8 = .875. Everything else you can build from those by halving or adding.

Material Grades at a Glance

What the number means and where it belongs

Carbon & Alloy Steel

Grade	Type	What it's for
1018	Low carbon	Weldable, case-hardens well, cheap. General shop steel.
1045	Medium carbon	Heat treatable. Shafts, gears, anything needing strength.
12L14	Leaded, free machining	Machines beautifully. Do not weld — the lead is a problem.
4140	Chrome-moly alloy	Tough and heat treatable. Tooling, shafts, high-stress parts.
A36	Structural	Weldable plate and bar. Wide chemistry tolerance — not for precision.
O1 / A2 / D2	Tool steel	O1 oil-hardening, A2 air-hardening, D2 wear-resistant.

Stainless

Grade	Type	What it's for
303	Free machining	Best machining stainless. **Not for welding** — sulphur causes cracking.
304	General purpose	The default. Weldable, formable, good corrosion resistance.
316	Marine / chemical	Molybdenum added — far better against chlorides and salt.
17-4 PH	Precipitation hardening	High strength plus corrosion resistance. Aerospace favourite.
416	Free machining, magnetic	Hardenable martensitic. Machines well, less corrosion resistant.

Aluminium

Grade	Type	What it's for
6061-T6	General purpose	The workhorse. Weldable, machinable, good all-round strength.
7075-T6	High strength	Aerospace. Stronger than many steels by weight. **Not weldable.**
2024-T3	Aerospace sheet	High fatigue strength. Poor corrosion resistance — usually clad.
5052	Formable / marine	Bends without cracking, excellent salt-water resistance.
MIC-6 / cast tool	Stable plate	Stress-relieved, stays flat. Fixture and tooling plate.

Temper designations. T6 = solution heat treated and artificially aged (the common one). T3 = solution treated, cold worked, naturally aged. O = annealed, dead soft. H32 etc. = strain hardened, for non-heat-treatable alloys like 5052.

Never substitute material without approval. It looks the same, machines the same, and fails differently.

Drill Sizes

Number, letter and fractional — smallest to largest

No.	Dec.
#1	.2280
#2	.2210
#3	.2130
#4	.2090
#5	.2055
#6	.2040
#7	.2010
#8	.1990
#9	.1960
#10	.1935
#11	.1910
#12	.1890
#13	.1850
#14	.1820
#15	.1800
#16	.1770
#17	.1730
#18	.1695
#19	.1660
#20	.1610
#21	.1590
#22	.1570
#23	.1540
#24	.1520
#25	.1495
#26	.1470
#27	.1440
#28	.1405
#29	.1360
#30	.1285

No.	Dec.
#31	.1200
#32	.1160
#33	.1130
#34	.1110
#35	.1100
#36	.1065
#37	.1040
#38	.1015
#39	.0995
#40	.0980
#41	.0960
#42	.0935
#43	.0890
#44	.0860
#45	.0820
#46	.0810
#47	.0785
#48	.0760
#49	.0730
#50	.0700
#51	.0670
#52	.0635
#53	.0595
#54	.0550
#55	.0520
#56	.0465
#57	.0430
#58	.0420
#59	.0410
#60	.0400

Ltr	Dec.
A	.234
B	.238
C	.242
D	.246
E	.250
F	.257
G	.261
H	.266
I	.272
J	.277
K	.281
L	.290
M	.295
N	.302
O	.316
P	.323
Q	.332
R	.339
S	.348
T	.358
U	.368
V	.377
W	.386
X	.397
Y	.404
Z	.413

Frac.	Dec.
1/32	0.0312
1/16	0.0625
3/32	0.0938
1/8	0.1250
5/32	0.1562
3/16	0.1875
7/32	0.2188
1/4	0.2500
9/32	0.2812
5/16	0.3125
11/32	0.3438
3/8	0.3750
13/32	0.4062
7/16	0.4375
15/32	0.4688
1/2	0.5000

Frac.	Dec.
17/32	0.5312
9/16	0.5625
19/32	0.5938
5/8	0.6250
21/32	0.6562
11/16	0.6875
23/32	0.7188
3/4	0.7500
25/32	0.7812
13/16	0.8125
27/32	0.8438
7/8	0.8750
29/32	0.9062
15/16	0.9375
31/32	0.9688
1	1.0000

The three systems overlap. Number drills run #80 (.0135) up to #1 (.228); letter drills pick up at A (.234) and run to Z (.413). Where you need a size between two fractions, a number or letter drill usually exists — that is what they are for. **A drilled hole is not a precision hole:** expect a twist drill to cut oversize, often by .002 to .005, and to wander on entry. If the print calls a tighter tolerance, drill undersize and ream or bore to size.

Speeds & Feeds

Starting points — not specifications

Two formulas do most of the work.

RPM = (SFM × 3.82) ÷ tool diameter in inches

Feed rate (IPM) = RPM × number of flutes × chip load

Material	HSS (SFM)	Carbide (SFM)
Aluminium (6061, 2024)	200–300	600–1200
Aluminium (cast)	150–250	400–800
Brass	150–250	400–800
Bronze	80–150	200–400
Cast iron (grey)	50–80	200–400
Low carbon steel (1018)	80–110	300–500
Medium carbon (1045)	60–90	250–400
Alloy steel (4140)	50–70	200–350
Tool steel (O1, A2, D2)	40–60	150–250
Stainless 303	70–100	250–400
Stainless 304	40–60	150–300
Stainless 316	35–50	150–250
Titanium	30–50	100–200
Inconel / superalloy	15–25	50–100
Plastics (acetal, nylon)	300–600	500–1000

End mill Ø	Chip load / tooth
1/8"	.0005 – .001
3/16"	.001 – .0015
1/4"	.001 – .002
3/8"	.002 – .003
1/2"	.003 – .004
5/8"	.0035 – .005
3/4"	.004 – .006
1"	.005 – .007

These are starting points and nothing more. Actual numbers depend on your machine's rigidity, the tool holder, depth of cut, coolant, and how the part is held. A rigid machining centre will run circles around a worn knee mill on the same material. Start conservative, listen to the cut, and work up.

Read the chips, not the chart. Long stringy chips mean the feed is too light. Blue or purple chips mean you are running too hot — usually too fast rather than too heavy. Powder means the tool is rubbing rather than cutting, which kills edges fast. Good chips come off in tight commas, and the colour tells you where the heat is going.

Depth of cut changes everything. These figures assume a moderate cut. Full-slot cutting wants roughly half the chip load and a slower feed; a light finishing pass will take more.

Surface Finish Reference

Ra in microinches — what each process actually produces

Process	Typical Ra (micro-in)	Where you see it
Flame cutting / sawing	250 – 1000	Rough, no finish requirement
Rough turning / milling	125 – 500	Stock removal
Finish turning / milling	32 – 125	General machined surfaces
Drilling	63 – 250	As-drilled bore
Reaming	32 – 125	Sized holes
Boring	16 – 125	Precision bores
Broaching	32 – 125	Keyways, internal forms
Grinding — rough	16 – 63	Bearing seats, shafts
Grinding — finish	8 – 32	Close-fit surfaces
Honing	4 – 16	Cylinder bores, hydraulic
Lapping	2 – 8	Sealing faces, gauges
Polishing / superfinish	1 – 8	Bearing races, seals
EDM — wire / sinker	32 – 250	Depends on settings

Common Print Callouts

Callout	What it means in practice
500	Rough machined, no cosmetic requirement
250	General machining, non-critical
125	Standard machined finish — the common default
63	Good machined finish, close-fitting parts
32	Fine — typically ground or finish-turned
16	Very fine — ground, sealing surfaces
8	Precision ground or honed

Ra is an average, not a maximum. It is the arithmetic average roughness — a surface can meet an Ra callout and still carry a scratch that fails the part. If the function depends on the worst defect rather than the average, the print should call Rz or Rmax instead.

Finer is not free. Every step down roughly doubles the cost. 125 comes off the machine, 32 usually means grinding, and 8 means honing or lapping and a separate operation. Do not chase a finish the print does not ask for.

Micrometres, if the print is metric. Divide microinches by 40 for a rough conversion — 125 micro-in is about 3.2 micrometres, 63 ≈ 1.6, 32 ≈ 0.8, 16 ≈ 0.4.

Reading a Title Block

Everything you need before you touch material

Field	What it tells you
Drawing number	The unique identifier. Must match your traveller and your material tag.
Revision	**Check this first, every time.** Rev C on the floor and Rev D released is the classic scrap job.
Part name / description	What it is. Useful for sanity-checking you have the right print.
Material	Grade and condition. Substituting is not a shop-floor decision.
Finish / treatment	Plating, anodise, heat treat, paint. Often affects final dimensions.
Scale	Never measure off the print. Scale is for reading, not for dimensioning.
Default tolerances	Applies to any dimension without its own tolerance. Usually by decimal place.
Units	Inch or millimetre. Dual-dimensioned prints are where mistakes live.
Drawn / checked / approved	Who to ask when something does not make sense.
Sheet x of y	Make sure you have all of them. Notes on sheet 3 still apply to sheet 1.

Default Tolerances — by Decimal Place

Dimension shown as	Common default	Note
X. (one place)	± .1" or ± .030"	Typical, but read your title block
X.XX (two places)	± .01" or ± .010"	Typical
X.XXX (three places)	± .005"	Typical
X.XXXX (four places)	± .0005"	Typical
Angles	± 0°30′ or ± 1°	Typical

“No tolerance shown” does not mean perfect. It means the default in the title block applies. It also does not mean “whatever you like”. The values above are common, but every company sets its own — read the block on the print in front of you.

Notes override the title block. The dimension overrides the notes. Specificity wins: a tolerance called on the dimension beats a general note, which beats the title block default.

Read the general notes before you start. They routinely carry things like “break all sharp edges”, “remove all burrs”, or a blanket surface finish — requirements that appear nowhere on the views but will fail the part at inspection.

Drawing Abbreviations & Symbols

The shorthand you will meet on almost every print

Symbol	Meaning
Ø	Diameter
R	Radius
SR	Spherical radius
Square symbol	Square feature
CL	Centreline
DIA	Diameter (written form)
THRU	Through
TYP	Typical — applies to all like features
REF	Reference only — not to be measured
BSC	Basic — theoretically exact, no tolerance
MAX / MIN	Maximum / minimum limit
NTS	Not to scale
EQ SP	Equally spaced
BC / BCD	Bolt circle / bolt circle diameter

Symbol	Meaning
CBORE	Counterbore
CSK	Countersink
SF	Spotface
DP	Depth (down-arrow symbol on prints)
TPI	Threads per inch
UNC / UNF	Unified coarse / fine thread
NPT	National pipe thread (tapered)
CHAM	Chamfer
FIL	Fillet
MACH	Machined
ASSY	Assembly
NOM	Nominal
GD&T	Geometric dimensioning & tolerancing
MMC / LMC	Maximum / least material condition

Three that cause the most trouble.

REF means reference only. It is there to help you understand the part, and it is not a dimension to work to or inspect against. If you machine to a REF dimension you are guessing.

TYP means the callout applies to every similar feature, not just the one the leader points at. Miss it and you machine one of eight holes correctly.

BSC or a boxed dimension is basic — theoretically exact, with no tolerance of its own. The tolerance comes from the geometric control that references it, not from the title block.

Hole callouts read top down. A note like “Ø.250 THRU, Ø.375 CBORE .190 DP” describes one feature in order of operation: the through hole first, then the counterbore diameter and depth.

GD&T Symbols

The fourteen geometric characteristics, ASME Y14.5

Form

- ⏤ Straightness
- ⏥ Flatness
- ○ Circularity
- ⌭ Cylindricity

Profile

- ⌒ Profile of a Line
- ⌓ Profile of a Surface

Orientation

- ∠ Angularity
- ⊥ Perpendicularity
- ∥ Parallelism

Location

- ⌖ Position
- ◎ Concentricity
- ⌯ Symmetry

Runout

- ↗ Circular Runout
- ⌰ Total Runout

Reading a feature control frame. Left to right: the characteristic symbol, then the tolerance value, then the datum references in order — primary, secondary, tertiary. The order matters: it tells you how the part is constrained for inspection, and setting it up in a different order means you are measuring a different part than the designer specified.

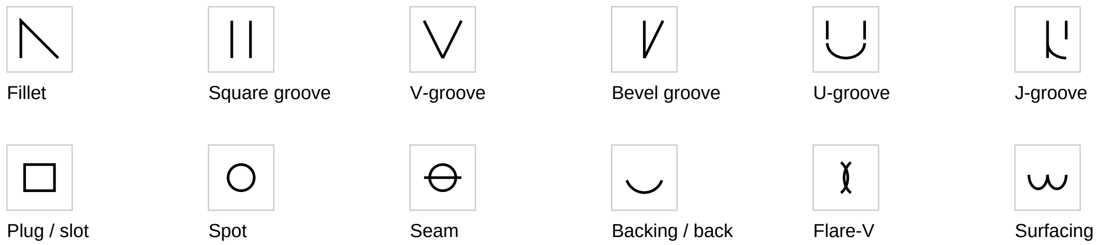
Two modifiers worth knowing. An M in a circle (MMC) means the tolerance gets larger as the feature departs from maximum material condition — free tolerance you have already been given. An L in a circle (LMC) is the reverse. A diameter symbol before the tolerance value means the zone is cylindrical rather than a flat band.

Basic dimensions — the ones in boxes — carry no tolerance of their own. They define the theoretically exact location, and the feature control frame does the tolerancing.

Weld Symbols

Reading the callout on the print

The rule everyone gets backwards. The symbol **below** the reference line means the weld goes on the **arrow side** — the side the arrow is pointing at. The symbol **above** the line means the **other side**. Symbols on both sides means weld both sides.



Supplementary symbols. A circle at the elbow (where the arrow meets the reference line) means *weld all around*. A filled flag at the same point means *field weld* — make it at installation, not in the shop. A tail at the far end carries the specification, process or WPS number.

Numbers and their positions. The figure to the *left* of the symbol is the weld size (leg length for a fillet, depth for a groove). Figures to the *right* are length and pitch — so 2-5 means a 2-inch weld every 5 inches, centre to centre. A groove angle sits above or below the symbol, and the root opening sits inside it.

If the size is missing on a fillet weld, the WPS or the general notes govern. Do not guess — an undersized fillet fails and an oversized one distorts the part and wastes filler.