

01 MACHINIST QUICK REFERENCE

A shop-floor reference for turning, milling, drilling, threading, measuring and common material decisions.

Built for fast answers at the machine.

20 compact pages - each designed to stand on its own when printed or laminated.

Page	Reference
02	Fraction / Decimal / Metric
03	Number, Letter & Fractional Drills
04	UNC Tap Drills
05	UNF Tap Drills
06	NPT Tap Reference
07	Thread Identification & Measurement
08	Lathe RPM / SFM
09	Milling RPM & Chip Load
10	Drilling Starting Speeds
11	Material Machining Notes
12	Common Steel Grades
13	Stainless & Aluminum
14	Fits & Clearances
15	Keys & Keyways
16	Surface Finish
17	Tapers & Angles
18	Bolt Circle / Coordinates
19	Machinist Formulas
20	Inspection & Setup Checklist

SHOP NOTE: Starting values only. Machine rigidity, tool geometry/coating, workholding, coolant, material condition and manufacturer recommendations always win.

02 FRACTION / DECIMAL / METRIC

Common inch fractions through 1 inch with decimal and millimeter equivalents.

Fraction	Decimal in	mm	Fraction	Decimal in	mm
1/64	0.0156	0.397	1/2	0.5156	13.097
1/32	0.0312	0.794	1/2	0.5312	13.494
3/64	0.0469	1.191	9/16	0.5469	13.891
1/16	0.0625	1.587	9/16	0.5625	14.287
5/64	0.0781	1.984	9/16	0.5781	14.684
3/32	0.0938	2.381	9/16	0.5938	15.081
7/64	0.1094	2.778	5/8	0.6094	15.478
1/8	0.1250	3.175	5/8	0.6250	15.875
9/64	0.1406	3.572	5/8	0.6406	16.272
5/32	0.1562	3.969	5/8	0.6562	16.669
11/64	0.1719	4.366	11/16	0.6719	17.066
3/16	0.1875	4.762	11/16	0.6875	17.462
13/64	0.2031	5.159	11/16	0.7031	17.859
7/32	0.2188	5.556	11/16	0.7188	18.256
15/64	0.2344	5.953	3/4	0.7344	18.653
1/4	0.2500	6.350	3/4	0.7500	19.050
1/4	0.2656	6.747	3/4	0.7656	19.447
1/4	0.2812	7.144	3/4	0.7812	19.844
5/16	0.2969	7.541	13/16	0.7969	20.241
5/16	0.3125	7.938	13/16	0.8125	20.637
5/16	0.3281	8.334	13/16	0.8281	21.034
5/16	0.3438	8.731	13/16	0.8438	21.431
3/8	0.3594	9.128	7/8	0.8594	21.828
3/8	0.3750	9.525	7/8	0.8750	22.225
3/8	0.3906	9.922	7/8	0.8906	22.622
3/8	0.4062	10.319	7/8	0.9062	23.019
7/16	0.4219	10.716	15/16	0.9219	23.416
7/16	0.4375	11.112	15/16	0.9375	23.812
7/16	0.4531	11.509	15/16	0.9531	24.209
7/16	0.4688	11.906	15/16	0.9688	24.606
1/2	0.4844	12.303	1	0.9844	25.003
1/2	0.5000	12.700	1	1.0000	25.400

1 in = 25.4 mm exactly | mm / 25.4 = inches | inches x 25.4 = mm

03 NUMBER, LETTER & FRACTIONAL DRILLS

Useful drill diameters for tap selection, clearance work and identifying an unknown drill.

Drill	in	mm	Drill	in	mm	Drill	in	mm
80	0.0135	0.34	70	0.0280	0.71	60	0.0400	1.02
50	0.0700	1.78	40	0.0980	2.49	36	0.1065	2.71
32	0.1160	2.95	30	0.1285	3.26	29	0.1360	3.45
28	0.1405	3.57	27	0.1440	3.66	26	0.1470	3.73
25	0.1495	3.80	24	0.1520	3.86	23	0.1540	3.91
22	0.1570	3.99	21	0.1590	4.04	20	0.1610	4.09
19	0.1660	4.22	18	0.1695	4.31	17	0.1730	4.39
16	0.1770	4.50	15	0.1800	4.57	14	0.1820	4.62
13	0.1850	4.70	12	0.1890	4.80	11	0.1910	4.85
10	0.1935	4.91	9	0.1960	4.98	8	0.1990	5.05
7	0.2010	5.11	6	0.2040	5.18	5	0.2055	5.22
4	0.2090	5.31	3	0.2130	5.41	2	0.2210	5.61
1	0.2280	5.79	A	0.2340	5.94	B	0.2380	6.05
C	0.2420	6.15	D	0.2460	6.25	E	0.2500	6.35
F	0.2570	6.53	G	0.2610	6.63	H	0.2660	6.76
I	0.2720	6.91	J	0.2770	7.04	K	0.2810	7.14
L	0.2900	7.37	M	0.2950	7.49	N	0.3020	7.67
O	0.3160	8.03	P	0.3230	8.20	Q	0.3320	8.43
R	0.3390	8.61	S	0.3480	8.84	T	0.3580	9.09
U	0.3680	9.35	V	0.3770	9.58	W	0.3860	9.80
X	0.3970	10.08	Y	0.4040	10.26	Z	0.4130	10.49

SHOP NOTE: For a full drill index, use the manufacturer's index card. This page emphasizes the number/letter range most often encountered in tapping.

04 UNC TAP DRILLS

Common 60-degree Unified thread tap drills. Values are practical shop references; verify special thread percentage requirements.

Thread	Tap drill	Decimal	Approx. mm
#4-40	#43	0.0890	2.26
#6-32	#36	0.1065	2.71
#8-32	#29	0.1360	3.45
#10-24	#25	0.1495	3.80
#12-24	#16	0.1770	4.50
1/4-20	#7	0.2010	5.11
5/16-18	F	0.2570	6.53
3/8-16	5/16	0.3125	7.94
7/16-14	U	0.3680	9.35
1/2-13	27/64	0.4219	10.72
9/16-12	31/64	0.4844	12.30
5/8-11	17/32	0.5312	13.49
3/4-10	21/32	0.6562	16.67
7/8-9	49/64	0.7656	19.45
1-8	7/8	0.8750	22.22
1-1/8-7	63/64	0.9844	25.00
1-1/4-7	1-7/64	1.1094	28.18
1-1/2-6	1-11/32	1.3438	34.13

Rule of thumb for inch threads: Tap drill dia. $\approx$ major diameter - (1 / TPI)

SHOP NOTE: Hard/tough materials often benefit from a slightly larger tap drill; thread-forming taps require a different drill size.

05 UNF TAP DRILLS

Common 60-degree Unified thread tap drills. Values are practical shop references; verify special thread percentage requirements.

Thread	Tap drill	Decimal	Approx. mm
#4-48	#42	0.0935	2.37
#6-40	#33	0.1130	2.87
#8-36	#29	0.1360	3.45
#10-32	#21	0.1590	4.04
#12-28	#14	0.1820	4.62
1/4-28	#3	0.2130	5.41
5/16-24	I	0.2720	6.91
3/8-24	Q	0.3320	8.43
7/16-20	25/64	0.3906	9.92
1/2-20	29/64	0.4531	11.51
9/16-18	33/64	0.5156	13.10
5/8-18	37/64	0.5781	14.68
3/4-16	11/16	0.6875	17.46
7/8-14	13/16	0.8125	20.64
1-12	59/64	0.9219	23.42
1-1/8-12	1-3/64	1.0469	26.59
1-1/4-12	1-11/64	1.1719	29.77
1-1/2-12	1-27/64	1.4219	36.12

Rule of thumb for inch threads: Tap drill dia. $\approx$ major diameter - (1 / TPI)

SHOP NOTE: Hard/tough materials often benefit from a slightly larger tap drill; thread-forming taps require a different drill size.

06 NPT TAP REFERENCE

Common National Pipe Taper thread sizes. NPT taper is 1 in 16 on diameter (3/4 in per foot).

NPT	Common tap drill	Drill dia. in
1/16-27	15/64	0.242
1/8-27	R	0.339
1/4-18	7/16	0.438
3/8-18	37/64	0.578
1/2-14	23/32	0.719
3/4-14	59/64	0.922
1-11.5	1-5/32	1.156
1-1/4-11.5	1-1/2	1.500
1-1/2-11.5	1-47/64	1.734
2-11.5	2-7/32	2.219

NPT taper = 1:16 on diameter | Included thread angle = 60°

SHOP NOTE: Pipe-tap drill recommendations vary by tap style, material and desired engagement. Use the tap maker's drill recommendation for production work; gauge finished NPT threads.

07 THREAD IDENTIFICATION & MEASUREMENT

Fast checks for Unified and pipe threads.

Feature	Quick reference
UNC / UNF	60° included angle; measure major diameter and TPI.
NPT	60° thread; tapered 1:16 on diameter. Gauge with proper NPT plug/ring gauge.
BSPT	55° Whitworth form; tapered. Do not treat as interchangeable with NPT.
Pitch	Distance from crest to crest. Inch pitch = 1 / TPI.
Major dia.	Largest thread diameter.
Minor dia.	Smallest thread diameter.
Pitch dia.	Imaginary diameter where thread and space widths are equal.
3-wire method	Accurate external pitch-diameter measurement when correct wire size and constants are used.

Best wire size for 60° thread $\approx 0.57735 \times \text{pitch}$

SHOP NOTE: Use a pitch gauge to identify TPI, then verify diameter. Never force an NPT fitting into BSPT because it 'almost fits.'

08 LATHE RPM / SFM

Starting spindle-speed reference. Reduce speed for interrupted cuts, poor rigidity, long overhang or difficult material.

$$\text{RPM} = (\text{SFM} \times 3.82) / \text{Diameter (in)}$$

Material	HSS SFM start	Carbide SFM start
Low-carbon steel	80-100	250-450
4140 prehard	45-70	180-300
304/316 stainless	35-60	120-220
Cast iron	60-90	200-350
Aluminum	200-400	600-1200
Brass / bronze	150-300	400-800

Dia.	50 SFM	100 SFM	200 SFM	300 SFM
0.25"	764	1528	3056	4584
0.5"	382	764	1528	2292
0.75"	255	509	1019	1528
1"	191	382	764	1146
1.5"	127	255	509	764
2"	96	191	382	573
3"	64	127	255	382
4"	48	96	191	286
6"	32	64	127	191
8"	24	48	96	143
10"	19	38	76	115
12"	16	32	64	96

09 MILLING RPM & CHIP LOAD

Starting-point formulas for end mills and face mills.

$$\text{RPM} = (\text{SFM} \times 3.82) / \text{Cutter Dia.} \quad | \quad \text{IPM} = \text{RPM} \times \text{Flutes} \times \text{Chip Load}$$

Tool dia.	Carbide end mill chip/tooth - light start
1/8	0.0005-0.0015
1/4	0.001-0.0025
3/8	0.0015-0.003
1/2	0.002-0.004
3/4	0.003-0.006
1	0.004-0.008

SHOP NOTE: These are conservative general starting ranges, not tool-maker data. Slotting usually needs less feed than side milling; radial engagement changes ideal chip thinning.

10 DRILLING STARTING SPEEDS

Conservative starting SFM for conventional twist drilling; adjust for drill material, coating, coolant and rigidity.

Material	HSS drill SFM	Carbide drill SFM
Mild steel	70-100	180-300
4140 / alloy steel	40-70	120-220
304 / 316 stainless	25-45	80-160
Cast iron	50-80	150-250
Aluminum	200-350	500-1000
Brass	150-250	350-700

$$\text{RPM} = (\text{SFM} \times 3.82) / \text{Drill Diameter}$$

SHOP NOTE: Stainless work-hardens: keep the drill cutting, use a rigid setup, sharp geometry and appropriate feed. Pecking strategy depends on hole depth and chip evacuation.

11 MATERIAL MACHINING NOTES

Quick behavior cues when the chips, finish or tool life are telling you something.

Material	What to expect	Shop response
1018 / mild steel	Can tear and produce stringy chips.	Sharp positive geometry; enough feed to get under built-up edge.
1045	Stronger, generally machines cleaner than gummy low-carbon steel.	Moderate speed; rigid setup.
4140 prehard	Higher cutting forces and heat.	Carbide preferred; reduce SFM if tool life falls.
304 / 316 SS	Work-hardens; stringy; poor thermal conductivity.	Positive sharp tool, steady feed, coolant; avoid rubbing.
Cast iron	Abrasive; short chips; graphite dust.	Carbide; often machined dry; protect ways/electronics from dust.
6061 aluminum	Free machining; can build up on edge.	High positive rake; polished/sharp edge; good chip evacuation.
Brass	Free cutting; can grab with aggressive drill rake.	Use suitable geometry and secure workholding.

12 COMMON STEEL GRADES

A practical identification guide - not a substitute for an MTR or positive material identification.

Grade	Type	Useful shop description
1018	Low-carbon	Weldable, formable, common shaft/fixture stock.
A36	Structural carbon	General structural plate/shapes; chemistry/mechanical range differs from 1018.
1045	Medium-carbon	Higher strength/wear potential; shafts, pins.
4140	Cr-Mo alloy	Strong, tough; common in shafts, tooling and oilfield parts.
4140 PH	Prehardened 4140	Often supplied around high-20s/low-30s HRC range depending product.
4340	Ni-Cr-Mo alloy	High toughness/strength; critical highly loaded parts.
A2	Air-hardening tool steel	Good dimensional stability and wear.
D2	High-Cr tool steel	High wear resistance; tougher to machine after heat treat.
O1	Oil-hardening tool steel	Common shop tool steel; easier to machine annealed.

SHOP NOTE: Do not identify alloy by sparks alone for critical service. Use traceability, MTRs or PMI as required.

13 STAINLESS & ALUMINUM

Common grades and machining behavior.

Grade	Family	Quick note
304	Austenitic SS	General corrosion resistance; work-hardens.
316	Austenitic SS	Mo-bearing; improved chloride resistance vs 304; work-hardens.
17-4 PH	Precipitation-hardening SS	High strength with heat-treatment options; generally machines well in Condition A.
410	Martensitic SS	Heat treatable; magnetic; moderate corrosion resistance.
6061-T6	Aluminum	Excellent general-purpose machinability and availability.
7075-T6	Aluminum	High strength; good machining; lower corrosion/weld friendliness than 6061.
2024	Aluminum	High strength/fatigue performance; common aerospace alloy; corrosion protection often needed.

14 FITS & CLEARANCES

Practical fit categories for shop planning. Exact tolerances belong on the drawing or in the governing fit standard.

Fit	Typical feel / purpose	General intent
Loose clearance	Easy assembly, dirt/temperature tolerance	Large positive clearance
Running clearance	Rotating/sliding parts	Controlled positive clearance
Locational clearance	Accurate location, easy assembly	Small positive clearance
Transition	May assemble by hand or light press	Near-zero clearance to slight interference
Press / interference	Permanent or torque-transmitting fit	Negative clearance / interference

Clearance = Hole diameter - Shaft diameter

SHOP NOTE: For actual tolerances use the drawing or a recognized fit system (e.g., ANSI B4.1 / ISO 286). Material, diameter, wall thickness, temperature and assembly method matter.

15 KEYS & KEYWAYS

Common square-key proportions for quick planning; verify the drawing/standard before cutting.

Shaft dia. range (in)	Common key width x height (in)
1/2 - 9/16	1/8 x 1/8
5/8 - 7/8	3/16 x 3/16
15/16 - 1-1/4	1/4 x 1/4
1-5/16 - 1-3/8	5/16 x 5/16
1-7/16 - 1-3/4	3/8 x 3/8
1-13/16 - 2-1/4	1/2 x 1/2
2-5/16 - 2-3/4	5/8 x 5/8
2-13/16 - 3-1/4	3/4 x 3/4

SHOP NOTE: Keyway depth, corner radius and tolerances are application-specific. Do not infer them solely from key width.

16 SURFACE FINISH

Approximate process capability ranges. Actual finish depends heavily on machine condition, material, tool geometry and setup.

Process	Typical Ra range (microinch)
Sawing / rough flame cut	250-1000+
Rough turning / milling	125-500
Finish turning / milling	32-125
Grinding	8-63
Honing	4-32
Lapping / polishing	1-16

Ra is arithmetic average roughness. 1 micrometer Ra $\approx$ 39.37 microinch Ra.

SHOP NOTE: Do not equate a process name with a guaranteed finish. Measure when surface finish is a requirement.

17 TAPERS & ANGLES

Useful formulas for turning tapers and setting compound angles.

$$\text{Taper per foot} = ((D - d) / L) \times 12$$

$$\text{Half-angle } \theta = \text{atan}((D - d) / (2L))$$

Example	Value
Large dia. D	2.000 in
Small dia. d	1.500 in
Taper length L	6.000 in
Taper per foot	1.000 in/ft
Compound half-angle	2.386°

SHOP NOTE: Lathe compound is normally set to the half-angle when feeding the compound to generate the taper.

18 BOLT CIRCLE / COORDINATES

Coordinates for equally spaced holes on a bolt circle.

$$X = R \cos(\theta) \mid Y = R \sin(\theta) \mid \theta = 360^\circ \times \text{hole index} / \text{number of holes}$$

Example: 8 holes on a 6.000 in bolt circle

Hole	Angle	X	Y
1	0.0	3.0000	0.0000
2	45.0	2.1213	2.1213
3	90.0	0.0000	3.0000
4	135.0	-2.1213	2.1213
5	180.0	-3.0000	0.0000
6	225.0	-2.1213	-2.1213
7	270.0	-0.0000	-3.0000
8	315.0	2.1213	-2.1213

SHOP NOTE: Coordinates above assume the bolt circle center is X0 Y0 and Hole 1 starts at 0° on +X.

19 MACHINIST FORMULAS

Keep these near the machine.

Need	Formula
Turning / milling RPM	$RPM = (SFM \times 3.82) / \text{diameter (in)}$
Surface speed	$SFM = RPM \times \text{diameter} / 3.82$
Milling feed	$IPM = RPM \times \text{flutes} \times \text{chip load}$
Inch thread pitch	$Pitch = 1 / TPI$
Approx. inch tap drill	$\text{Major dia.} - (1 / TPI)$
Circle circumference	$C = \pi D$
Circle area	$A = \pi D^2 / 4$
Cylinder volume	$V = \pi D^2 L / 4$
Metric conversion	$mm = in \times 25.4; in = mm / 25.4$
Clearance	Hole dia. - shaft dia.
Taper per foot	$((D - d) / L) \times 12$
Bolt circle X/Y	$X = R \cos\theta; Y = R \sin\theta$

20 INSPECTION & SETUP CHECKLIST

A compact pre-cut / final-inspection routine for one-off and repair work.

Stage	Check
Before cutting	Confirm material / traceability if required; review drawing revision; identify datums; verify stock allowance; inspect workholding; confirm tool clearance.
Before threading	Confirm thread system, diameter, pitch/TPI, class/gauge requirement, chamfer, tap/drill size and thread direction.
Before boring	Measure current bore in multiple locations; check taper/out-of-round; confirm final size and finish; account for thermal condition.
Before final pass	Recheck tool offset; leave measurable stock; confirm measuring instrument resolution and calibration status.
Final inspection	Size, location, runout/concentricity as required, thread gauge, surface condition, burr removal, cleanliness, documentation.
Repair work	Record as-found dimensions before removing evidence; photograph damage; preserve customer markings/traceability; document deviations and app

REFERENCE BASIS / LIMITS

This guide is intentionally a quick shop reference. Thread forms and pipe dimensions should be verified against the applicable current ASME/ANSI standard and the tool or fastener manufacturer's data. Cutting speeds, feeds, finishes and fit descriptions are starting guidance only. Critical dimensions, pressure-retaining parts, customer specifications and safety-related work require the governing drawing, procedure, code and inspection plan.

Common standards to consult: ASME B1.1 (Unified inch threads), ASME B1.20.1 (NPT), ASME B36.10/B36.19 (pipe dimensions), ANSI B4.1 / ISO 286 (fits), and applicable manufacturer cutting-data catalogs.