Thread Tap Drill Chart

Common cutting-tap starting drills for metric, UNC, and UNF internal threads, with practical checks for sheet metal drawings.

Important: These values are common cutting-tap starting sizes, not universal limits or guaranteed finished minor diameters. Verify the selected tap manufacturer data, thread class, material, hole-making process, coating sequence, and inspection method. Forming taps require a different, normally larger pre-hole.

Metric Coarse Threads

For ISO-style metric threads, **M** is the nominal major diameter in millimeters and the value after x is pitch P in millimeters. The listed drill is a common nominal cutting-tap size.

METRIC THREAD	PITCH P (MM)	COMMON CUTTING-TAP DRILL (MM)	QUICK CHECK: D - P (MM)
M2 × 0.4	0.40	1.60	1.60
M2.5 × 0.45	0.45	2.05	2.05
M3 × 0.5	0.50	2.50	2.50
M3.5 × 0.6	0.60	2.90	2.90
M4 × 0.7	0.70	3.30	3.30
M5 × 0.8	0.80	4.20	4.20
M6 × 1.0	1.00	5.00	5.00
M8 × 1.25	1.25	6.80	6.75
M10 × 1.5	1.50	8.50	8.50
M12 × 1.75	1.75	10.20	10.25
M14 × 2.0	2.00	12.00	12.00
M16 × 2.0	2.00	14.00	14.00
M18 × 2.5	2.50	15.50	15.50
M20 × 2.5	2.50	17.50	17.50

Reference: D - P is an arithmetic check, not a tolerance specification. When the calculation is not a preferred drill diameter, select a nearby common drill. Specify the thread and acceptance criteria when a controlled internal minor diameter is required.

Metric Fine Threads

Fine-pitch threads can preserve more wall thickness around a hole and may suit limited engagement length. Confirm that the mating fastener and tap use the same pitch.

METRIC THREAD	PITCH P (MM)	COMMON CUTTING-TAP DRILL (MM)	QUICK CHECK: D - P (MM)
M3 × 0.35	0.35	2.65	2.65
M4 × 0.5	0.50	3.50	3.50
M5 × 0.5	0.50	4.50	4.50
M6 × 0.75	0.75	5.25	5.25
M8 × 1.0	1.00	7.00	7.00
M10 × 1.25	1.25	8.75	8.75
M12 × 1.5	1.50	10.50	10.50
M14 × 1.5	1.50	12.50	12.50
M16 × 1.5	1.50	14.50	14.50
M18 × 1.5	1.50	16.50	16.50
M20 × 1.5	1.50	18.50	18.50

Design control: The thread designation, class or tolerance position, and gauge requirement remain the controlling product definition.

UNC and UNF Threads

For Unified inch threads, the number after the dash is threads per inch (TPI). The inch drill designation and decimal-inch diameter are controlling; metric equivalents are rounded only for tool-room comparison.

THREAD	SERIES	COMMON DRILL	DRILL DIAMETER (IN)	METRIC EQUIVALENT (MM)
#2-56	UNC	#50	0.0700	1.78
#2-64	UNF	#50	0.0700	1.78
#4-40	UNC	#43	0.0890	2.26
#4-48	UNF	#42	0.0935	2.37
#6-32	UNC	#36	0.1065	2.71
#6-40	UNF	#33	0.1130	2.87
#8-32	UNC	#29	0.1360	3.45
#8-36	UNF	#29	0.1360	3.45
#10-24	UNC	#25	0.1495	3.80
#10-32	UNF	#21	0.1590	4.04
1/4-20	UNC	#7	0.2010	5.11
1/4-28	UNF	#3	0.2130	5.41
5/16-18	UNC	F	0.2570	6.53
5/16-24	UNF	I	0.2720	6.91
3/8-16	UNC	5/16 in	0.3125	7.94
3/8-24	UNF	Q	0.3320	8.43
7/16-14	UNC	U	0.3680	9.35
7/16-20	UNF	25/64 in	0.3906	9.92
1/2-13	UNC	27/64 in	0.4219	10.72
1/2-20	UNF	29/64 in	0.4531	11.51
9/16-12	UNC	31/64 in	0.4844	12.30
9/16-18	UNF	33/64 in	0.5156	13.10
5/8-11	UNC	17/32 in	0.5313	13.49
5/8-18	UNF	37/64 in	0.5781	14.68
3/4-10	UNC	21/32 in	0.6563	16.67
3/4-16	UNF	11/16 in	0.6875	17.46
7/8-9	UNC	49/64 in	0.7656	19.45
7/8-14	UNF	13/16 in	0.8125	20.64
1-8	UNC	7/8 in	0.8750	22.23
1-12	UNF	59/64 in	0.9219	23.42

Reference: ASME B1.1 defines Unified thread form, series, class, allowance, tolerance, and designation. Check common drills against the tap supplier's calculator or catalog when tool life, thread class, difficult material, or a gauged production process matters.

Quick Cutting-Tap Drill Check

THREAD SYSTEM	SHOP-FLOOR APPROXIMATION	INTERPRETATION
Metric	$D_d \sim= D - P$	D_d is estimated pre-hole diameter; D is nominal major diameter and P is pitch in millimeters.
Unified inch	$D_d \sim= D - (1 / N)$	N is threads per inch. The calculation is a reasonableness check, not a complete thread-design calculation.

Examples: M5 × 0.8: 5.0 - 0.8 = 4.2 mm. 1/4-20 UNC: 0.250 - 1/20 = 0.200 in, close to a #7 drill at 0.2010 in. These approximations do not establish thread class, allowance, drill tolerance, material response, or the tap manufacturer's recommendation.

Cutting vs. Forming Tap Example

A cutting tap removes material; a forming or fluteless tap displaces ductile material and starts from a larger hole. Confirm exact form-tap holes directly from the tool supplier and material application data.

M5 X 0.8 EXAMPLE	CALCULATOR SETTING	CALCULATED PRE-HOLE (MM)
Cutting tap	75% thread	4.221
Forming tap	75% thread	4.592

Note: Calculator outputs are shown to 0.001 mm because that is how the source reports them; they are not a ShincoFab process tolerance. Form-tap recommendations change with material ductility, tap limit, lubrication, and hole tolerance.

Drawing Release Checks for Sheet Metal Parts

CHECK	WHAT TO CONFIRM BEFORE RELEASE
Thread callout	Confirm metric versus Unified, coarse versus fine, hand, and the required class or tolerance position.
Thread-making method	Select cutting tap, forming tap, thread mill, extruded-and-tapped hole, weld nut, rivet nut, or self-clinching hardware before freezing the pre-hole.
Usable engagement	Check sheet thickness and tap lead. Change the joint when flat sheet cannot provide the required engagement.
Hole-making process	Account for the final pre-hole created by laser cutting, punching, drilling, or reaming, including taper, burrs, roundness, and size control.
Access and geometry	Verify tap, holder, and gauge access; check nearby edges, bends, welds, and distortion risk.
Finish sequence	Define whether threads are produced before or after plating, anodizing, paint, or powder coating, including masking or restoration allowances.
Inspection	Specify the thread gauge, class, inspection stage, sampling plan, and any needed functional fastener test.

Common Problems and Corrective Options

SYMPTOM OR CONSTRAINT	CORRECTIVE OPTION	TRADE-OFF OR CHECK
Tap torque is high or taps chip early	Evaluate a slightly larger pre-hole or lower thread percentage using supplier data.	Confirm thread strength, minor diameter, and gauge acceptance before release.
Thread strips in thin sheet	Add an extrusion, increase local thickness, or use inserted hardware.	Changes tooling, access, appearance, and sometimes finish sequence.
Form tap is specified with a cutting-tap hole	Replace the hole with the exact form-tap recommendation.	Material ductility, tap limit, lubrication, and hole tolerance become critical.
Coating fills or tightens the thread	Mask the thread, tap after finish, or define an approved restoration step.	Each option changes corrosion protection, cosmetics, handling, or inspection.
Hole is too close to a bend or edge	Move the hole, change the bend sequence, or use separate hardware.	Requires a drawing-level check of access, distortion, and joint load path.
Tapped joint is difficult to service	Use a rivet nut, weld nut, or self-clinching nut.	Adds component cost and installation controls but can provide a replaceable or deeper thread.