MECHANICAL · CHEAT SHEET

# Tap drill *chart*.

Tap drill and clearance hole sizes for every common screw thread — imperial UN (UNC + UNF) and metric M-thread. Sized for **75% thread engagement**, the standard for general-purpose work.

## The chart

| THREAD | SYSTEM | TAP DRILL | TAP DRILL (DECIMAL) | CLEARANCE DRILL |
|---|---|---|---|---|
| **#0-80** | UN imperial | 3/64 | 0.0469″ | #52 |
| **#1-64** | UN imperial | #53 | 0.0595″ | #48 |
| **#1-72** | UN imperial | #53 | 0.0595″ | #48 |
| **#2-56** | UN imperial | #50 | 0.0700″ | #43 |
| **#2-64** | UN imperial | #50 | 0.0700″ | #43 |
| **#4-40** | UN imperial | #43 | 0.0890″ | #32 |
| **#4-48** | UN imperial | #42 | 0.0935″ | #32 |
| **#6-32** | UN imperial | #36 | 0.1065″ | #27 |
| **#6-40** | UN imperial | #33 | 0.1130″ | #27 |
| **#8-32** | UN imperial | #29 | 0.1360″ | #18 |
| **#8-36** | UN imperial | #29 | 0.1360″ | #18 |

| THREAD | SYSTEM | TAP DRILL | TAP DRILL (DECIMAL) | CLEARANCE DRILL |
|---|---|---|---|---|
| **#10-24** | UN imperial | #25 | 0.1495″ | #9 |
| **#10-32** | UN imperial | #21 | 0.1590″ | #9 |
| **1/4-20** | UN imperial | #7 | 0.2010″ | 17/64 |
| **1/4-28** | UN imperial | #3 | 0.2130″ | 17/64 |
| **5/16-18** | UN imperial | F | 0.2570″ | 21/64 |
| **5/16-24** | UN imperial | I | 0.2720″ | 21/64 |
| **3/8-16** | UN imperial | 5/16 | 0.3125″ | 25/64 |
| **3/8-24** | UN imperial | Q | 0.3320″ | 25/64 |
| **7/16-14** | UN imperial | U | 0.3680″ | 29/64 |
| **7/16-20** | UN imperial | 25/64 | 0.3906″ | 29/64 |
| **1/2-13** | UN imperial | 27/64 | 0.4219″ | 33/64 |
| **1/2-20** | UN imperial | 29/64 | 0.4531″ | 33/64 |
| **5/8-11** | UN imperial | 17/32 | 0.5313″ | 41/64 |
| **5/8-18** | UN imperial | 37/64 | 0.5781″ | 41/64 |
| **3/4-10** | UN imperial | 21/32 | 0.6562″ | 49/64 |
| **3/4-16** | UN imperial | 11/16 | 0.6875″ | 49/64 |
| **1-8** | UN imperial | 7/8 | 0.8750″ | 1-1/64 |
| **1-12** | UN imperial | 59/64 | 0.9219″ | 1-1/64 |
| **M2 × 0.4** | metric | 1.6 mm | – | 2.4 mm |

| THREAD | SYSTEM | TAP DRILL | TAP DRILL (DECIMAL) | CLEARANCE DRILL |
|---|---|---|---|---|
| **M2.5 × 0.45** | metric | 2.05 mm | – | 2.9 mm |
| **M3 × 0.5** | metric | 2.5 mm | – | 3.4 mm |
| **M3.5 × 0.6** | metric | 2.9 mm | – | 3.9 mm |
| **M4 × 0.7** | metric | 3.3 mm | – | 4.5 mm |
| **M5 × 0.8** | metric | 4.2 mm | – | 5.5 mm |
| **M6 × 1.0** | metric | 5.0 mm | – | 6.6 mm |
| **M8 × 1.25** | metric | 6.8 mm | – | 9.0 mm |
| **M8 × 1.0** | metric | 7.0 mm | – | 9.0 mm |
| **M10 × 1.5** | metric | 8.5 mm | – | 11.0 mm |
| **M10 × 1.25** | metric | 8.8 mm | – | 11.0 mm |
| **M12 × 1.75** | metric | 10.2 mm | – | 13.5 mm |
| **M12 × 1.25** | metric | 10.8 mm | – | 13.5 mm |
| **M14 × 2.0** | metric | 12.0 mm | – | 15.5 mm |
| **M16 × 2.0** | metric | 14.0 mm | – | 17.5 mm |
| **M20 × 2.5** | metric | 17.5 mm | – | 22.0 mm |

**About thread engagement.** 75% engagement is the standard compromise between thread strength and tapping effort. Higher engagement (90%) needs a smaller tap drill, more torque to tap, and only adds about 5% thread strength. Lower (50%) is faster to tap and used in soft materials, but the threads are weaker.

# Common applications

| USE CASE | RECOMMENDATION | WHY |
| --- | --- | --- |
| Tapped hole in aluminum | 75% engagement | Standard. Use cutting fluid. |
| Tapped hole in cast iron | 75% engagement | Standard. Dry tap or with light oil. |
| Tapped hole in mild steel | 75% engagement | Standard. Use cutting fluid. |
| Tapped hole in stainless steel | 65% engagement | Reduce tap stress; use sulfur-bearing fluid. |
| Tapped hole in brass | 75% engagement | Use a 'gun tap' or cut tap with no rake. |
| Tapped hole in plastic | 50% engagement | Plastic deforms; larger tap drill prevents thread tearout. |
| Tapped hole in titanium | 65% engagement | High strength; reduce engagement to ease tapping. |

# Common pitfalls

- **75% engagement is not 75% of the OD.** It's 75% of the thread depth, which means a smaller difference. For M6 × 1.0: nominal OD 6.0 mm, tap drill 5.0 mm = 17% smaller hole.
- **Hand tap vs machine tap differs.** Hand taps use a 75% spec. CNC machine taps with chip evacuation may use 50–65% in production.
- **Through-holes vs blind holes.** In a blind hole, leave enough depth for chip clearance below the threaded portion — typically 2–3× the thread pitch.
- **Tapered taps vs bottoming taps.** A tapered tap has 8–10 chamfered threads (easier to start). A bottoming tap has 1–2 (cuts threads closer to the bottom). Don't tap a blind hole with a tapered tap and expect full-

depth threads.

- **Clearance drill ≠ tap drill.** A 'clearance hole' is for a bolt to pass through freely. A 'tap drill' is for cutting threads. They're not the same hole.

## Common questions

### What's the right tap drill for 1/4-20?

#7 drill (0.201") for 75% thread engagement — the standard. For aluminum or soft materials, 1/4-20 can also use #8 (0.199") for slightly more thread. For high-pressure or repeated assembly, #7 is the right choice. Going larger (e.g. 13/64") reduces thread strength; going smaller (e.g. #9) increases tapping torque significantly.

### Why not just drill the tap to bolt diameter?

Tap-drill size = bolt diameter minus 1/(threads per inch) — leaving material for the thread crests to form. A 1/4" tap drill would leave no material for threads; the bolt would just spin freely. The 75% thread rule balances strength (enough thread engagement) against tapping torque (not too much material to cut).

### What's the difference between 'tap drill' and 'clearance drill'?

Tap drill creates the hole that gets tapped (threaded). Clearance drill creates the hole in the mating part that the bolt passes through without threading. For 1/4-20: tap drill = #7 (0.201"), clearance drill = 17/64" (0.266") for close fit or 9/32" (0.281") for normal fit. Mixing these up means bolts won't pass through.

### Can I tap aluminum with the same drill as steel?

Aluminum threads tear out more easily than steel, so a slightly larger tap drill (less thread engagement) can be safer — fewer broken taps, but slightly weaker threads. For critical applications, use Heli-Coils or thread inserts in aluminum to get steel-strength threads. Standard tap drill works for most uses; just use cutting fluid and slower speeds.

### How deep should a tapped hole be relative to bolt diameter?

Minimum 1× bolt diameter for steel-in-steel (1/4-20 bolt → 1/4" minimum thread depth). For aluminum or brass, 1.5-2× diameter. For maximum strength, 3× diameter — but you don't gain meaningful strength past 1.5-2× because the bolt fails before the threads strip. Account for chip clearance below the threads if using a bottoming tap.

## Sources

- **Imperial:** Machinery's Handbook 31st ed., Table 3 — Tap Drill Sizes for 75% Thread.
- **Metric:** ISO 2306 — Drills for Use Prior to Tapping Screw Threads.
- **Formula (imperial):** tap drill = nominal OD − (1.0825 ÷ TPI) for 75% engagement.
- **Formula (metric):** tap drill = nominal OD − pitch (for 75% engagement, approximate).

**Disclaimer.** Tap drill sizes shown produce ~75% engagement in general-purpose applications. Material, thread length, and required strength can shift the right answer.