Hole-to-Bend Distance Chart

First-pass reference values for keeping round holes clear of the sheet metal bend zone. Start with material thickness and the planned inside bend radius, then confirm the distance convention with the fabricator.

Hole-to-Bend Distance Formula

D = 2.5T + R

D = distance from the flat-pattern bend reference line to the nearest edge of the round hole | T = sheet thickness | R = planned inside bend radius after forming

For a drawing that locates the hole by its center, add half the hole diameter: Dcenter = 2.5T + R + d/2, where d = hole diameter. The bend reference line is the bend-location line in the flat pattern, commonly the bend centerline. The selected R should agree with the material and production tooling.

Quick Distance Multipliers

The formula expressed as a multiple of thickness when the inside radius is known as an R/T ratio.

INSIDE RADIUS	CALCULATION	HOLE-EDGE DISTANCE D
R = 0.5T	2.5T + 0.5T	3.0T
R = 1.0T	2.5T + 1.0T	3.5T
R = 1.5T	2.5T + 1.5T	4.0T
R = 2.0T	2.5T + 2.0T	4.5T
R = 3.0T	2.5T + 3.0T	5.5T

Note: A larger bend radius increases the bend-line-to-hole distance under this convention, even though another supplier may state the clear hole-edge-to-tangent distance separately.

Hole-to-Bend Distance Chart (R = T)

Assumes a 90-degree bend with an inside radius equal to material thickness, so D = 3.5T. Useful for layout and early quoting; the actual inside radius and tooling should control the released design. Metric values control this table. Inch equivalents are converted from millimeters using 1 in = 25.4 mm and rounded to the nearest 0.001 in.

SHEET THICKNESS T	ASSUMED INSIDE RADIUS R	HOLE-EDGE DISTANCE D	INCH EQUIVALENT OF D
0.50 mm	0.50 mm	1.75 mm	0.069 in
0.80 mm	0.80 mm	2.80 mm	0.110 in

SHEET THICKNESS T	ASSUMED INSIDE RADIUS R	HOLE-EDGE DISTANCE D	INCH EQUIVALENT OF D
1.00 mm	1.00 mm	3.50 mm	0.138 in
1.20 mm	1.20 mm	4.20 mm	0.165 in
1.50 mm	1.50 mm	5.25 mm	0.207 in
2.00 mm	2.00 mm	7.00 mm	0.276 in
2.50 mm	2.50 mm	8.75 mm	0.344 in
3.00 mm	3.00 mm	10.50 mm	0.413 in
4.00 mm	4.00 mm	14.00 mm	0.551 in
5.00 mm	5.00 mm	17.50 mm	0.689 in
6.00 mm	6.00 mm	21.00 mm	0.827 in

Why it matters: Material on the outside of a bend stretches while material toward the inside compresses. A hole entering this strained region can become oval or teardrop-shaped and shift relative to the formed flange, even when it still looks acceptable. Inch equivalents are rounded conversions, not additional tolerance requirements.

Worked Example

Example: For T = 1.5 mm, R = 2.0 mm, and round-hole diameter d = 6.0 mm, the distance to the nearest hole edge is D = 2.5(1.5) + 2.0 = 5.75 mm. If the drawing dimensions to the hole center, Dcenter = 5.75 + 6.0/2 = 8.75 mm. On an inch drawing these become 0.226 in and 0.344 in, rounded to the nearest 0.001 in from the controlling metric calculations.

How to Use This Chart

STEP	GUIDELINE
1	Confirm the specified sheet thickness rather than relying only on gauge. Gauge-to-thickness relationships vary by material.
2	Confirm the planned inside bend radius from the material, grain direction, bend angle, tooling, and forming method.
3	Calculate D = 2.5T + R for a round through-hole made before bending.
4	Measure D to the nearest hole edge. If the drawing locates the center, add d/2.
5	Label the bend reference on the drawing. If the supplier uses a tangent or outside-corner convention, reconcile the two origins before releasing the flat pattern.
6	Review slots, countersinks, extruded holes, louvers, hardware holes, and large cutouts separately; the round-hole formula may not represent their deformation risk.
7	Ask the fabricator to confirm the value when the hole is function-critical, carries a tight position tolerance, or sits near another bend, edge, relief, weld, or inserted fastener.

Why Hole-to-Bend Rules Differ

Published values look contradictory because they describe different processes or measure from different locations. Compare both the number and its definition: a hole-edge-to-tangent distance cannot be substituted directly for a hole-center-to-bend-line or outside-corner distance. The references below are not numerically interchangeable without checking the original diagram or drawing convention.

REFERENCE CONTEXT	PUBLISHED STARTING GUIDANCE	MEASUREMENT OR SCOPE NOTE
General sheet metal design guide	Round holes: 2.5T + R	Geomiq describes distance away from the bend; this page applies that value to its explicitly defined bend-line-to-hole-edge convention.
General sheet metal design guide	Slots: 4T + R	A slot has a longer unsupported edge and may distort more readily; orientation and slot size still require review.
Progressive-die or metal stamping guidance	2T + R for holes or slots below 0.100 in; 2.5T + R for larger features	Xometry presents these as stamping recommendations, not universal press-brake requirements.
Mild-steel job-shop press-brake experience	Keep the hole edge at least 2T from the start of the bend radius	The Fabricator states this clear tangent-based distance for mild steel about 0.02–0.13 in thick.
Supplier-specific bend table	Material- and thickness-specific minimum-hole values	Protocase publishes values tied to its stocked materials and its outside-corner convention; supplier tables should control when that supplier is selected.

What to Do When the Hole Is Too Close to the Bend

Options for a hole that would otherwise sit in the bend zone, from repositioning the feature to creating it after forming.

OPTION	WHEN IT HELPS	TRADE-OFF
Move the hole farther from the bend	The surrounding envelope and mating part have available space.	May require a mating-part or bracket-layout change.
Move the bend or lengthen the flange	The hole location is functionally fixed but the formed envelope can change.	Can affect overall size, stiffness, material use, and tool access.
Create the hole after forming	Final hole position relative to formed datums is critical and machining access is available.	Adds setup, fixturing, cycle time, and cost.
Change the feature into a reviewed relief or open-ended notch	The feature can intersect the bend region without carrying a fastener or sealing function.	Removes material and may reduce strength, appearance, or edge protection.
Use specialized forming or localized restraint	Production volume or part value supports dedicated development.	Requires supplier-specific tooling, trials, and documented acceptance criteria.
Relax or redefine the positional requirement	The current tolerance is tighter than the assembly needs.	Requires engineering approval and may not solve visible distortion.

OPTION	WHEN IT HELPS	TRADE-OFF
Cut a pilot feature, then finish after bending	Accurate final location is needed but full post-form material removal is undesirable.	Adds operations and requires reliable datum transfer.

Reference values for early DFM review of round through-holes cut or punched before forming — not guaranteed production values. Material grade and temper, hole size, bend angle, inside radius, grain direction, tooling, forming method, and the drawing's measurement origin can all change the distance needed for an acceptable part.