Sheet Metal Bend Allowance and Bend Deduction Chart

Estimate a flat pattern from finished outside dimensions, or understand how a CAD system compensates for material in a bend. The formulas and tables below make the bend-angle convention, inside radius, thickness, and K-factor explicit.

Important: This chart is an early DFM reference for a single bend. It assumes a constant inside radius and K-factor through the bend. Actual values depend on material grade and temper, thickness, rolling direction, bend angle, tooling, forming method, and the press-brake setup. Confirm production values with the fabricator or a measured trial bend before releasing critical flat dimensions.

Bend Allowance and Bend Deduction Formulas

The bend angle A is the angle through which the flat sheet is bent: 0° is flat and 90° is a right-angle bend. L1 and L2 are finished outside flange dimensions measured to the virtual sharp intersection. Use one unit system throughout a calculation, keeping R, T, BA, BD, OSSB, L1, and L2 in the same unit.

$$BA = (\pi / 180) \times A \times (R + K \times T)$$

$$OSSB = \tan(A / 2) \times (R + T)$$

$$BD = 2 \times OSSB - BA$$

$$\text{Flat length} = L1 + L2 - BD$$

SYMBOL	MEANING
BA	Bend allowance: neutral-axis arc length through the bend
BD	Bend deduction subtracted from the sum of the two outside flange dimensions
OSSB	Outside setback: distance from the bend tangent to the virtual sharp
A	Bend angle in degrees (0° flat, 90° right-angle bend, per the convention above)
R	Formed inside bend radius
K	K-factor: neutral-axis location from the inside surface divided by T
T	Material thickness

90° Bend Allowance and Bend Deduction Chart

Calculated for one 90° bend with R = T and K = 0.33, giving BA = 2.089T and BD = 1.911T. T is material thickness and R is the inside radius (R = T, both mm); BA is the bend allowance and BD the bend deduction, listed in millimetres and inches. Inch values are the millimetre results divided by 25.4 and rounded to four decimal places. Metric values control.

T (MM)	R (MM)	BA (MM)	BD (MM)	BA (IN)	BD (IN)
0.5	0.5	1.045	0.955	0.0411	0.0376
0.8	0.8	1.671	1.529	0.0658	0.0602
1.0	1.0	2.089	1.911	0.0823	0.0752
1.2	1.2	2.507	2.293	0.0987	0.0903
1.5	1.5	3.134	2.866	0.1234	0.1128
2.0	2.0	4.178	3.822	0.1645	0.1505
2.5	2.5	5.223	4.777	0.2056	0.1881
3.0	3.0	6.267	5.733	0.2468	0.2257
4.0	4.0	8.357	7.643	0.3290	0.3009
5.0	5.0	10.446	9.554	0.4113	0.3761
6.0	6.0	12.535	11.465	0.4935	0.4514

Do not select a production bend deduction from thickness alone. Recalculate the row if the actual radius or K-factor differs, or use a validated bend table. Check the proposed inside radius against the material-specific minimum bend radius chart before treating R = T as feasible.

Bend-Angle Multipliers for R = T and K = 0.33

Estimate BA and BD for other bend angles by multiplying the dimensionless coefficient by material thickness T. These are calculated values for R/T = 1.00 and K = 0.33, not measured shop data.

BEND ANGLE A	BA / T	BD / T
30 degrees	0.696	0.375
45 degrees	1.045	0.612
60 degrees	1.393	0.917
90 degrees	2.089	1.911
120 degrees	2.786	4.143
135 degrees	3.134	6.523

Large-angle bend deductions grow quickly because the virtual-sharp intersection moves farther from the tangent points. Hems, curls, closed bends, and geometries approaching 180° need process-specific rules rather than extrapolation from this table.

How to Use This Bend Deduction Chart

STEP	ACTION
1	Confirm the finished part's material specification, actual thickness, bend angle, intended inside radius, and forming process.
2	Use the chart only when the bend is 90°, R = T, and the validated K-factor is 0.33. Otherwise, use the formulas.
3	Measure both outside flange dimensions to the same virtual-sharp convention used by the formula.
4	Subtract one BD for each bend from the sum of the corresponding finished outside dimensions.
5	For multiple bends, calculate each bend separately. Do not reuse one value when radius, angle, material orientation, or tooling changes.
6	Compare the result with the CAD flat pattern and the fabricator's bend table before cutting production blanks.

Worked example — 1.5 mm sheet, 90° bend: T = 1.5 mm, R = 1.5 mm, K = 0.33, A = 90°, finished outside flanges of 40 mm and 60 mm.
BA = (π / 180) × 90 × (1.5 + 0.33 × 1.5) = 3.134 mm
OSSB = tan(45°) × (1.5 + 1.5) = 3.000 mm
BD = 2 × 3.000 - 3.134 = 2.866 mm
Flat length = 40 + 60 - 2.866 = 97.134 mm
This matches the 1.5 mm row after rounding. It remains a first-pass estimate until the radius and K-factor are confirmed for the production setup.

How K-Factor Changes Bend Allowance

The K-factor locates the neutral axis within the sheet thickness. A larger K increases bend allowance and therefore reduces bend deduction when angle, radius, and thickness stay constant. Sensitivity for a 90° bend with R = T:

K-FACTOR	BA / T	BD / T	INTERPRETATION
0.30	2.042	1.958	Smaller neutral-axis offset; shorter bend allowance
0.33	2.089	1.911	Assumption used in this page's quick charts
0.40	2.199	1.801	Larger neutral-axis offset; longer bend allowance
0.45	2.278	1.722	Sensitivity case, not a material recommendation
0.50	2.356	1.644	Mid-thickness neutral-axis limit in the geometric model

This table shows mathematical sensitivity only. It does not assign K-factor values to aluminum, steel, or stainless steel. Production K-factor should come from a supplier-specific bend table or from test coupons formed with the intended material, grain direction, radius, die opening, punch, and method.

How to Determine K-Factor From a Trial Bend

A trial bend is the most defensible way to connect a CAD flat pattern with a specific process setup.

STEP	ACTION
1	Record the coupon's material grade and temper, thickness, rolling direction, punch and die, programmed angle, and forming method.
2	Measure the original flat length and the two finished straight flange lengths using a documented tangent or virtual-sharp convention.
3	Derive the measured bend allowance as BA = flat length - straight length 1 - straight length 2 when the straight lengths are measured to the bend tangency points.
4	Rearrange the bend allowance formula: $K = \{[(BA \times 180) / (\pi \times A)] - R\} / T$.
5	Repeat the bend and measurement. Use the mean only if the samples and measurement method are stable.
6	Store the resulting value in a bend table tied to the exact material and tooling conditions.

> If the flange measurements are taken to a virtual sharp instead of tangent points, derive BD first and convert it using $BA = 2 \times OSSB - BD$. Mixing tangent and virtual-sharp measurements is a common source of incorrect K-factor results.

Bend Allowance, Bend Deduction, and K-Factor Compared

TERM	WHAT IT REPRESENTS	WHERE IT IS USED	MAIN INPUT RISK
Bend allowance	Neutral-axis arc length through the bend	Building the flat from tangent-length dimensions	Wrong K-factor or formed radius
Bend deduction	Amount removed from summed outside dimensions	Building the flat from outside flange dimensions	Ambiguous virtual-sharp measurement
K-factor	Neutral-axis offset divided by thickness	Calculating bend allowance	Treating a setup-dependent value as universal
Outside setback	Tangent-to-virtual-sharp distance	Converting between BA and BD	Using the wrong angle convention

> Bend allowance and bend deduction are two routes to the same developed length when dimensions and conventions are consistent. Do not add bend allowance and subtract bend deduction in the same bend calculation.

Design Checks Before Releasing a Drawing

Specify material grade, alloy, temper, and nominal thickness rather than relying only on gauge; translate legacy gauge callouts with the sheet metal gauge thickness chart.
Define the finished inside radius and bend angle, and state whether the angle is the included angle or the bend rotation.
Identify critical finished dimensions, datums, and whether dimensions apply before or after coating.

Confirm minimum flange support against the intended tooling; review the minimum flange length chart for early layout checks.
Check holes, slots, hardware, embosses, and cosmetic surfaces near the bend for distortion or tool contact.
Mark rolling direction where cracking, springback, or appearance makes orientation important.
Ask which bend table or K-factor the fabricator will use before sending a manufacturing flat pattern.
Keep the formed model as the design authority unless the fabricator explicitly requests a controlled flat pattern.

When the Flat Pattern Does Not Match

The formulas model the neutral-axis arc, but the actual bend comes from a material-tooling system: air bending can produce a radius influenced by the die opening and material response, and bottoming, coining, wipe forming, and hemming impose different contact conditions. Springback, thickness tolerance, tensile properties, grain direction, punch radius, die wear, lubrication, and measurement technique can all shift the finished geometry. The error usually appears as flange-length drift, hole-position error after forming, cumulative mismatch across multiple bends, or poor assembly fit.

OPTION	WHEN IT HELPS	TRADE-OFF
Verify angle and dimension conventions	CAD and manual results disagree by a large amount	Requires redimensioning to a common basis
Measure the formed inside radius	The assumed radius came from the drawing, not the part	Radius measurement can be method-sensitive
Run material-and-tooling coupons	Repeat production justifies calibrated data	Adds setup and inspection work
Adjust the K-factor or bend table	Error is repeatable for one controlled setup	The correction may not transfer to another material or tool
Change the radius or die opening	Cracking, marking, or flange support is unacceptable	Changes BA, BD, springback, and possibly the finished design
Dimension the formed part and let the shop unfold it	Supplier controls cutting and bending	Requires clear agreement on design authority and revision control

Tight finished dimensions need a drawing-specific tolerance strategy rather than extra decimal places in a generic chart. For production parts, have the bend sequence, tooling access, radius, flange support, and flat-pattern assumptions reviewed together.