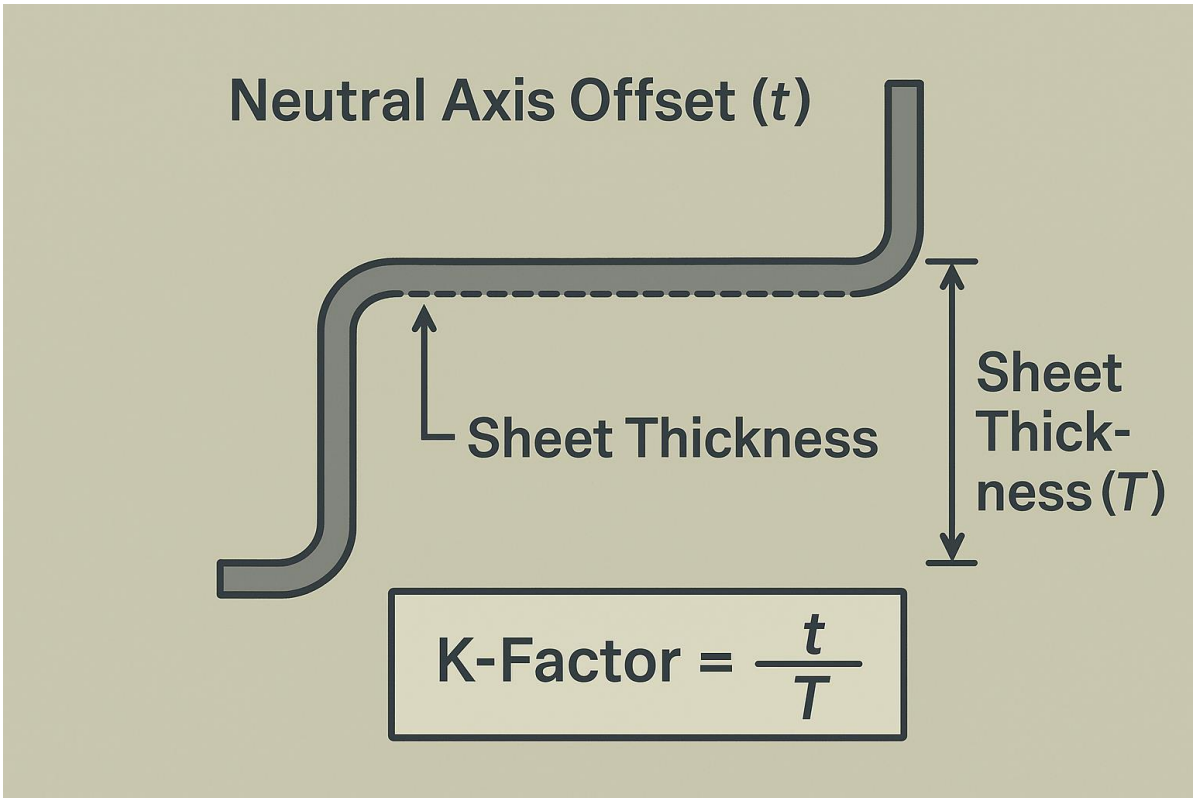
inductaflex

Understanding K–Factor and Bend Allowance in Sheet Metal Bending



In sheet metal bending, accuracy goes beyond applying force. Before the part reaches the press brake, calculations must be in place. If the flat blank is wrong, the final bend won't line up. This is where the K-Factor and bend allowance play a key role. These values help define how much the material will stretch and how to lay out the flat pattern correctly.

What Is the K-Factor?

The K-Factor explains the internal shift that happens in a bend. As the metal forms, the inside compresses while the outside stretches. The neutral axis sits between those two zones and doesn't stretch or compress. It moves depending on conditions.

The K-Factor equals the distance from the inside surface to the neutral axis (t) divided by the total material thickness (T):

$$K = \frac{t}{T}$$

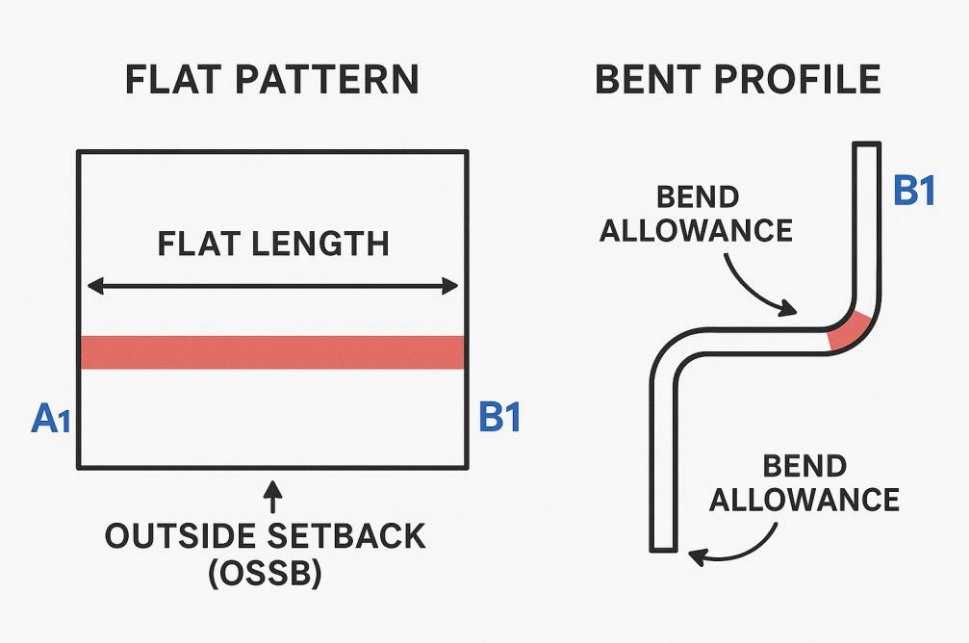
This ratio helps estimate how much elongation occurs in the bend.

For aluminum, this value often falls between 0.30 and 0.50, depending on factors like material properties, bend radius, and forming method. While K-Factors between 0.30 and 0.50 are possible, most practical aluminum bending jobs, especially those using air bending, fall between 0.33 and 0.45. Values near 0.50 generally indicate large bend radii or soft tempers. For example:

- In air bending, the K-Factor often ranges from 0.33 to 0.45
- In bottoming or coining, it tends to stay closer to 0.30

This number varies with setup. Bending method, angle, tooling, and alloy all influence it. It's best to test with your actual setup when accuracy is critical.

What Is Bend Allowance (BA)?



When creating a flat layout, you need to include the extra length taken up by the bend. That's the bend allowance (BA): the arc length along the neutral axis in the bent section. Leaving it out leads to undersized parts.

Here's how to calculate it using degreess:

$$BA = \frac{\pi}{180} x\ A\ x\ (R + K\ x\ T)$$

Where:

- A is the bend angle in degrees
- R is the inside bend radius
- T is material thickness
- K is the K-Factor

This formula works for angles in degrees. When working in radians, remove the

Π/180 multiplier. The formula assumes a consistent material thickness, no thinning during bending, and that the bend angle is the included angle, not the complementary angle.

Adding the correct bend allowance helps you achieve the intended shape after forming. Bend allowance is used to calculate the flat length by combining it with the lengths of adjacent flanges. It's not added as a separate extension but rather part of the total developed length.

Bend Deduction vs. Bend Allowance

There are two ways to approach material compensation in the bend zone: bend allowance and bend deduction (BD). They both adjust for material movement, but the starting point differs.

Method	Use Case
Bend Allowance	Add to flat length to account for stretching in the bend
Bend Deduction	Subtract from total flat length when you already know flange lengths

Bend deduction works when you have flange sizes and want to find the flat layout. If you're building the blank from part geometry, bend allowance is the more direct option.

CAD software may default to one method based on tooling data or machine setup. Pick the one that fits your workflow and stay consistent.

Factors That Affect the K-Factor and BA

Several conditions affect how the K-Factor and bend allowance behave. Each one alters the bend outcome and how you should approach your flat pattern:

- Material Type and Alloy: Different aluminum grades react differently when bent.
- Material Thickness: Thicker sheets shift the neutral axis more and affect stretch amounts.
- Bend Radius: A tighter radius increases elongation; a larger one reduces it.
- Bend Angle: As bend angles become sharper, elongation increases.
- Forming Method
    - Air bending causes more springback and gives a higher K-Factor
        - Air bending does not directly increase the K-Factor but often results in looser bend control, which is associated with higher K-Factor values due to limited tool contact.
    - Bottoming adds control and results in a lower K-Factor
    - Coining compresses the material fully, which gives the most reliable result but takes more force
- Grain Direction: Bending along the grain often leads to cracking. To reduce the risk, bend across the grain whenever possible.
    - Cracking risk increases when bending along the grain, especially in harder or heat-treated alloys like 6061-T6. Softer or annealed tempers are more forgiving.

Skipping any of these variables may lead to incorrect dimensions, added scrap, or extra setup time.

Real-World Example: Flat Pattern Calculation

Let's apply the values:

- Material thickness: 2 mm
- Bend angle: 90°
- Inside radius: 2 mm
- K-Factor: 0.33

Now, insert the numbers:

$$BA = \frac{\pi}{180} x\ 90\ x\ (2 + 0.33\ x\ 2) = \frac{\pi}{180} x\ 90\ x\ 2.66 \approx 5.57\ mm$$

Rounded to one decimal, the bend allowance is about 5.6 mm.

Measure Twice, Calculate Once

Accuracy in bending depends on more than machine setup. The way material stretches in the bend must be accounted for before the part is cut. Using proper K-Factor and bend allowance values makes the difference between a good part and a reject.

Applying these numbers gives better fit, cleaner assemblies, and less scrap. You won't need to rely on trial and error when values are backed by real calculations.

These principles hold up across tools. Getting the math right upfront helps production stay on track, cuts down rework, and keeps tolerances where they need to be.

Follow Inductaflex for more information.

inductaflex