EDU[**CAD**]TION **STUDY GUIDE**

How to Calculate Bend Allowance and Bend Deduction

Chapter 5, Lesson 5

SendCutSend

Bend Allowance

- Bend allowance represents the length of the adjusted centerline after bending. It's the amount of material in the bend region that will stretch and become part of your overall flat length. This is an important calculation to know (see Figure 1.1).

- The main factors influencing bend allowance are:
    - Bend Angle (A)
    - Bend Radius (R)
    - Material Thickness (T)
    - K-Factor (K) Mathematically, bend allowance uses π (3.14) and combines all these variables to estimate the required material length for a bend.

Bend Allowance

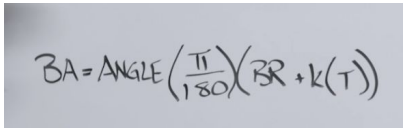


Figure 1.1

Understanding the Stretch

• As metal bends, the outer face stretches and the inner face compresses, like a rubber band being pulled.

• Bend allowance measures that stretch, ensuring the finished part matches the intended size. Larger bend radii or greater angles require higher bend allowances

• Different materials stretch differently.

Example: Aluminum stretches more than steel. This is why the K-Factor, which defines the neutral axis position, is essential. A lower K-Factor (e.g., 0.3) means more stretch; a higher one (e.g., 0.5) means less.
Note: *SendCutSend provides K-Factor data for each supported material and thickness.*

Bend Deduction

• Bend deduction tells you how much to subtract from your flat layout before bending so that your final part comes out to the correct size.

• It's the difference between your flat pattern and the total of both flange lengths. In short: **Bend Deduction** = What you remove now to account for stretching later (see Figure 1.2).

• **Example**: You're designing a part with an overall length of 6 inches and a flange that should be 2 inches after bending. You must shorten your flat layout by the bend deduction amount so that it stretches into the correct final length after bending. If you do not subtract from the overall length the flange will come out longer than intended, (see Figure 1.3) for Jake's example drawing.

Bend Deduction

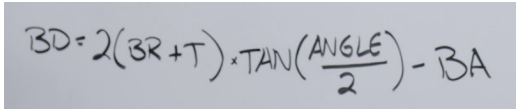


Figure 1.2

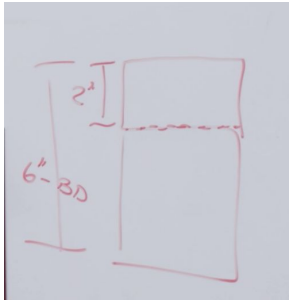


Figure 1.3

Practical Analogy

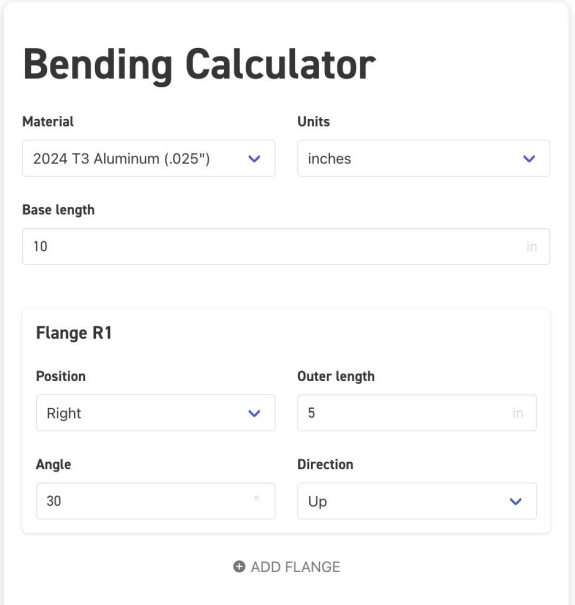
• Think of bend allowance as the path the metal takes while bending, and bend deduction as the adjustment you make to the flat pattern.

• You're predicting the stretch, like throwing a football to where the receiver will be, not where they are.

Why It Matters in CAD:

• Incorrect bend data in CAD can cause flange lengths or hole placements to be inaccurate. Always verify bend radii, K-Factors, and deduction data using SendCutSend's online Bend Calculator.

• Autodesk Fusion and SolidWorks use these values to generate accurate flat patterns.



Summary

Bend allowance and bend deduction convert a theoretical flat pattern into a functional, real-world bent part. By understanding these calculations, you can design with precision, avoid manufacturing errors, and ensure your parts fit together correctly.

Learn more at https://sendcutsend.com/education/