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

How to Design Like a Pro by Knowing Bending Terms

Chapter 5, Lesson 4

SendCutSend

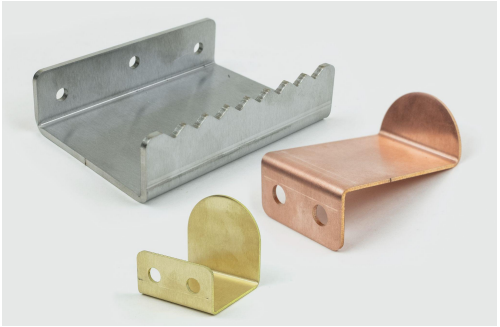
Bending in Sheet Metal

- When a flat sheet of metal is bent, one side compresses and the other side stretches.

- The inner surface of the bend experiences compression, while the outer surface experiences tension.

- The amount of this deformation depends on material thickness, bend radius, and the tooling used (the punch and die).

- To accurately predict how a bend affects part dimensions, we must understand the following core terms.



Key Terms

• **Apex**: The apex is an imaginary point where the two outer tangent lines of adjacent flanges intersect. It's not used in calculations but serves as a reference for key measurements like bend angle, setback, and flange length (see Figure 1.1).

• **Bend Angle:** The bend angle is measured between the outer faces of the flanges, not the inside (see Figure 1.2).

- This means an internal 80° bend corresponds to an external 100° angle.
- When exporting to SendCutSend, ensure your design's bend angles match your intent. STEP files are automatically interpreted correctly, while DXFs show the bend angle during checkout for verification.

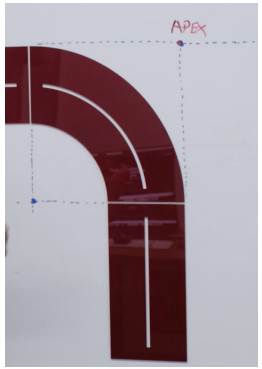


Figure 1.1

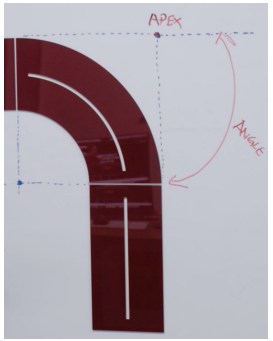


Figure 1.2

Key Terms Continued

• **Setback (SB)**: Setback is the distance from the tangent point of the bend radius to the apex. Each flange has its own setback, which is later used in bend allowance and bend deduction calculations. It helps determine how much flat material is required before bending (see Figure 1.3).

• **Flange Length (FL)**: The flange length is measured from the apex to the end of the flange, never from the inside corner. Measuring from the inside ignores material thickness and leads to incorrect total length. Accurate flange measurement ensures that the finished part dimensions match design intent (see Figure 1.4).

• **Material Thickness (T)**: This refers to the unbent sheet's nominal thickness. It can vary slightly due to manufacturing tolerances in the metal coil. Always use the average or nominal value provided by the supplier when setting up your sheet metal rules in CAD (see Figure 1.5).

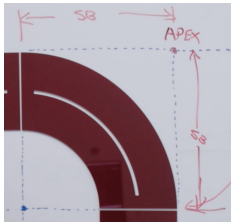


Figure 1.3

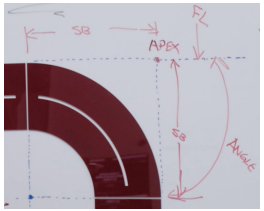


Figure 1.4

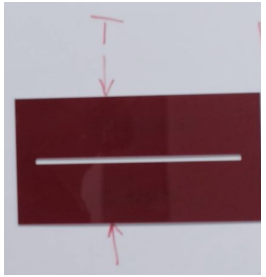


Figure 1.5

Key Terms Continued

• **Centerline (CL)**: The centerline represents the midpoint of the material's thickness in the flat section. In unbent areas, it equals half the material's thickness. In Jake's diagram, the centerline corresponds to the cutout portion of the model (see Figure 1.6).
• However, during bending, the centerline shifts due to stretching and compression, which brings us to the concept of the adjusted centerline.

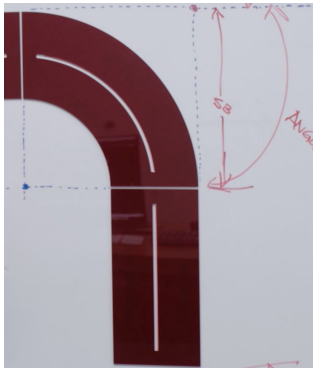


Figure 1.6

• **Bend Radius (BR)**: The bend radius is measured on the inside of the bend, the region under compression (see Figure 1.7).
• It's determined by the punch tool used to form the bend. Different materials and thicknesses require specific bend radii to prevent cracking or deformation.
• SendCutSend provides standard bend radius values for each material and thickness combination on their material guidelines page.

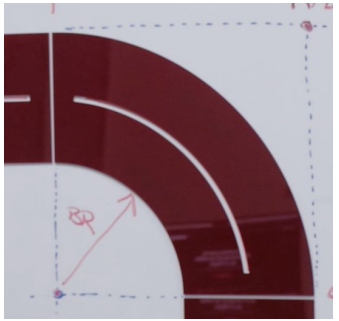


Figure 1.7

Key Terms Continued

• **Adjusted Centerline (t)**: As the metal bends, the material on the outer side stretches, similar to pulling on a rubber band.
• This causes the neutral axis (the line that experiences neither tension or compression) to shift inward.
• The new position of this neutral axis is called the adjusted centerline, and it determines how much the material elongates during bending (see Figure 1.8).

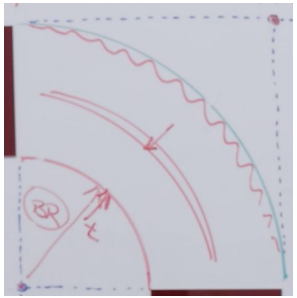


Figure 1.8

• **K-Factor**: The K-Factor is a ratio that defines where the neutral axis lies within the material's thickness during bending. **K = t / T**
• **t** = Distance from the inner bend surface to the neutral axis (adjusted centerline)
• **T** = Material thickness K-Factor values typically range between 0.30 and 0.50 depending on material, thickness, and bend radius.
• A smaller K-Factor indicates more inward shifting of the neutral axis (more stretching). This value is essential for accurate flat pattern calculations in CAD.

Why K-Factor Matters

- In CAD software like **Autodesk Fusion** and **SOLIDWORKS**, the K-Factor is built into sheet metal rules.

- It ensures your flat pattern dimensions match the real-world bent part. If your K-Factor is incorrect, your flanges may come out too long or too short after bending.

- SendCutSend provides downloadable K-Factor and Bend Radius tables for supported materials, along with a Bend Calculator tool to assist with these values.

Bend Calculator: https://sendcutsend.com/bending-calculator/

Summary

Bending metal requires balancing geometry, material science, and precision. By mastering these key terms, apex, setback, flange length, bend radius, and K-Factor, you can confidently design parts that bend accurately and fit perfectly after fabrication.

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