FLAT LAYOUT CHEAT SHEET

Quick Reference Guide for Metal Fabricators

Prepared by: Greg Sheldon
Date: September 2025

FLAT LAYOUT CHEAT SHEET

In metal fabrication, getting the flat layout right is critical.

The flat is the starting point; the piece you cut before bending. If the flat is off, the finished part will be too short or too long, leading to wasted material, lost time, and poor fit.

This cheat sheet gives you a quick reference to the essential formulas and steps for calculating flat layouts using both Inside Dimensions (ID) and Outside Dimensions (OD). It explains the key terms, shows where Bend Allowance (BA) and Bend Deduction (BD) come from, and walks you through worked examples.

Whether you're checking a CAD drawing, programming a press brake, or laying out by hand, these steps will help you understand where the numbers come from and how to apply them with confidence.

LEGEND OF SYMBOLS:

1. **BA = Bend Allowance**: The extra length you add for the bend. Equal to the arc length of the neutral axis through the bend.
2. **BD = Bend Deduction**: The amount you subtract from the total of the outside leg lengths (OD method) to get the correct flat.
3. **OSSB = Outside Setback:** Distance from the tangent point of the bend to the outside edge of the flange.
4. **FSB = Flange Setback** (sometimes used for ID method): Distance from the tangent point of the bend to the end of the inside leg.
5. **θ (theta) = Bend Angle:** The angle of the bend in degrees. A right-angle bend is 90°.
6. **π (pi):** The constant ≈ 3.14159.
7. **R = Inside Bend Radius:** The curved surface radius on the inside of the bend.
8. **T = Material Thickness:** Thickness of the sheet or plate.
9. **K = K-Factor (Neutral Axis Factor)**
   Ratio that defines the neutral axis location relative to the thickness. Typical values range 0.30–0.44.
10. **NA = Neutral Axis:** The theoretical line within the thickness that does not stretch or compress during bending.
11. **FL = Flat Length:** The total developed length of the sheet before bending.
12. **/ = Forward slash:** Symbol for division.

K-FACTOR IN SIMPLE TERMS

When you bend sheet metal:

- The outside of the bend stretches.
- The inside of the bend squeezes (compresses).
- Somewhere in between is a line that doesn't change length. That's the neutral axis.

The **K-Factor** tells you **where that neutral axis is inside the thickness**.

- If the neutral axis is right in the middle, K = 0.50.

- If the neutral axis is right at the inside surface, K = 0.00.
- In real bends, it's usually somewhere in between (0.30–0.45).

Think of K-Factor as a "slider" between inside and middle of the sheet.

Why It Matters

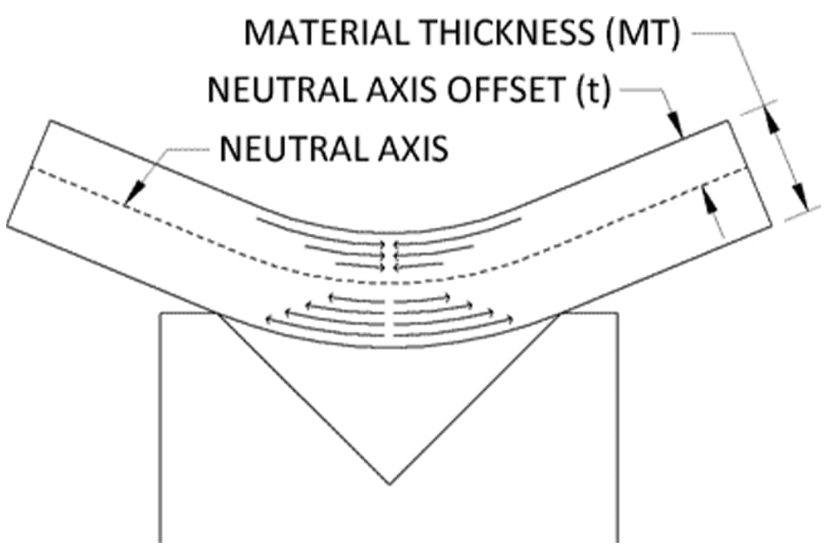
The K-Factor controls the **bend allowance (BA)** you add to your flat.

- **Higher K** → neutral axis closer to the middle → BA is **larger**.
- **Lower K** → neutral axis closer to the inside → BA is **smaller**.

Common K-Factors

- Mild Steel: **0.38**
- Aluminum: **0.33**
- Stainless Steel: **0.44**

NEUTRAL AXIS AND WHY IT MATTERS:



What the diagram shows:

- **Material Thickness (MT):** The full thickness of your sheet/plate.
- **Neutral Axis:** The line inside the bend that doesn't stretch or shrink.
- **Neutral Axis Offset (t):** How far the neutral axis shifts from the center of the thickness toward the inside of the bend.

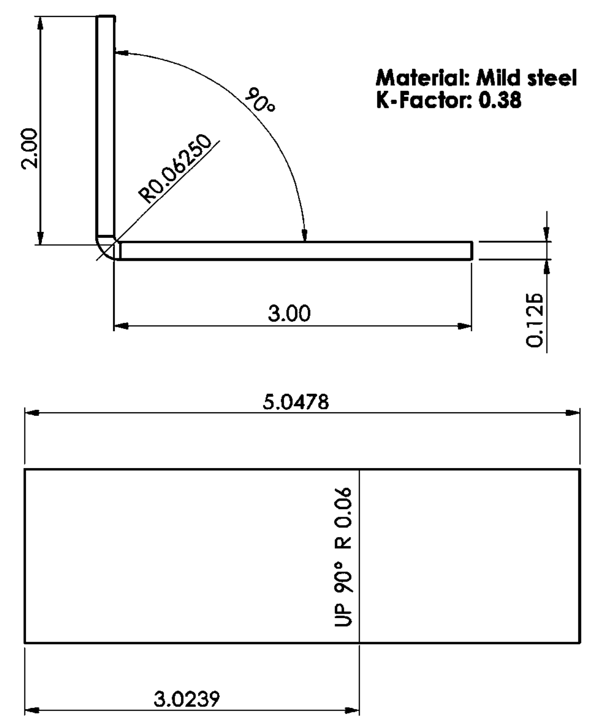
Why it matters:

- The outside of the bend **stretches** the inside **compresses**.
- The neutral axis is where the material length stays the same.

- Your bend allowance is based on the **arc length of this neutral axis**.
- If you ignore the neutral axis, your flat layout will be off; parts come out short or long.
- Different materials and bend radii move the neutral axis by different amounts (that's what the **K-Factor** measures).
- Knowing where it is gives you **accurate flat patterns**, less scrap, and faster fitting parts.

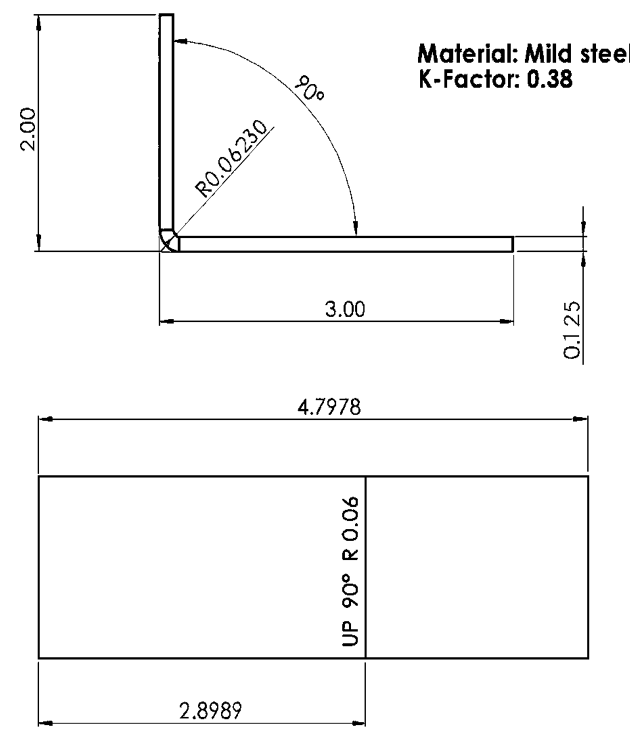
Tip: Always test a bend in the same material and thickness first. Once you know your neutral axis offset (K-Factor), you can trust your flats to come out right every time.

<u>THE PROCESS</u>

<table>
<tr><th>INSIDE DIMENSIONS (ID)</th><th>OUTSIDE DIMENSIONS (OD)</th></tr>
<tr><td>


STEP 1: IDENTIFY YOUR VALUES

First, you need to find the essential values from your part drawing.

- **Leg 1 Dimension:** 2.00 in.
- **Leg 2 Dimension:** 3.00 in.
- **Inside Radius (R):** 0.06250 in.
- **Material Thickness (T):** 0.125 in.
- **K-Factor (K):** 0.38
</td><td>


STEP 1. IDENTIFY THE VALUES:

First, you need to find the essential values from your part drawing.

- **Outside Leg 1 Dimension:** 2.00 in.
- **Outside Leg 2 Dimension:** 3.00 in.
- **Inside Radius (R):** 0.06250 in.
- **Material Thickness (T):** 0.125 in.
- **K-Factor (K):** 0.38
</td></tr>
</table>

- **Bend Angle (A):** The drawing shows a 90° angle. For this type of bend, the bend angle is also 90°.

STEP 2: CALCULATE THE BEND ALLOWANCE (BA)

The **Bend Allowance** is the amount of material that stretches during the bend. Use this formula:

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

Plug in the values you identified:

- $BA = 90 \times \pi/180 \times (0.06250 + 0.38 \times 0.125)$

- $BA = 0.1728$ in.

STEP 3: CALCULATE THE TOTAL FLAT LAYOUT LENGTH

The total flat length is the sum of the straight sections of the part plus the bend allowance.

Remember to subtract the inside radius (R) from each leg's length to find its true flat length.

- **Flat Leg 1 Length:** (Leg 1 Dimension) - R = 2.00 − 0.06250 = 1.9375 in.

- **Flat Leg 2 Length:** (Leg 2 Dimension) - R = 3.00 − 0.06250 = 2.9375 in.

Now, add these flat lengths to the bend allowance:

Total Flat Length = (Flat Leg 1 Length) + (Flat Leg 2 Length) + BA

Total Flat Length = 1.9375 + 2.9375 + 0.1728

TOTAL FLAT LENGTH = 5.0478 IN.

- **Bend Angle (A):** 90°

STEP 2. CALCULATE THE BEND ALLOWANCE (BA).

This step is the same as the inside dimension method because the bend allowance formula doesn't change.

$BA = 90 \times \pi/180 \times (0.06250+0.38\times0.125)$

$BA = 0.1728$ in.

STEP 3. CALCULATE THE BEND DEDUCTION (BD).

Now, use the outside dimensions formula.

- $BD = 2 \times (R + T) - BA$

- $BD = 2 \times (0.06250 + 0.125) - 0.1728$

- $BD = 2 \times (0.1875) - 0.1728$

- $BD = 0.375 - 0.1728$

- $BD = 0.2022$ in.

STEP 4. CALCULATE THE TOTAL FLAT LAYOUT LENGTH.

Subtract the bend deduction from the sum of the outside leg dimensions.

Total Flat Length = (Outside Leg 1) + (Outside Leg 2) - BD

Total Flat Length = 2.00 + 3.00 − 0.2022

Total Flat Length = 5.00 − 0.2022

TOTAL FLAT LENGTH = 4.7978 IN.

<u>CONCLUSION</u>

Flat layout calculations always come down to the same principle: the flat length equals the sum of the straight legs plus or minus the bend zone adjustment.

When dimensions are given to the inside corners (ID), you must add Bend Allowance (BA).

When dimensions are given to the outside corners (OD), you must subtract Bend Deduction (BD).

The key variables are the inside bend radius (R), material thickness (T), bend angle (θ), and K-factor (K), which controls where the neutral axis lies.

By correctly identifying these values and applying the proper method, you ensure the flat pattern matches the finished part.

Always remember:

- **ID drawing → Flat = Leg1 + Leg2 + BA**
- **OD drawing → Flat = Leg1 + Leg2 − BD**

For highest accuracy, verify your K-factor with a sample bend and use that value consistently. This guarantees your hand calculations, CAD software, and press brake all agree, reducing scrap and saving time on the shop floor.

Best,
Greg Sheldon