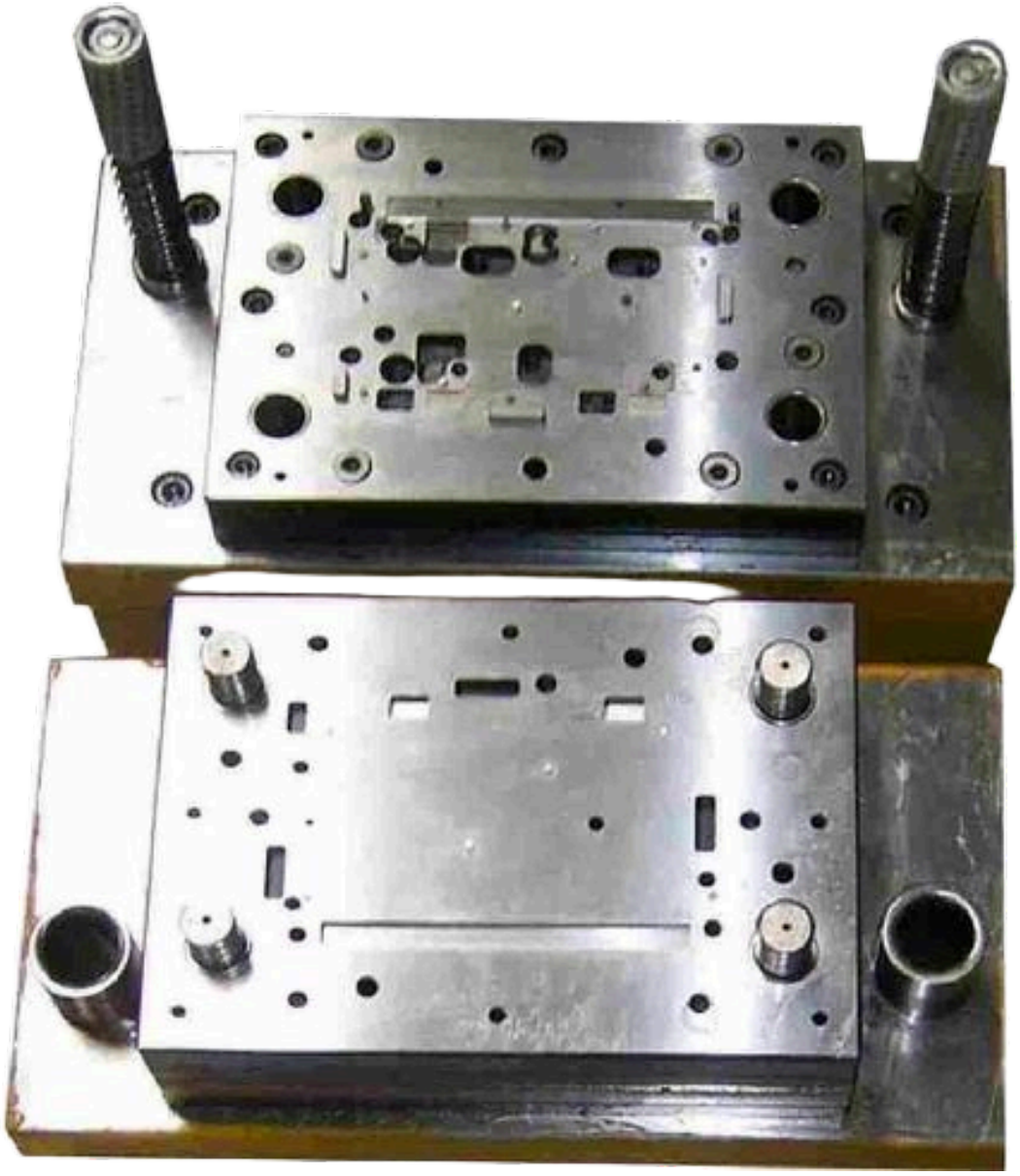
The Number That Decides Every Flat Pattern Is One You Cannot Measure On The Drawing

The K-factor sits between the drawing and the blank. Get it wrong and every formed dimension moves - here is where it comes from and how to control it.



DEEP DRAWING & METAL STAMPING
Zhuji, Zhejiang, China

BALFORD.NET
shawn@balford.net
2026-10-13

A customer sends a bracket. It is simple: two bends, a 90 degree angle, dimensions that all make sense. We cut the flat blank, form it, and the finished part is 0.6 mm long on one leg. Nothing about the drawing is wrong. What was wrong was the number nobody wrote down, because it does not live on the drawing at all.

That number is the K-factor. It describes how far the neutral axis sits from the inside surface of the bend, as a fraction of the sheet thickness. Metal on the outside of a bend stretches, metal on the inside compresses, and somewhere between them there is a layer that does neither. That layer is not in the middle of the sheet, and its position is what decides how much flat material a bend actually consumes.

In the bend allowance formula the K-factor is the only input you cannot read off the print. Every other term - thickness, radius, bend angle - is either drawn or specified. The K-factor has to be supplied, and when it is supplied by default it is usually supplied wrong for that particular part.

What surprised me when I started tracking this properly is how wide the legitimate range is. Most general bending sits around 0.40 to 0.45, which is why that has become the usual CAD default. But a tight bend in material that work-hardens quickly pulls the neutral axis well inside, down toward 0.33, while a generous radius in soft aluminium or copper lets it drift toward the middle of the sheet at 0.50. That is not a small spread. On a 2 mm sheet it is the difference between a flat pattern that fits and one that does not.

Four things move it, and only one of them is the material.

The ratio of inside radius to thickness is the biggest influence. A tight bend in 1 mm sheet and a generous bend in 3 mm sheet are not the same behaviour even in the same grade, because the strain distribution through the thickness is completely different.

Then there is how the material behaves. The harder a material work-hardens, the more the neutral axis shifts inward during the bend. That is why 304 and 430 stainless sit at the low end of the range while soft aluminium and copper sit at the high end. Same geometry, different answer.

The tooling matters more than most people expect. In air bending the die opening sets the inside radius that actually forms - a drawing that prints R2 may come off the press at R1.8 or R2.4 depending on the die it was run in. If the radius the calculation used is not the radius the tool produced, the flat pattern inherits the error.

And the rolling direction is real. On a part with several bends, running some of them across the grain versus along it explains the flat pattern that fits in one orientation and not the other. It is small, but it is the kind of small that shows up as a scrap pile.

So what do we actually do? We start from the drawing with a K-factor in the normal range for that material and R/T ratio, cut and form the first part, measure the finished legs, and correct before the batch runs. For a single-bend bracket that loop takes two minutes. For a welded frame with a dozen bends, it is the thing that stops the frame arriving at the welding table a couple of millimetres out over its length, which is a much more expensive conversation.

One detail worth keeping straight: where a part is formed in a die rather than on a press brake, the geometry comes from the tool and the flat pattern from the strip layout. Those are different calculations and they should never be mixed. A part that is bent on a brake and a part that is formed in a die do not share a flat pattern method just because they share a material.

If you want a flat pattern from us, the useful things to send are: the finished part drawing with every bend angle and inside radius stated; the material and thickness, including the temper if it is not the standard condition; whether the part will be air bent, bottomed or coined, if that is already decided; the dimensions that actually have to be held, whether that is the overall outside size, a hole-to-bend distance, or the blank itself; and any bend that runs across the rolling direction where the visible grain matters.

The conclusion I would put in front of any engineer is this. The K-factor is an input to a calculation, not a property of a material. It is only correct for the radius-to-thickness ratio, the material and the tooling it was measured on. Confirm it on the first article, keep it with the part record, and the flat pattern becomes repeatable for the life of the programme.

The longer version, with the range table and the bend allowance formula, is here: https://balford.net/k-factor-sheet-metal/

I am the founder of Balford, a deep drawing and metal stamping manufacturer in Zhuji, China. We deep draw and stamp metal housings on presses from 25 t to 350 t, to a maximum draw diameter of 250 mm, and quote from the drawing.

SEND A DRAWING, GET A REAL ANSWER

Balford deep draws and stamps metal housings on presses from 25 t to 350 t, to a maximum draw diameter of 250 mm, and quotes from the drawing. **balford.net** · **rfq.balford.net** · shawn@balford.net