

Sheet-metal design guide

Designing bent sheet-metal parts — bend allowance and the K-factor, minimum bend radii and reliefs, hole and edge rules, and the DFM that keeps parts cheap and accurate.

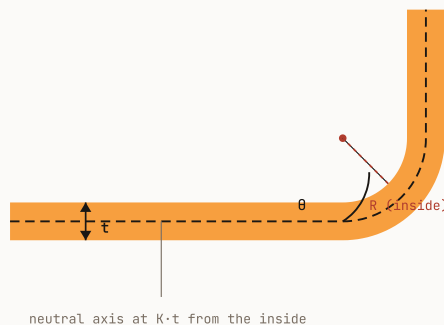
REVISION	ISSUED	OWNER	COMPANION
1.0	June 2026	Ideambox engineering	PDF + Excel bend calculator

ABSTRACT

Sheet-metal parts are cut flat (laser/punch) and folded on a press brake, so the design problem is two-fold: lay out features that survive bending, and compute a flat pattern that folds to the right finished size. The bend allowance and K-factor handle the second; min-radius, relief and edge rules handle the first.

Section 1 is DFM fundamentals. Section 2 is bend allowance and the K-factor. Section 3 is minimum bend radius and reliefs. Section 4 is holes, slots and edge distances. Section 5 is forming features and hardware. Section 6 is cost and a checklist. A companion **Excel bend calculator** computes BA / BD / flat length live.

SHEET-METAL BEND — ALLOWANCE & K-FACTOR



$$BA = \theta_{rad} \cdot (R + K \cdot t)$$

$$BD = 2(R+t) \cdot \tan(\theta/2) - BA$$

$$flat = A + B - BD$$

$$K \approx 0.33-0.45 (\approx 0.40 \text{ typical})$$

A BEND STRETCHES THE OUTSIDE AND COMPRESSES THE INSIDE; THE NEUTRAL AXIS SITS AT $K \cdot t$ FROM THE INSIDE. GET THE K-FACTOR AND BEND ALLOWANCE RIGHT AND THE FLAT PATTERN COMES OUT THE CORRECT SIZE FIRST TIME.

CONTENTS

- | | |
|------------------------------------|------------------------------------|
| 1. DFM fundamentals | 4. Holes, slots and edge distances |
| 2. Bend allowance and the K-factor | 5. Forming features and hardware |
| 3. Minimum bend radius and reliefs | 6. Cost and checklist |

1. DFM fundamentals

Sheet metal is one constant thickness, cut flat then formed — so design with that grain:

- **One material thickness per part; pick a standard gauge.**
- **Bends are cheap, cuts are cheap; complexity in setups is not**
minimise the number of bends and unique bend angles.
- **Uniform everything**
consistent bend radius across the part lets one tool do all bends.
- **Typical process: laser/punch the flat blank → press-brake the bends → hardware (PEM) → finish.**

Neutral axis	The layer that neither stretches nor compresses in a bend — sits at $K \cdot t$ from the inside
K-factor	Position of the neutral axis as a fraction of thickness ($\approx 0.33-0.45$)
Bend allowance (BA)	Arc length of the neutral axis through the bend — added to flat legs
Bend deduction (BD)	Amount subtracted from the summed outside dimensions to get the flat length
Bend relief	A small notch at a bend end that stops tearing

2. Bend allowance and the K-factor

When sheet bends, the outside stretches and the inside compresses; the **neutral axis** (length unchanged) lies at $K \cdot t$ from the inside surface. The flat pattern length follows from it:

- Bend allowance $BA = \theta_{\text{rad}} \cdot (R + K \cdot t)$ (θ = bend angle, R = inside radius)
- Outside setback $OSSB = (R + t) \cdot \tan(\theta/2)$
- Bend deduction $BD = 2 \cdot OSSB - BA$
- Flat length (single bend, outside legs A and B) $= A + B - BD$

K depends on radius/thickness ratio and material; use ≈ 0.40 as a default, then refine from test bends or the brake's data:

CONDITION	TYPICAL K
Soft material / small R ($R < t$)	~ 0.33
General ($R \approx t$)	~ 0.40
Hard material / large R ($R > 2t$)	~ 0.45

The companion **Excel calculator** computes BA, BD and flat length from t , θ , R and K live.

3. Minimum bend radius and reliefs

- Minimum inside radius rises with thickness and hardness. Bending tighter cracks the outside fibre, especially across the grain:

MATERIAL	MIN INSIDE RADIUS
Soft aluminium (5052-O)	$0.5-1 \times t$
Mild steel	$\sim 1 \times t$
Stainless / half-hard	$1-2 \times t$
Hard tempers (6061-T6)	$2-3 \times t$

- Bend relief: add a relief notch ($\geq t$ wide, $\geq t$ deep) where a bend ends at an edge, or the corner tears.
- Bend orientation: bend across the rolling grain where possible
bends parallel to the grain crack sooner.

4. Holes, slots and edge distances

Punched/laser features need clearance or they distort during bending:

FEATURE	RULE OF THUMB
Min hole / slot diameter	\geq material thickness t
Hole / slot to edge	$\geq 2 \times t$
Hole / slot to bend	$\geq 2.5 \times t + R$ (else it deforms)
Min flange length (leg)	$\geq 4 \times t + R$ (to grip the brake)
Notch / tab width	$\geq 2 \times t$

5. Forming features and hardware

- **Hems (folded edges) stiffen and remove sharp edges; allow ~ flat-hem clearance.**
- **Countersinks in thin sheet are shallow**
limit to $\sim 0.6 \times t$ depth or use extruded holes.
- **Louvers, ribs, embosses stiffen a panel cheaply versus adding thickness.**
- **Press-in hardware (PEM nuts/studs) add threads without welding**
respect the vendor's min sheet thickness and edge distance.
- **Tolerances: bends carry ± 0.1 – 0.5 mm and angular ± 0.5 – 1° typically; don't stack tight tolerances across multiple bends (see Tolerance stack-up).**

6. Cost and checklist

- **Fewer bends, one radius, standard gauge**

the three biggest cost levers.

- **Avoid tight tolerances across bends**

each bend adds variation.

- **Nest-friendly flat**

simple outline, minimal scrap.

- **Checklist: standard thickness → consistent bend radius \geq min for the material → features \geq rules above from edges/bends → bend reliefs at edges → compute the flat pattern (BA/BD via the calculator) → realistic bend tolerances → hardware edge distances.**