

## 'K' Factors For Solidworks

Material	Thickness	Bend Radius	Normal Bend	Hem Bend	B/A
Mild Steel	0.7mm	1.3mm	K 0.509	K 0.625	0.0
	0.9mm	1.3mm	K 0.395213		0.0
	1.0mm	1.3mm	K 0.355213		0.0
	1.2mm	1.3mm	K 0.455	K 0.625	0.2
	1.5mm	1.3mm(8mmV)	K 0.415	K 0.625	0.41
	1.5mm	1.5mm(10mmV)	K0.3794	K 0.625	0.25
	2.0mm	2.0mm	K 0.4325	K 0.625	0.5
	2.5mm	2.5mm	K 0.46425		0.75
	3.0mm	2.0mm(12mmV)	K 0.501		1.5
	3.0mm	2.6mm(16mmV)	K0.4855		
	3.0mm	3.0mm	K 0.4855		1.0
	4.0mm	4.0mm	K 0.512		1.5
	5.0mm	4.0mm	K 0.4425		1.75
	6.0mm	6.0mm(40mm V)	K 0.4857		2.0
St, Steel	6.0mm	4.0mm(25mm V)	K0.501		3.0
	10.0mm	8.0mm	K0.675		7.16mm
	0.9mm	1.3mm	K 0.3955		0.0
	1.0mm	1.3mm	K 0.3555		0.0
	1.2mm	1.3mm	K 0.29601	K 0.625	0.0
	1.5mm	1.5mm	K 0.3801		0.25
	1.5mm	1.5mm	K 0.27326		0.0
	2.0mm	2.0mm	K 0.27324		0.0
	2.5mm		K		0.0
	3.0mm	3.0mm	K 0.321		0.0
Aluminium	0.9mm	1.3mm	K 0.355		0.0
	1.2mm	1.3mm	K 0.454		0.25
Air Bend	1.5mm	1.5mm	K 0.4855		0.5
Coin Bend	1.5mm	1.5mm	K0.55		0.65
	2.0mm	2.0mm	K 0.5500		0.90
	2.5mm	2.5mm	K		
	3.0mm	3.0mm	K 0.59156		1.5
	5.0mm	5.0mm	K0.5279		2.0

Bending is a manufacturing process by which metal can be deformed by plastically deforming the material and changing its shape. The material is stressed beyond its yield strength but below its ultimate tensile strength. There is little change to the materials surface area. Bending generally refers to deformation about one axis only.

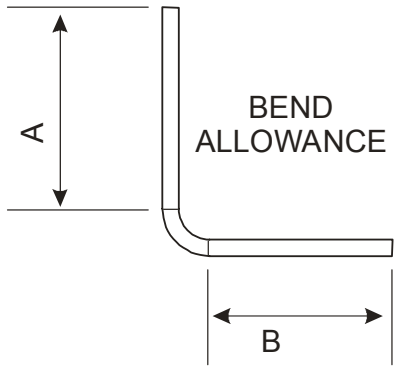
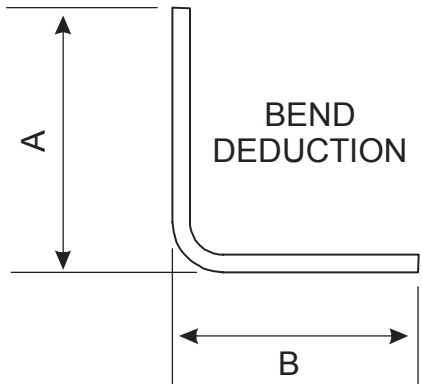
Bending is a flexible process by which a variety of different shapes can be produced though the use of standard die sets or bend brakes. The material is placed on the die, and positioned in place with stops and/or gages. It is held in place with hold-downs. The upper part of the press, the ram with the appropriately shaped punch descends and forms the v-shaped bend.

Bending is done using Press Brakes. Press Brakes can normally have a capacity of 20 to 200 tons to accommodate stock from 1m to 4.5m (3 feet to 15 feet). Larger and smaller presses are used for diverse specialized applications. Programmable back gages, and multiple die sets currently available can make bending a very economical process.

## BEND ALLOWANCES

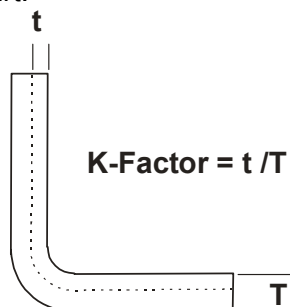
When sheet metal is bent, the inside surface of the bend is compressed and the outer surface of the bend is stretched. Somewhere within the thickness of the metal lies its Neutral Axis, which is a line in the metal that is neither compressed nor stretched.

What this means in practical terms is that if we want a work piece with a 90 degree bend in which one leg measures A, and the other measures B, then the total length of the flat piece is NOT A + B as one might first assume. To work out what the length of the flat piece of metal needs to be, we need to calculate the **Bend Allowance** or **Bend Deduction** that tells us how much we need to add or subtract to our leg lengths to get exactly what we want.

	
$L_t = A + B + BA$	$L_t = A + B - BD$
<p>where:  <math>L_t</math> is the total flat length            A and B are shown in the illustration            BA is the bend allowance value</p>	<p>where:  <math>L_t</math> is the total flat length            A and B are shown in the illustration            BD is the bend deduction value</p>

The location of the neutral line varies depending on the material itself, the radius of the bend, the ambient temperature, direction of material grain, and the method by which it is being bent, etc. The location of this line is often referred to as the **K factor**.

K-factor is a ratio that represents the location of the neutral sheet with respect to the thickness of the sheet metal part.



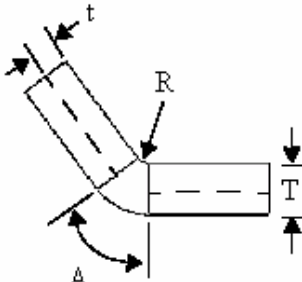
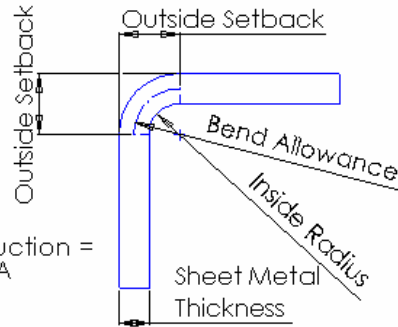
The only truly effective way of working out the correct bend allowance is to reverse engineer it by taking a measured strip of material, bending it, and then measuring it

to calculate the bend allowance. These bend allowance can be measured for many materials and scenarios and then tabulated so that the table can be used by CAD programs to produce accurate sheet-metal work.

Many CAD programs, however, also work out bend allowances automatically by using K-factor calculations.

(Or Y-factor in the case of Pro-E where Y-factor = K-factor \*  $\pi$  / 2).

Bend allowances are calculated using a K-factor as follows:


$BA = \pi(R + KT) A / 180$
<p>where:</p> <p>BA = bend allowance</p> <p>R = inside bend radius</p> <p>K = K factor, which is t / T</p> <p>T = material thickness</p> <p>t = distance from inside face to neutral sheet</p> <p>A = bend angle in degrees (the angle through which the material is bent)</p>
 <p style="text-align: left;">Bend Deduction = <math>2 * OSSB - BA</math></p>

This works extremely well and is pretty straight forward, **providing we know the correct K-factor to use.**

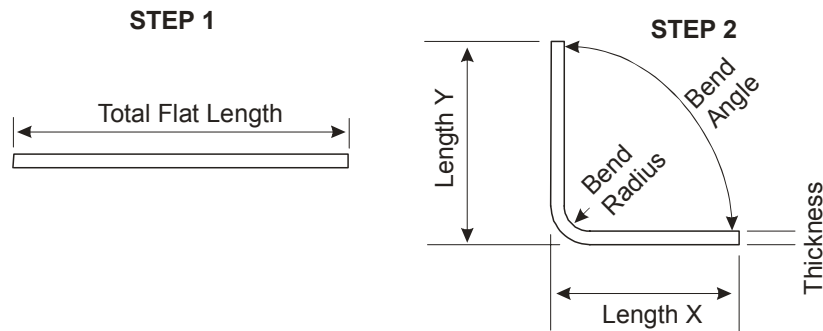
Once again, the most accurate way of finding the correct K-factor to use in your CAD program is by using the reverse engineering method described above, and calculating the K-factor to use as described in the following section:

## REVERSE ENGINEERING THE K-FACTOR

First, cut a strip of material and measure its length and thickness as accurately as possible. The width of the strip is not that critical but generally somewhere around 100mm (4 inches) or so usually does the trick.

Then, bend the strip to 90 degrees, and measure its Length X and Length Y as shown in the diagram below.

**Note that it is very important to bend the sample piece in exactly the same manner as you plan to bend your real pieces, so that whatever you measure now becomes reproducible later.**



The bend radius can be extremely difficult to measure accurately but, in this case, is not critical (within reasonable limits, of course!). The reason it is not critical is that what we are interested in is a number to use in our CAD program that, with the bend radius used in our CAD program, will produce the results you are measuring in real life.

In other words the K-factor you calculate now will take into account any small inaccuracies in the bend radius measurement and compensate for it. If, for example, we are using a Bend radius of 0.5 in our CAD program, it does not matter if our real tooling radius is actually 0.4, as the K-factor, which was worked out from our real tooling, corrects for this. The only implication of this is that we may occasionally get a K-factor that seems odd (higher than 0.5, for example) if our real radius is very different from our CAD program radius. Remember though that most CAD programs such as Solidworks only accept K-factor values from 0 to 1, so if the calculated K-factor is outside these limits, then you may need to double-check your numbers.

The correct K-factor to use in your CAD program can now be calculated as follows:

$$\text{BendDeduction} = \text{Length X} + \text{Length Y} - \text{Total Flat Length}$$

$$\text{OutSideSetBback} = (\text{Tan}(\text{BendAngle} / 2)) * (\text{thickness} + \text{BendRadius})$$

$$\text{BendAllowance} = (2 * \text{OutSideSetBback}) - \text{BendDeduction}$$

$$\text{K-Factor} = (-\text{BendRadius} + (\text{BendAllowance} / (\pi * \text{BendAngle} / 180))) / \text{thickness}$$

Using this method will produce the most acceptable results other than by using a bend table.

There are however also some general rules of thumb that can be used for K-factors that will generally give results that are within acceptable tolerances for non-precision sheet metal work. Some of these sample K-factors are given in the methods of bending section below.

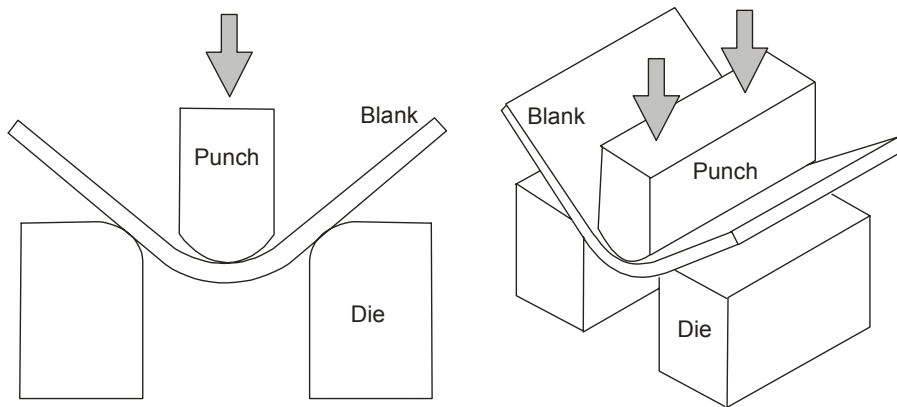
## PRINCIPLE TYPES OF BENDING

### AIR BENDING

*Air Bending* is a bending process in which the punch touches the work piece and the work piece does not bottom in the lower cavity. As the punch is released, the work piece springs back a little and ends up with less bend than that on the punch (greater included angle). This is called spring-back.

The amount of spring back depends on the material, thickness, grain and temper. The spring back will usually range from 5 to 10 degrees. The same angle is usually used in both the punch and the die to minimize set-up time. The inner radius of the bend is the same as the radius on the punch.

In air bending, there is no need to change any equipment or dies to obtain different bending angles because the bend angles are determined by the punch stroke. The forces required to form the parts are relatively small, but accurate control of the punch stroke is necessary to obtain the desired bend angle.



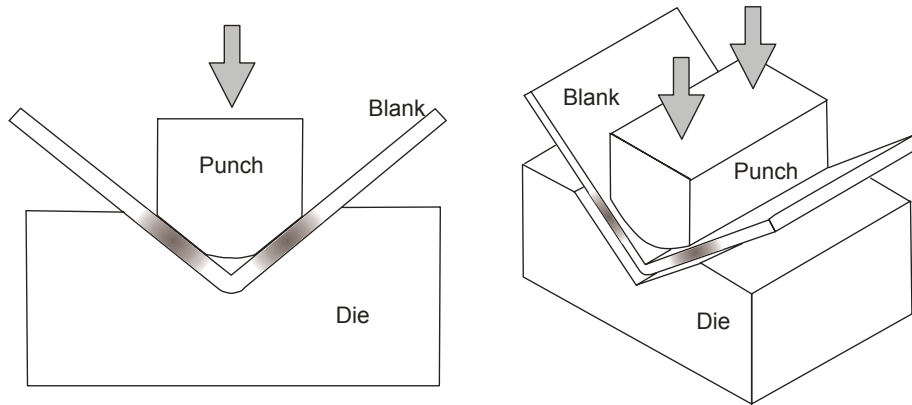
#### K-Factor Rule of thumb for Air Bending

Radius	Soft-material	Medium Material	Hard Material
0 to thickness	0.33	0.38	0.4
Thickness to 3 x thickness	0.4	0.43	0.45
Greater than 3 x thickness	0.5	0.5	0.5

### BOTTOMING

*Bottoming* is a bending process where the punch and the work piece bottom on the die. This makes for a controlled angle with very little spring back. The tonnage required on this type of press is more than in air bending. The inner radius of the work piece should be a minimum of 1 material thickness.

In bottom bending, spring-back is reduced by setting the final position of the punch such that the clearance between the punch and die surface is less than the blank thickness. As a result, the material yields slightly and reduces the spring-back. Bottom bending requires considerably more force (about 50%~60% more) than air bending.

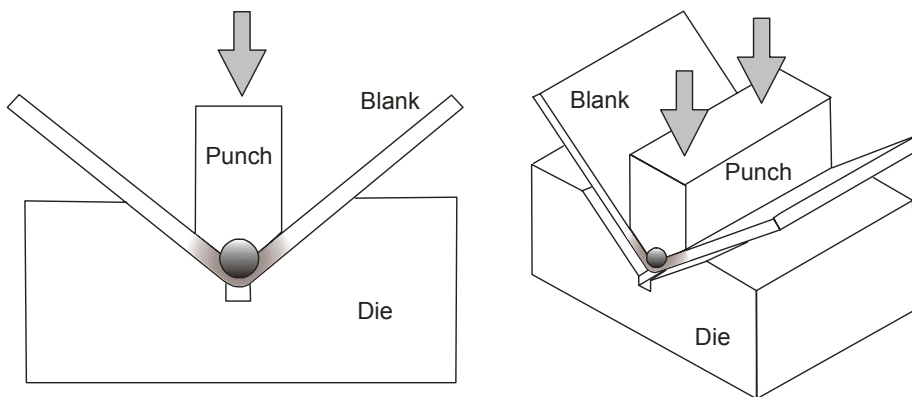


K-Factor Rule of thumb for Bottoming

Radius	Soft-material	Medium Material	Hard Material
0 to thickness	0.42	0.44	0.46
Thickness to 3 x thickness	0.46	0.47	0.48
Greater than 3 x thickness	0.5	0.5	0.5

## COINING

Coining is a bending process in which the punch and the work piece bottom on the die and compressive stress is applied to the bending region to increase the amount of plastic deformation. This reduces the amount of spring-back. The inner radius of the work piece should be up to 0.75 of the material thickness.



K-Factor Rule of thumb for Coining

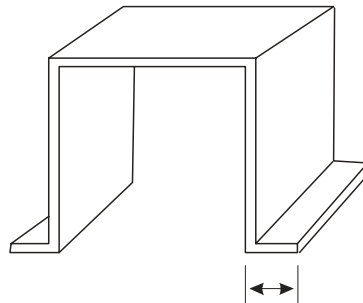
Radius	Soft-material	Medium Material	Hard Material
0 to thickness	0.38	0.41	0.44
Thickness to 3 x thickness	0.44	0.46	0.47
Greater than 3 x thickness	0.5	0.5	0.5

## TIPS AND TRICKS

---

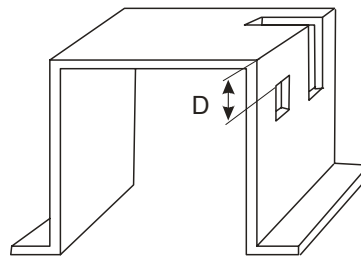
General bending guidelines are as follows:

- The bend radius should, if possible, be kept the same for all radiuses in the part to minimize set up changes.
- For most materials, the ideal minimum inner radius should be at least 1 material thickness.
- As a general rule, bending perpendicular to the rolling direction is easier than bending parallel to the rolling direction. Bending parallel to the rolling direction can often lead to fracture in hard materials. Thus bending parallel to the rolling direction is not recommended for cold rolled steel > Rb 70. And no bending is acceptable for cold rolled steel > Rb 85. Hot rolled steel can however be bent parallel to the rolling direction.
- The minimum flange width should be at least 4 times the stock thickness plus the bending radius. Violating this rule could cause distortions in the part or damage to tooling or operator due to slippage.



$$\text{Min Flange Width} = 4 \times \text{Thickness} + \text{Bend radius}$$

- Slots or holes too close to the bend can cause distortion of these holes. Holes or slots should be located a minimum of 3 stock thickness plus the bend radius. If it is necessary to have holes closer, then the hole or slot should be extended beyond the bend line.



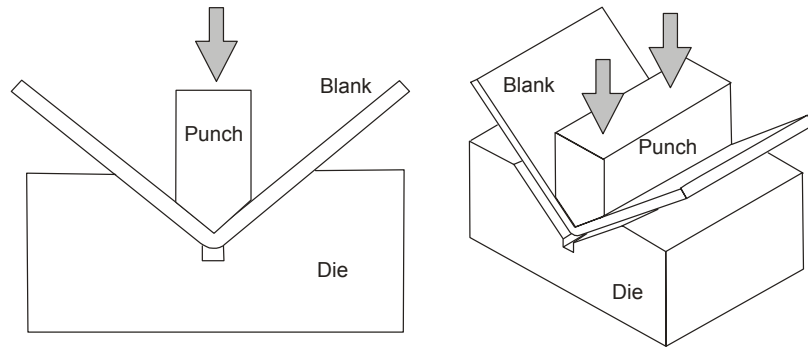
$$D = 3 \times \text{Thickness} + \text{Bend radius}$$

- Dimensioning of the part should take into account the stack up of dimensions that can happen and mounting holes that can be made oblong should be.
- Parts should be inspected in a restrained position, so that the natural flexure of the parts does not affect measurements. Similarly inside dimensions in an inside bend should be measured close to the bend.

## OTHER COMMON TYPES OF BENDING

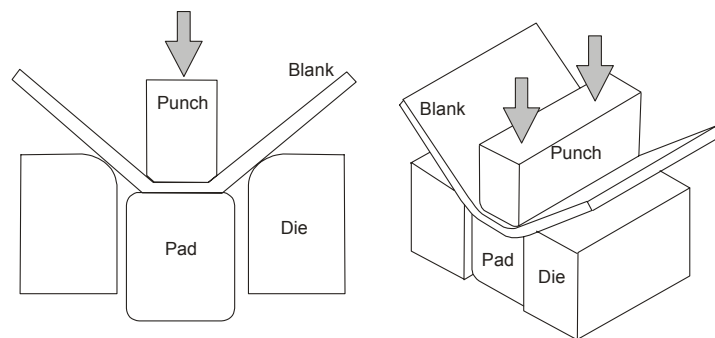
### V Bending

In V-bending, the clearance between punch and die is constant (equal to the thickness of sheet blank). It is used widely. The thickness of the sheet ranges from approximately 0.5 mm to 25 mm.



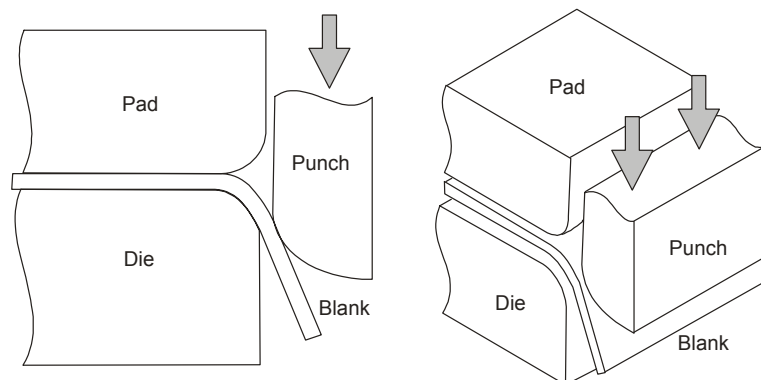
### U DIE BENDING

U-die bending is performed when two parallel bending axes are produced in the same operation. A backing pad is used to force the sheet contacting with the punch bottom. It requires about 30% of the bending force for the pad to press the sheet contacting the punch.



### WIPING DIE BENDING

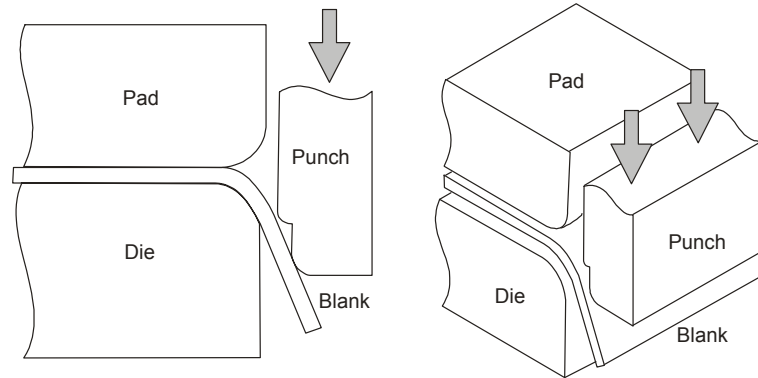
Wiping die bending is also known as flanging. One edge of the sheet is bent to 90° while the other end is restrained by the material itself and by the force of blank-holder and pad. The flange length can be easily changed and the bend angle can be controlled by the stroke position of the punch.





## DOUBLE DIE BENDING

Double die bending can be seen as two wiping operations acting on the work piece one after another. Double bending can enhance strain hardening to reduce spring-back.



## ROTARY BENDING

Rotary bending is a bending process using a rocker instead of the punch. The advantages of rotary bending are:

- Needs no blank-holder
- Compensates for spring-back by over-bending
- Requires less force
- More than 90 degree bending angle is available

