The Fine Art of Sheet Metal Fabrication

Posted on Dec 16, Posted by P&A International Category General Talk

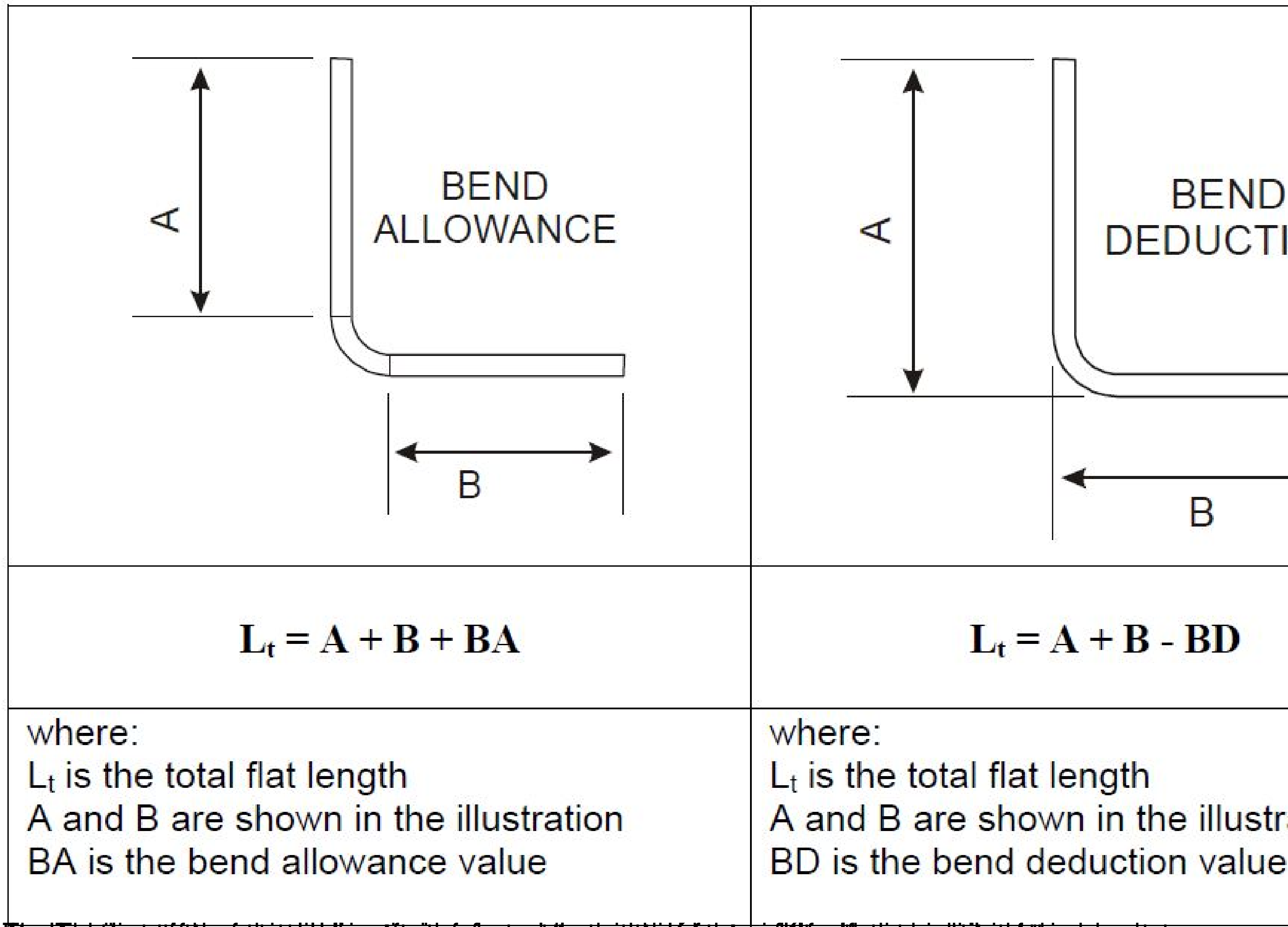
Bending is a manufacturing process by which sheet metal can be deformed by plastically deforming the material and changing its shape. The material is stressed beyond its yield strength but below its ultimate tensile strength. There is little change to the materials surface area. Bending generally refers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

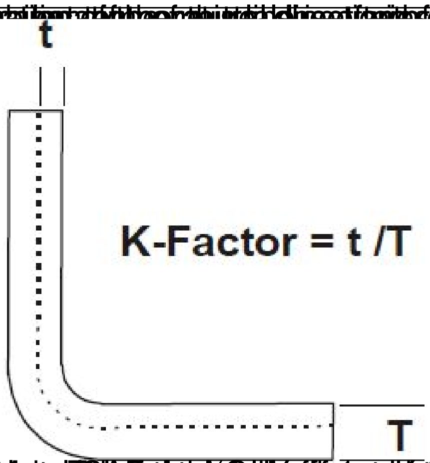
Bending is a flexible process by which a variety of different shapes can be produced though the use of standard die sets or bend brakes. The material is placed on the die, and positioned in place with stops and/or gages. It is held in place with hold-downs. The upper part of the press, the ram with the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

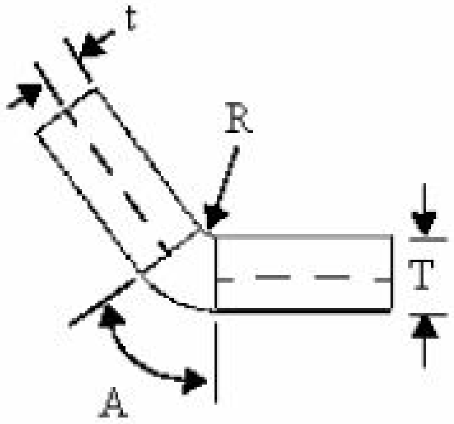
Bending is done using Press Brakes. Press Brakes can normally have a capacity of 20 to 200 tons to accommodate stock from 1m to 4.5m (3 feet to 15 feet). Larger and smaller presses are used for diverse specialized applications. Programmable back gages, and multiple die sets currently available can make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

BEND ALLOWANCES

During sheet metal fabrication , when material 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.

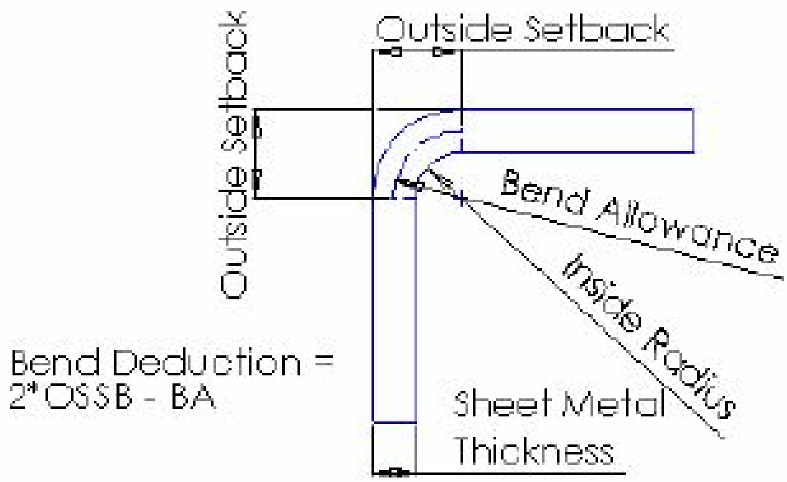

BEND
ALLOWANCE
A
B
$L_t = A + B + BA$
where:
L_t is the total flat length
A and B are shown in the illustration
BA is the bend allowance value
BEND
DEDUCTI
A
B
$L_t = A + B - BD$
where:
L_t is the total flat length
A and B are shown in the illustr
BD is the bend deduction value

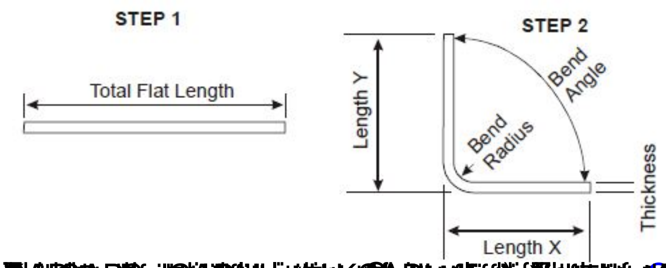

t
K-Factor = t /T
T



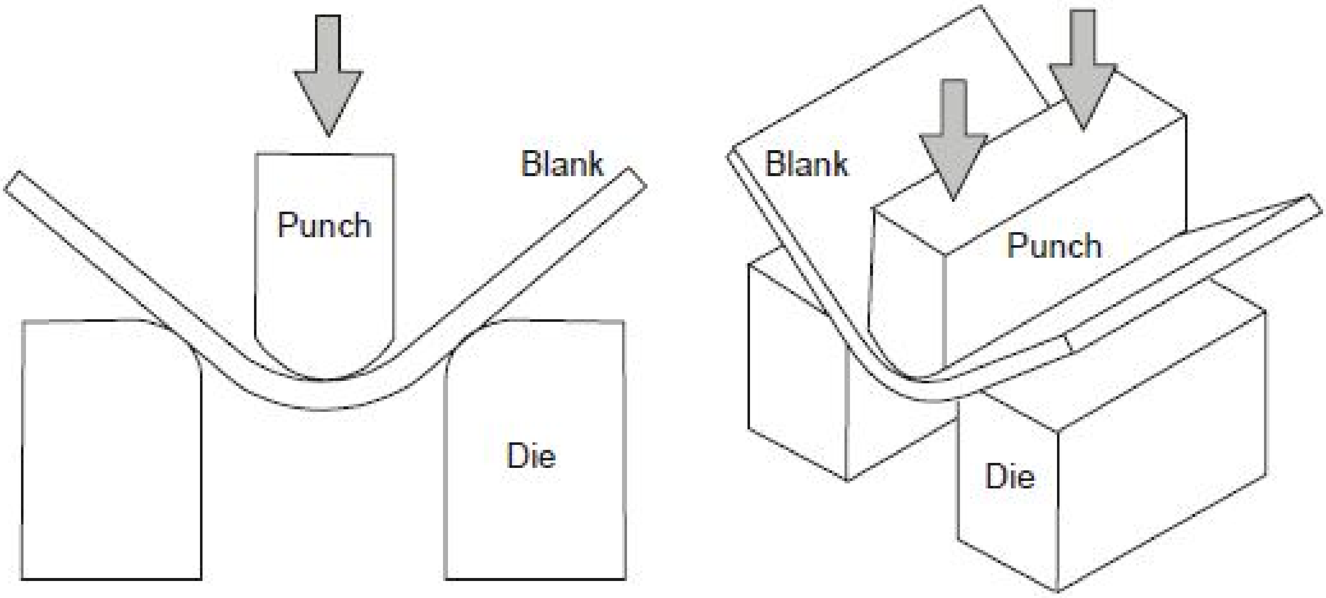
$$BA = \pi(R + KT)\ A/180$$

where:
BA = bend allowance
R = inside bend radius
K = K factor, which is t / T
T = material thickness
t = distance from inside face to neutral sheet
A = bend angle in degrees (the angle through which the material is bent)





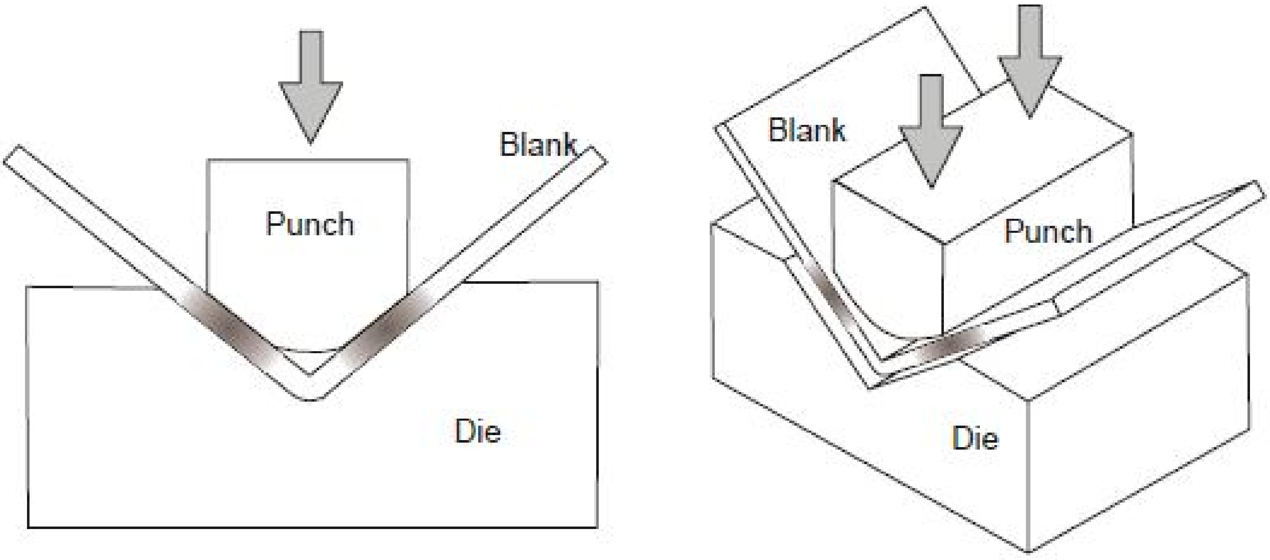
[illegible]



K-Factor Rule of thumb for Air Bending

Radius	Soft-material	Medium Material	Hard M
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

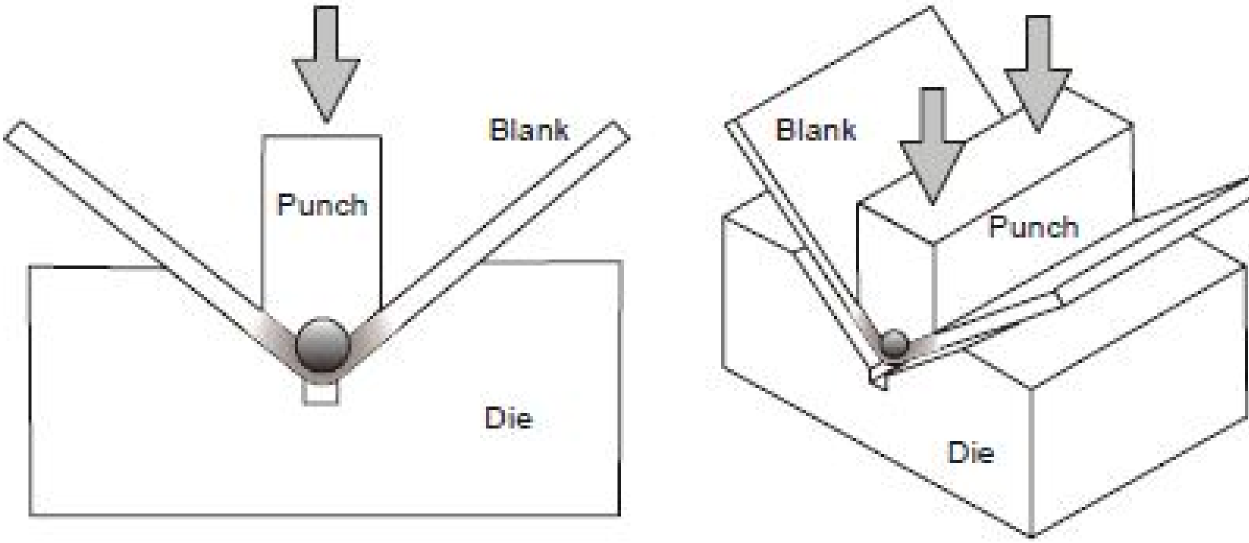
[illegible]



K-Factor Rule of thumb for Bottoming

Radius	Soft-material	Medium Material	Hard Mat
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

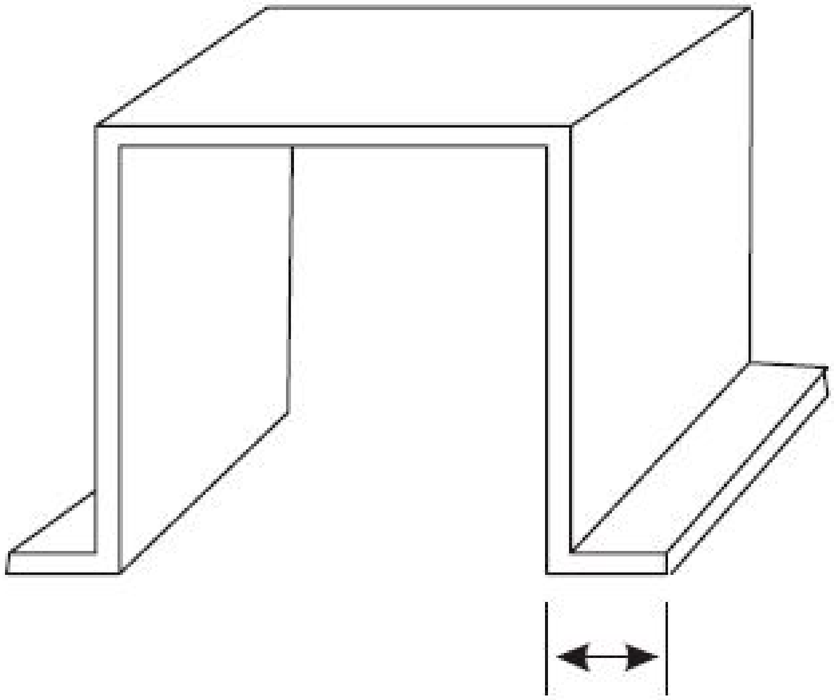
[illegible]



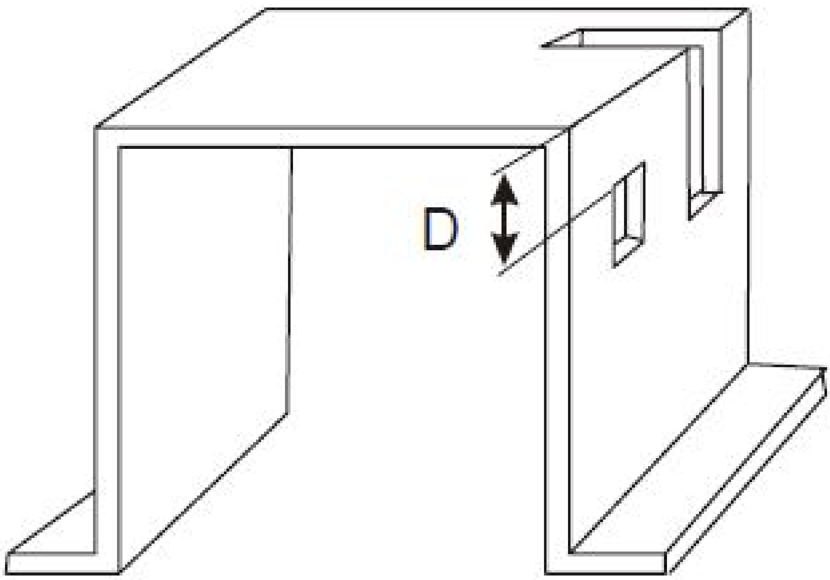
K-Factor Rule of thumb for Coining

Radius	Soft-material	Medium Material	Hard Ma
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

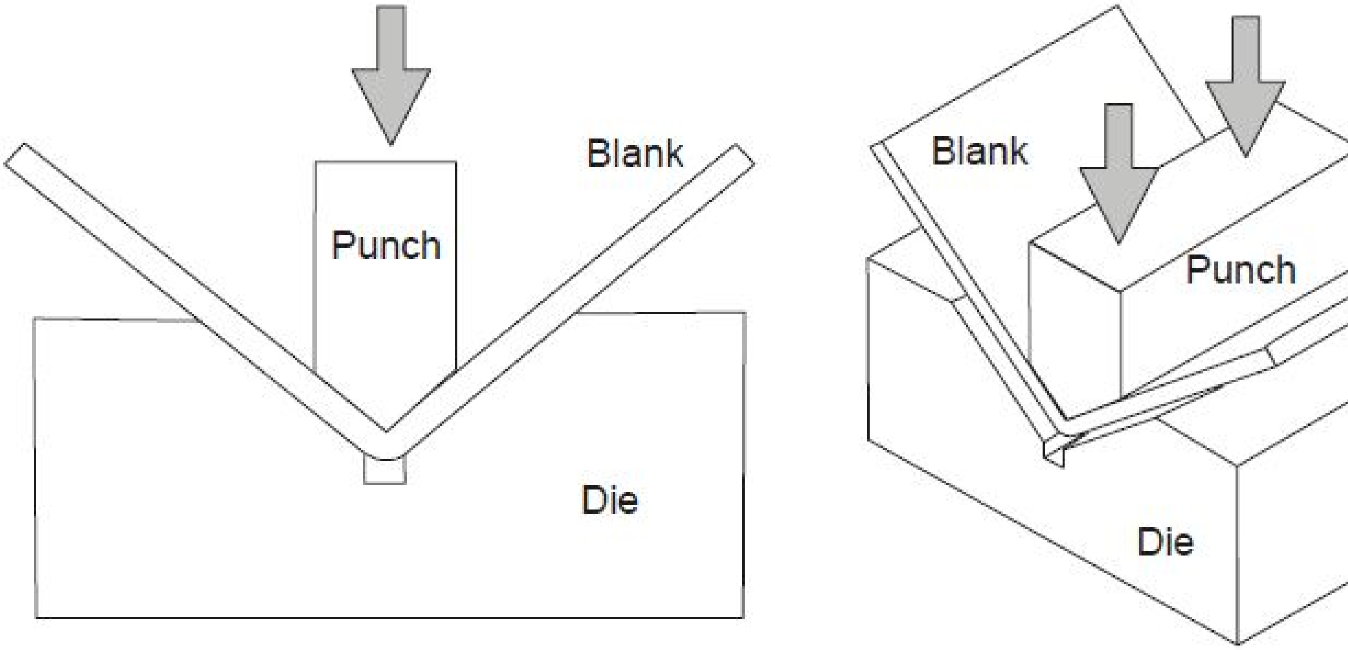
[illegible]



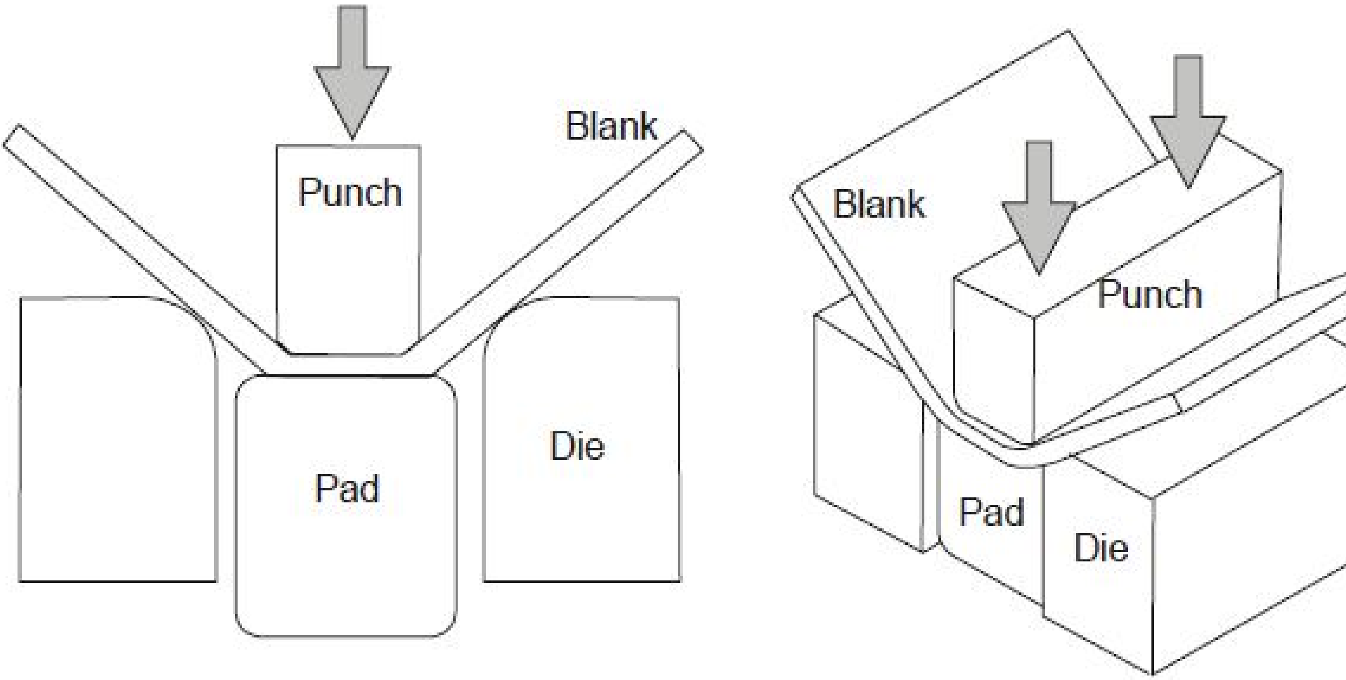
Min Flange Width = 4 x Thickness + Bend radius



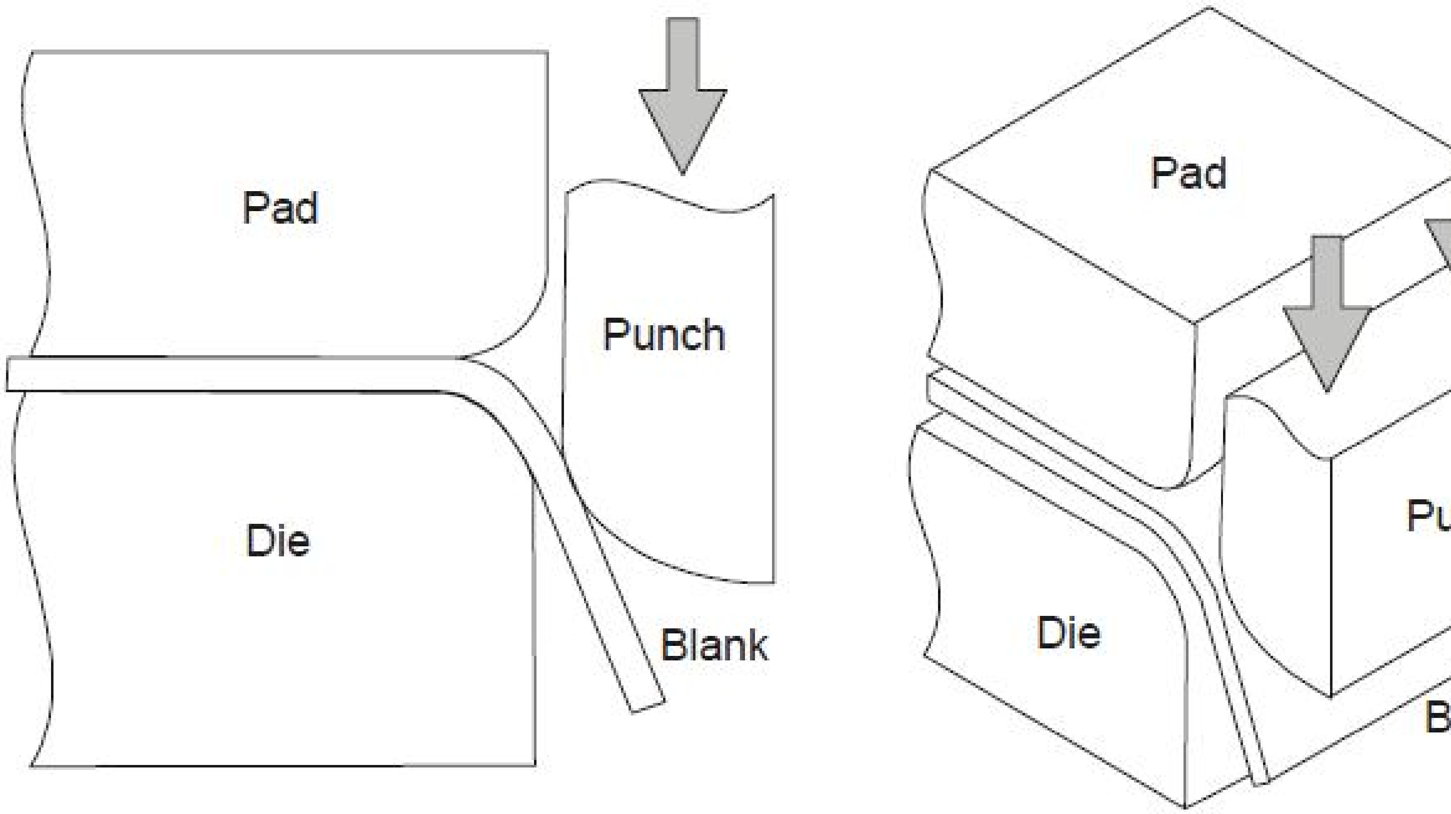
D = 3 x Thickness + Bend radius



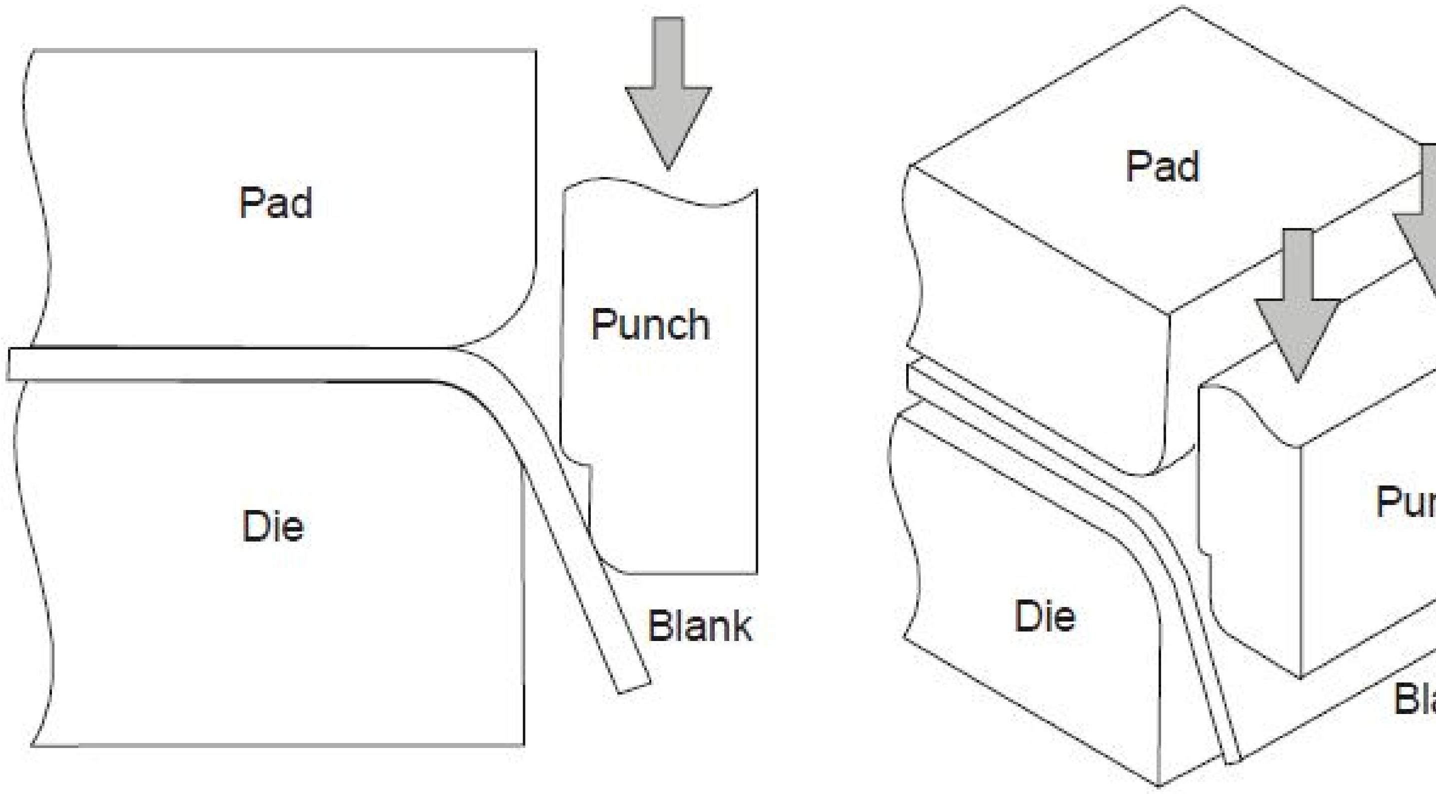
[illegible] It requires



[illegible] position

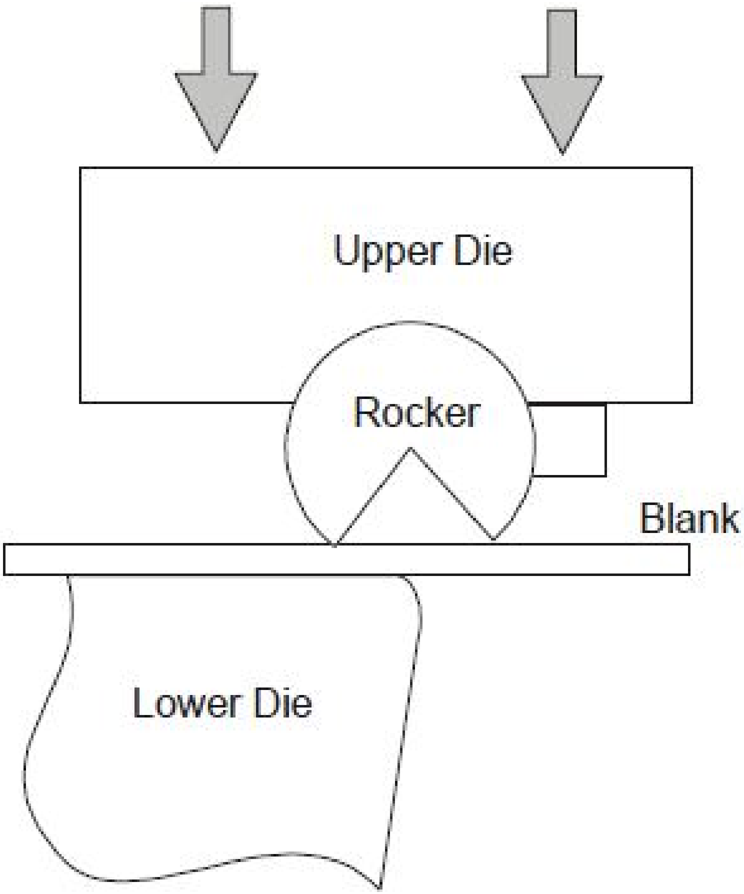


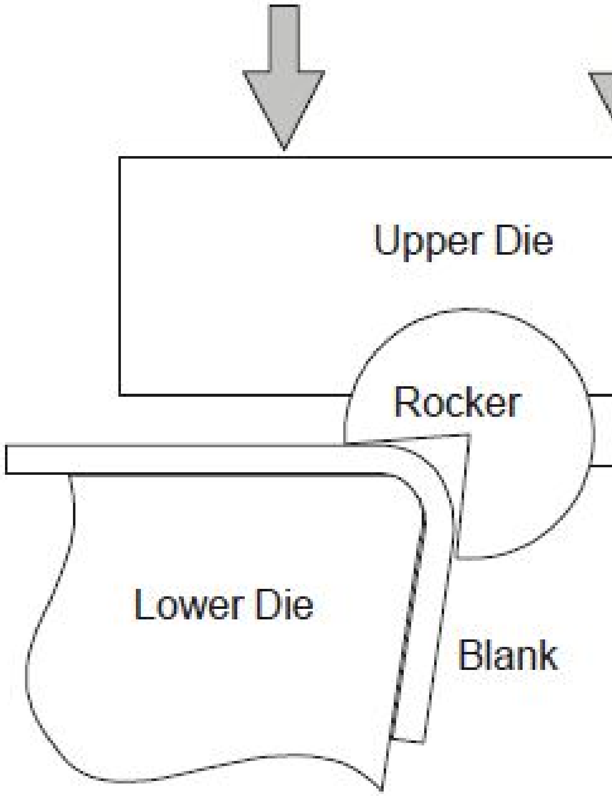
DOUBLE DIE BENDING ... workpiece one after



ROTARY BENDING ... bending process using a rocker instead of the punch. The advantages of

More than 90 degree bending angle is available





REFERENCES

Sheet Metal Fabrication

Tags: Sheet Metal, Sheet Metal Bending