BEND ALLOWANCE

Bend Deduction, Setback, Bend Allowance, these terms all across the country seem to have no consistent meaning. They are all used to mean the same thing or different things. I will define these terms by what is usually accepted by most sheet metal mechanics that I know. These terms may be used differently by your company but for the sake of consistency in this manual here are our definitions. By the way my favorite term so far is fudge factor. That seems to be what we are really doing.

BEND ALLOWANCE:

Sheet metal stretches minutely as it is being formed. The greater the inside bend radius is the less the stretching. Also material hardness or elasticity has a lot to do with how much it stretches. Therefore bend allowance is that dimensional adjustment required to allow for the forming of sheet metal. The mathematical expression most normally used is based on the empirical formula. Empirical means by trial and error. This is how the formula was discovered. They kept adjusting the numbers until they found a combination that consistently and satisfactorily worked out. Bend allowance and deduction formulas are used for air bending. Different results will occur if coining or bottom bending the material.

EMPIRICAL FORMULA:

Bend allowance = (.0078 x MT + .01743 x BR) x A°

MT = Material thickness
BR = bend radius to inside of material
A° = Angle from flat