

Sheet Metal Calculations

There are many different methods which sheet metal designers and shops employ to determine the flat length of sheet stock which will give desired dimensions on the finished bent part. Some of these methods are simply "rules of thumb" which individuals use based on experience. Often these rules of thumb relate to the material type and thickness, bend radius and angle, machine type, process speeds, etc. It really is somewhat of an art when the sheet metal process is implemented.

Computers on the other hand are very analytical in nature and therefore when a computer program simulates sheet metal bending or unbending it requires an analytic means by which to represent the process. Certainly specialized programs could be written for each individual shop to perform the calculations based on that shop's rules of thumb. However, most commercial CAD/Solid Modeling applications are for general use and therefore, provide a general solution. In most cases, such programs do provide a means by which you can give input to the calculations based on your experience and rules of thumb. This is certainly true for the SolidWorks sheet metal functionality.

There are two most widely accepted analytic representations of simple sheet metal bending. One representation is based on *bend allowance* and the other is based on *bend deduction*. The SolidWorks software as of this writing (up to SolidWorks 2000) employs the Bend Allowance method.

To give users an understanding of sheet metal calculations in general and how they are used in SolidWorks, this article will review the following:

- The definition of both bend allowance and bend deduction methods and how each of them relates to the actual sheet metal geometry.
- How bend deduction relates to bend allowance so users who employ the bend deduction method can easily convert their data to the bend allowance method.
- The definition of K-factor and how it is used in practice, including common ranges of values which are used for different material types.

Bend Allowance

To understand bend allowance, please consider Figure 1 which represents a single bend in a sheet metal part and Figure 2 which represents the flat form of that same part.

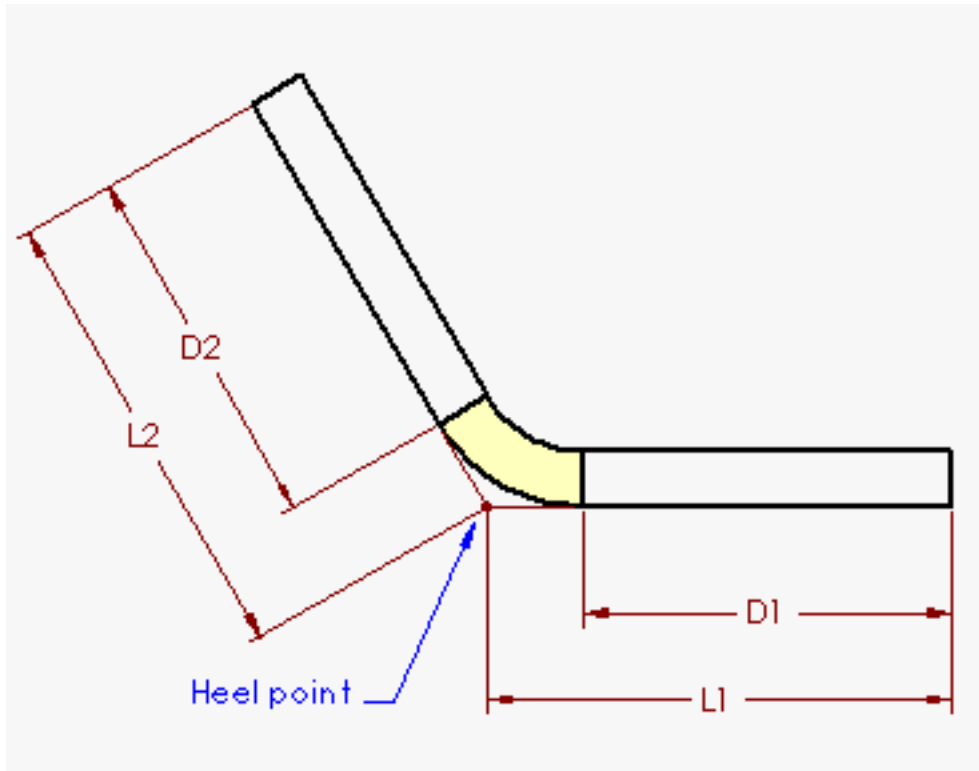


Figure 1

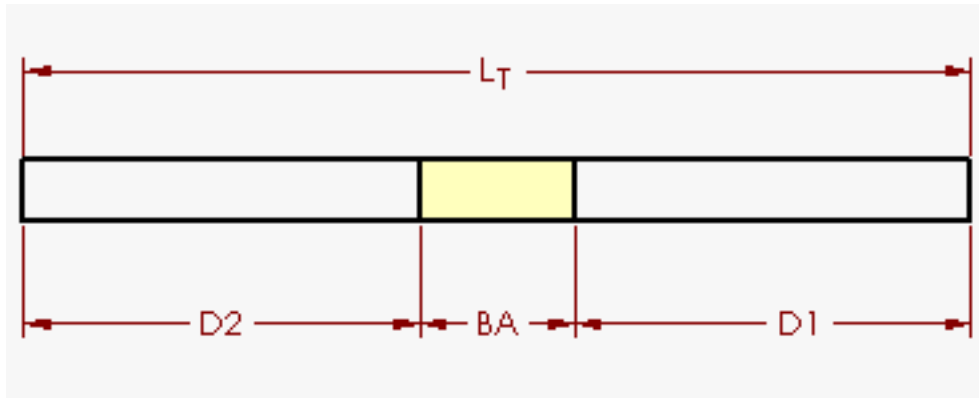


Figure 2

The bend allowance method describes the flattened length of the part (L_T) as the sum of the lengths of each of the flat portions of the part plus the length of the flattened bend region. The length of the flattened bend region is represented by the value "bend allowance" (BA). Therefore, the total length of the part can be represented by the equation:

$$L_T = D1 + D2 + BA \quad (1)$$

The bend region, represented as the yellow area, is the region in which all of the deformation theoretically occurs during the bend operation. In simple terms, to determine the geometry of the

flattened part you must:

1. Cut the bend region out of the bent part.
2. Put the two remaining flat sections down onto a table.
3. Calculate the length of the bend region after it is flattened out.
4. Glue the flattened bend region between the two flat sections. The result is the flat part.

The somewhat difficult part is determining what the flattened bend region length is, represented by the value BA. In general, the value BA will be different for each combination of material type, material thickness, bend radius and bend angle. Other factors which can cause the BA to vary are machine process, machine type, machine speed, etc.

Where does the value BA come from? There are quite a few sources for BA including sheet metal suppliers, experimental data, experience, and engineering handbooks. One resource is the Machinery's Handbook, Twenty-Fifth Edition. There is a good discussion of bend allowance and example tables for 90 degree bends on pages 1246-1250.

In SolidWorks, you have the ability to input BA explicitly, provide one or more tables with BA values, or use another means to calculate BA which is K-factor (which will be described in depth later). For all of these methods, you can enter the information to be the same for all bends in the part or you can explicitly set the information for individual bends if required. The bend table approach is the most accurate approach as you can specify different bend allowance values based on different thickness, radius and angle values. You generally will have one table for each material or each material/process combination. It can take a little bit of time to set up the tables initially, but once you have them, they can be re-used for each part. For more information on the format of SolidWorks [bend tables, click here.](#)

Bend Deduction

Bend deduction, also often referred to as setback (sometimes represented by setback divided by two), is simply a different means by which to represent the sheet metal bending process. Again referring to Figures 1 and 2, the bend allowance method refers to the flattened length of the part (L_T) as the sum of the lengths of the theoretical flat sections extended to the "heel point" (virtual intersection of the flat sides) minus the bend deduction (BD). Therefore, the total length of the part can be represented by the equation:

$$L_T = L1 + L2 - BD \quad (2)$$

Bend deduction is again a value which is determined from sheet metal suppliers, experimental data, experience, or handbooks with equations or tables for different materials.

How Bend Allowance Relates to Bend Deduction

Since SolidWorks uses the bend allowance method, it is important for users familiar with the bend deduction method to know how the two methods relate. It is fairly easy to derive an equation to relate the two values using the geometry of the bent and flat parts. So far, we have 2 known equations by the definition of each of the methods:

$$L_T = D1 + D2 + BA \quad (1)$$

$$L_T = L1 + L2 - BD \quad (2)$$

These equations can be combined to form:

$$D1 + D2 + BA = L1 + L2 - BD \quad (3)$$

Please consider the same geometry in Figure 1 with some additional labels as shown in Figure 3:

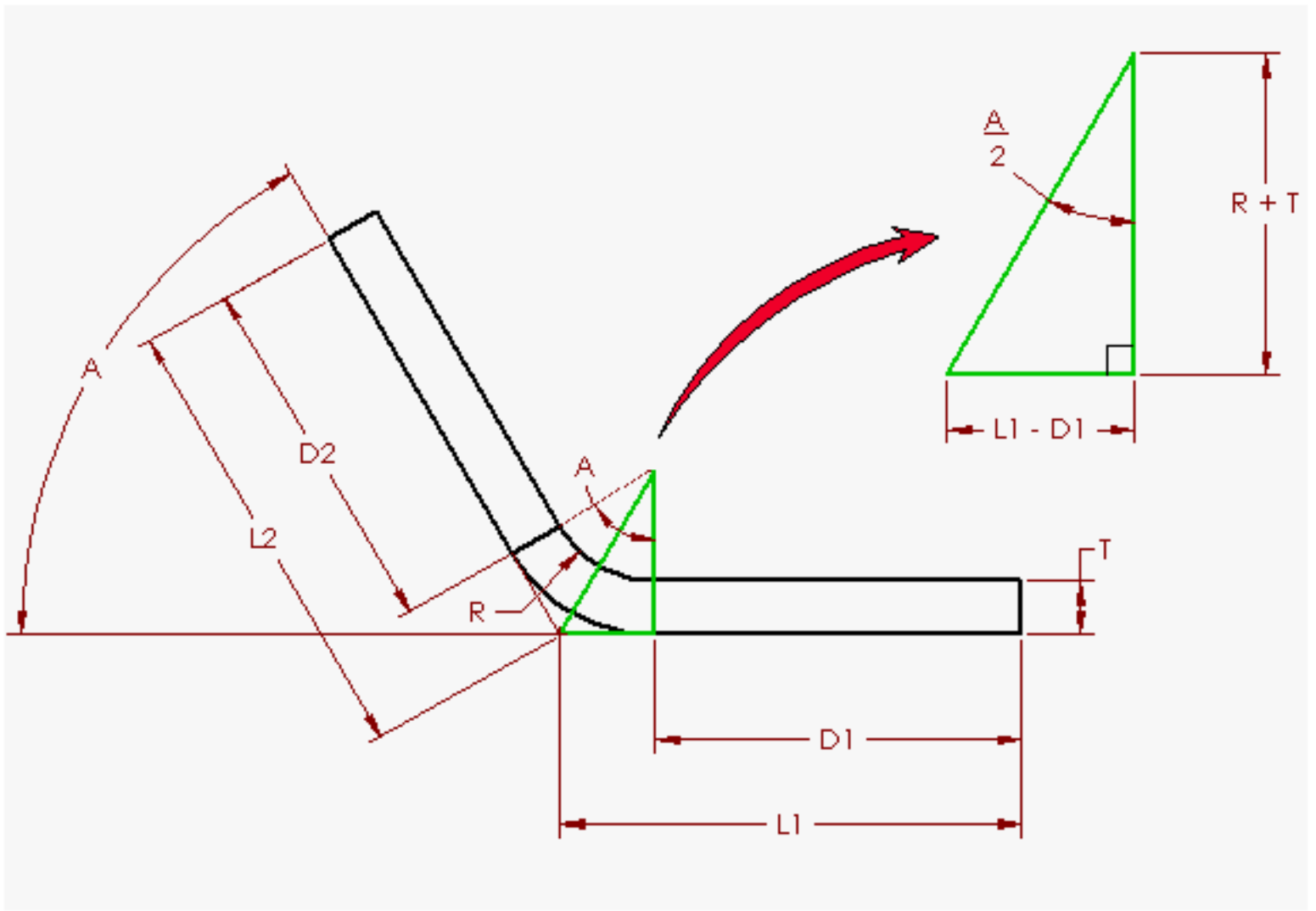


Figure 3

The angle A represents the bend angle or the angle which the part sweeps through when bending in degrees. This angle also describes the angle of the arc representing the bend region so it is shown in two locations in Figure 3. The inside bend radius is represented by R . Finally, T represents the thickness of the sheet metal part. A right angle triangle used to help form the relationship has been highlighted in green and re-labeled in Figure 3. Based on the labeled dimensions on this right triangle, we can derive the following equation:

$$\text{TAN}\left(\frac{A}{2}\right) = \frac{L1 - D1}{R + T}$$

Put in terms of $D1$, yields:

$$D1 = L1 - (R + T)\text{TAN}\left(\frac{A}{2}\right) \quad (4)$$

The same approach can be used on the other half of the bend region to derive D2:

$$D2 = L2 - (R + T)TAN\left(\frac{A}{2}\right) \quad (5)$$

Equations (3), (4), and (5) can be combined as follows:

$$L1 + L2 - 2(R + T)TAN\left(\frac{A}{2}\right) + BA = L1 + L2 - BD$$

Then it can be simplified to yield:

$$BA = 2(R + T)TAN\left(\frac{A}{2}\right) - BD \quad (6)$$

This equation can be simplified even further for 90 degree bends since $TAN(90/2) = 1$, yielding:

$$BA = 2(R + T) - BD \quad (7)$$

Equations (6) and (7) represent a method by which those users who are familiar with bend deduction values can convert those bend deduction values to bend allowance values based on the related thickness, bend angle and radius for use in SolidWorks. The bend allowance values can then be used for an entire part, individual bends, or to populate a bend table.

K-factor

K-factor is a single value which can be used to represent how sheet metal bends fold/unfold over a wide range of geometric parameters. It is a single value which is used to calculate the bend allowance (BA) over a range of material thickness', bend radii and angles. Figures 4 and 5 will be used to help define K-factor.

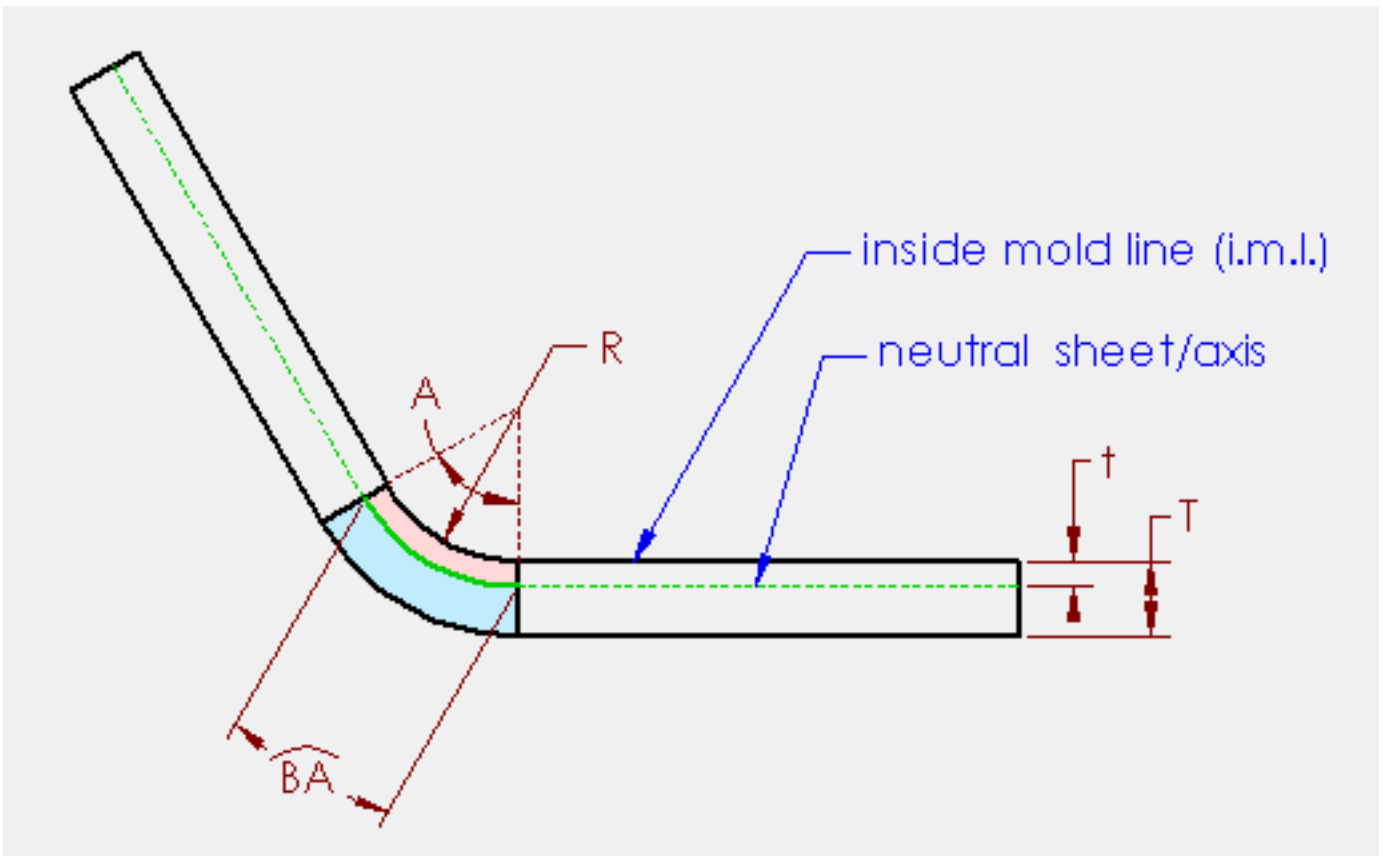


Figure 4

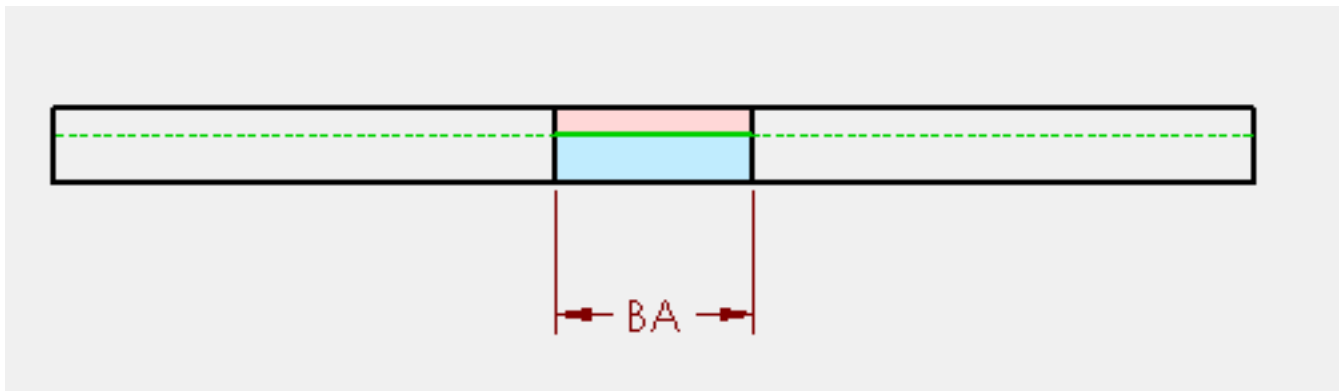


Figure 5

Consider a neutral sheet or axis which is defined as an imaginary section through the bend region where there is no stretching or compression. It represents the only location within the bend region which does not deform. This concept is represented by the pink and blue regions in Figures 4 and 5. The Pink region compresses during bending where the blue region stretches. If this neutral sheet does not deform, then the arc in the bend region along the neutral sheet has the same length when it is bent as it does when it is flat. Therefore, the value BA is equal to the length of the arc of the neutral sheet in the bend region. This arc is represented in Figure 4 by the green arc. The location of the neutral sheet is dependant upon the

specific material properties such as ductility. The neutral sheet location is expressed as "t", the distance into the sheet metal part from the inside mold line (i.m.l.). The radius of the neutral sheet arc can therefore be represented by (R + t). Using this expression and the bend angle, the neutral sheet arc length (BA) can be expressed as:

$$BA = \pi(R + t) \frac{A}{180}$$

To simplify the definition of the neutral sheet for use with all material thickness', the expression k-factor is used. K-factor is a ratio which represents the distance that the neutral sheet is into the sheet metal part as follows:

$$K = t/T$$

K is always a value between zero and one. A k-factor of 0.25 means that the neutral sheet is 25% of the way into the part, 0.5 means that the neutral sheet is 50% of the way into the part and so-on. Combining the previous 2 expressions, we get equation (8):

$$BA = \pi(R + K * T) \frac{A}{180} \quad (8)$$

This is the equation which is represented in the SolidWorks manuals and on-line help. The values A, R, and T are all dictated by the geometry, so where does the value K come from? Again, there are quite a few sources for K including sheet metal suppliers, experimental data, experience, and handbooks. In some cases, however, the value is not expressed as "K" and may not be in the same exact form as equation (8). However, even if the form is not exactly the same, you can generally find a relation. For example, the book McGraw-Hill Machining and Metalworking Handbook states that the neutral axis is "generally accepted as being located at 0.445 x material thickness inside the inside mold line". This translates to a k-factor of K=0.445. They further give the following equation:

$$\text{Bend allowance (B.A.)} = A (0.01745R + 0.00778T)$$

This equation is actually just a specific representation of equation (8) at K=0.445. To prove this, equation (8) is re-written below with the constants factored out and some variables re-arranged:

$$BA = A (0.01745 R + 0.01745 K * T)$$

Comparing this equation to the McGraw Hill equation, you can see that 0.01745 K=0.00778 or K=0.445.

Similarly, the Machinery's Handbook Twenty-Fifth Edition uses similar representations. This book uses

the term "L" instead of BA, but it represents the same value. Page 1246 lists the following equations for 90 degree bends:

For soft brass and soft copper:

$$L = (0.55 * T) + (1.57 * R)$$

For half-hard copper and brass, soft steel and aluminum:

$$L = (0.64 * T) + (1.57 * R)$$

For bronze, hard copper, cold-rolled steel, and spring steel:

$$L = (0.71 * T) + (1.57 * R)$$

Simplifying equation (7) for 90 degree bends and factoring out constants and rearranging variables gives the following expression:

$$BA = (1.57 * K * T) + (1.57 * R)$$

Therefore, for soft brass and soft copper, according to the Machinery's Handbook equation, $1.57 K = 0.55$ or $K = 0.35$. The same concept can be applied for the other material types listed such that you get the following three values:

For soft brass and soft copper:

$$K = 0.35$$

For half-hard copper and brass, soft steel and aluminum:

$$K = 0.41$$

For bronze, hard copper, cold-rolled steel, and spring steel:

$$K = 0.45$$

As discussed earlier, there are quite a few sources for K including sheet metal suppliers, experimental data, experience, and handbooks. If you are to use K-factor to drive your sheet metal models, you must find a source or values for K which meets your engineering requirements and gives you results which match your physical part results within your desired accuracy. You can see that the two reputable sources here list two different possibilities for K. One source gives a single value for all materials where

the other source breaks it down into three material categories. In some cases, using the K-factor approach may not give you accurate enough results across your entire range of input values as it is only one number representing a wide range of possible bending situations. In such cases, to achieve more accurate results, you should use BA values directly for the entire part, individual bends, or use a bend allowance table to represent the BA values across the entire range of A, R, and T. You can even use equations to generate the data for bend tables like the Machinery's Handbook has on pages 1247-1249. If needed, you can then tweak individual cells of the table based on experimental data or experience.

For further information on sheet metal calculations for bending and unbending, please consult the following resources:

Machinery's Handbook Twenty Fifth Edition, Industrial Press Inc., New York, 1996, pgs. 1246-1251.

McGraw-Hill Machining and Metalworking Handbook, New York, 1994, pgs.1200-1204.

"Precise Bend Allowances Equal Quality Parts", Welding Design & Fabrication, July 1996, pgs. 21-26.

"CAD/CAM Software for Stamping Die Design", The Fabricator, October 1996, pg. 90.