Sheetmetal Drawings

Many objects, such as cardboard and metal boxes, cake pans, mail boxes, HVAC ducts, roof gutters, are made from flat sheet material that, when folded, formed, or rolled, will take the shape of an object. Since a definite shape and size are desired, a regular orthographic drawing of the object is made first: then a development drawing is made to show the complete surface or surfaces laid out in a flat plane. Sheetmetal or surface development drawings are sometimes referred to as pattern drawings.
**Beads:**
Raised or depressed areas in sheet-metal parts, usually provided for purpose of rigidity.

**Bend Allowance:**
Length of material required for a bend from bend line to bend line.

**Bend Angle:**
Full line through which metal is bent: not to be confused with angle between flange and adjacent leg.

**Bend Line:**
Tangent line where bend changes to a flat surface. Each bend has two bend lines.

**Blank:**
A flat sheet metal piece, approximately correct size, on which a pattern has been laid out and is ready for machining and forming.

**Center Line of bend:**
A radial line, passing through bend radius, which bisects included angle between bed lines.

**Developed Length:**
Length of flat pattern layout. This length is always shorter than the sum of mold line dimensions on part.

**Flat Pattern:**
Pattern used to lay out sheet metal part blank.

**Leg:**
The flat or straight section of a part after bending or forming.

**Mold Line:**
A line of intersection formed by projection of two flat surfaces.

**Setback:**
Amount of deduction in length resulting from a bend to be developed in a flat pattern.

**Straight-Line Development:**
The term given to the development of an object that has surfaces on flat plane of projection (Flat Pattern).

**Bend Relief Cutouts:**
When bends intersect, a relief (notch) must be cut out to prevent the part from buckling or wrinkling.
Here you are given some of the most common sheet-metal hems and joints used in the industry. They are used to bend single sheets of sheetmetal or join to pieces of sheet-metal together to form a bond.

Here is you are given a typical sheet-metal call out you might see on a drawing. Each part of the call out is broken down into its individual meaning.

- **GAGE #**
- **TYPE OF GAGE**
- **SHEETMETAL THICKNESS**
- **DEVELOPE WIDTH**
- **DEVELOPE LENGTH**

#16 USS (.063) X 12.50 X 26.00
When sheet metal is bent, the medium line along the interior of the metal remains true length throughout the bending process. The metal on the outside of this line is stretched and the metal on the inside of the bend is compressed. For most metals, the median line is approximately 44 percent of the distance from the interior face. The location of the median line forms the basis for the bend allowance calculations. The harder the metal, the greater the bend length. In actual practice some experimentation may need to be done with the different metals and the different shipments of the same metal alloy to arrive at the correct allowance.

Sheetmetal is usually dimensioned using the undirectional method of dimensioning. Dimension are read from top to bottom and from left to right. Under the undirectional method of dimensioning two types of dimensioning styles may be used, they are the **linear**, and **datum** styles. The linear dimension style, dimensions are displayed horizontal and vertically on the object. In the datum dimensioning style all dimensions are taken from a 0 reference point and may be used by a CNC machine to drill and cut holes into the sheetmetal.