CADALYST SALUTES 15 HOT NEW PRODUCTS

NOW INCLUDING CADENCE
  - LYNN ALLEN
  - CIRCLES AND LINES
  - ROBERT GREEN
  - MANAGER'S VPOINT

CADALYST LABS REVIEW AUTODESK BUILDING SYSTEMS 2004 TACKLES ALL AEC ENGINEERING TASKS

- STEP/IGES TRANSFERS: TIPS FOR SUCCESS
- NEW TOOLS DELIVER BIM DATA DOWNSTREAM
- BONUS TUTORIAL SHEET-METAL FLAT PATTERNS
Few CAD users seem to know how to create a sheet-metal flat pattern with proper bend allowance. Except for those that use 3D CAD software that automatically creates sheet-metal flat patterns, most companies have their engineering departments create a dimensioned drawing of formed sheet-metal parts and leave the flat-pattern calculations to the shop.

The manufacturing trend for sheet-metal parts, especially for prototypes and small production runs, is to use computer-controlled laser or water-jet cutting machines. These precision machines cut with outstanding accuracy, and they can import CAD geometry for a cutting pattern.

I formulated a method to produce accurate sheet-metal flat patterns using AutoCAD to perform the calculations while constructing geometry. This method is ideal for small production runs of accurately cut and formed parts without a custom-made die. However, it works for any sheet-metal flat pattern, even if total precision isn’t needed.

One machinist at a shop I contacted was skeptical. He had received flat patterns from customers before but was never able to use them. After reviewing my geometry and finding nothing wrong with it, he used my flat pattern. He later informed me that the part formed perfectly. Without my flat pattern, he would have formed the part in two pieces that would then be joined with 9” of welding. My pattern required a total of 10” of welding.

With this method, you construct an end view of the formed part in AutoCAD. Then the inside radius of each bend is offset to define the neutral axis of the bend. To begin with, let’s review how to calculate an offset value.

**FIGURE OUT THE MATH**

The Machinery’s Handbook lists the following formulas to calculate sheet-metal bending allowance, where \( L \) = length of straight material required to form the bend, \( T \) = thickness of sheet metal, and \( R \) = inside radius of bend:

\[
L = (0.55 \times T) + (1.57 \times R)
\]

for 90° bend in soft brass/copper.

\[
L = (0.64 \times T) + (1.57 \times R)
\]

for 90° bend in half-hard brass, copper, soft steel, aluminum.

\[
L = (0.64 \times T) + (1.57 \times R)
\]

for 90° bend in hard copper, bronze, cold-rolled steel, spring steel.

You may recognize 1.57 as 0.5 of \( \pi \). This isn’t a coincidence. As you’ll remember from your first geometry class, the formula for finding the circumference of a circle is \( C = 2\pi \times R \), where \( C \) = circumference and \( R \) = radius.

Because a 90° bend forms an arc equal in length to 0.25 of the circumference of a circle, \( C/4 = (2\pi \times R)/4 \), and 0.25 = 0.5 \( \pi \) \( \times \) R. Each formula includes 1.57 \( \times \) R to find the length of a 90° arc. The portion of the formula that varies with metal type and hardness compensates for changes in the metal during bending. Remember that metal near the outside radius is stretched when bending, while metal near the inside radius is compressed. Softer metals tend to stretch more easily, which explains the different factors in the formulas for different metals.

Using the proper formula returns the length of a straight section of sheet metal equal to the arc length. Observe that the arc defined by the inside radius of the formed part is less than the straight length, while the arc defined by the outside radius is greater than the straight length. A plane through the thickness of a flat piece of sheet metal where no stretching and compressing occurs while bending is known as its neutral plane.

Because a plane is defined as a flat surface, and the neutral plane is not flat after bending, the location of the neutral plane in a curved part is referred to as the neutral axis. The arc length through the neutral axis after bending and the flat length of the sheet metal before bending are equal. The differences between the outside radius and flat length and the inside radius and flat length are not equal. This means that the neutral axis is not in the center of the material. It’s closer to the inside radius—the softer the material, the nearer its neutral axis is to the inside radius.

The formulas are simple to use, but what if you need to calculate a bend allowance for a bend other than 90°? For a 45° bend, you use the appropriate formula and divide the result by two. For a 22.5° bend, multiply the result by 1.5. Luckily, we have calculators.

If you define the neutral axis relative to the inside radius of the bend, you can create AutoCAD geometry and have AutoCAD find the proper bend allowance calculations.

First, let’s examine the second formula, as it applies to more commonly used general-purpose materials:

\[
L = (0.64 \times T) + (0.5\pi \times R)
\]

using 1" for the sheet-metal thickness and 1" for the inside radius, so \( L = (0.64 \times 1) + (0.5\pi \times 1) \).

Performing the multiplication of both thickness and inside radius by 1 leaves: \( L = 0.64 + 0.5\pi \), and then \( L = 2.21 \).

It takes a straight section 2.21" long to form a 1" inside radius bend in 1"-thick sheet metal.

Remember that the straight length and arc length through the neutral axis are equal. Use the formula:

\[
A = 0.5\pi \times R
\]

where \( A \) = length of arc through neutral axis (and straight length) and \( R \) = radius of arc through neutral axis.

If 2.21 = 0.5\pi \times R, then 2.21/0.5\pi = R and 2.21/0.5\pi = 1.407.

The radius to the neutral axis is 1.407", and when you subtract 1" (the inside radius of the bend), you’re left with 0.407—an offset factor that defines the neutral axis relative
to the inside radius. For a bend in half-hard metal, such as brass, copper, soft steel, and aluminum, a line offset from the inside radius by 0.407 of the thickness defines the neutral axis.

**HOW TO USE THE OFFSET FACTOR**

The procedure to use the offset factor may sound complicated but is extremely simple in practice. As it depends on properly constructed geometry, start by constructing an end view of the formed part. In the example, I use a thickness of 0.25" so the lines don't appear so close together. Here I also use an inside radius equal to the sheet-metal thickness, because it's the minimum recommended.

First, draw lines that represent one edge of the end view of the sheet-metal part—draw all lines without any radii at intersections (figure 1a).

Then use the Offset command and enter the sheet-metal thickness for the offset distance. Offset the lines drawn in the first step. Pay attention to offset the correct side of the line to maintain desired overall dimensions (figure 1b).

Next, use the Radius command and set the radius to either the inside or outside radius of the part. I usually dimension an outside radius, so I'll use 0.5" here. Make sure that Trim is turned on and radius the lines that represent all outside edges of the part. End the Radius command (figure 1c).

Restart Radius and set the radius to the desired inside radius for the part. Instead of typing a value, select Center Snap and click on one of the outside radii you just created. Now select Perpendicular Snap and click on an inside edge near the selected radius. This sets the radius relative to the geometry you constructed. Use Radius on lines that represent inside edges (figure 1d).

Note that lighter gauge sheet-metal thickness is not an even fractional value: 11-gauge steel is 0.1196" thick, not 0.125" thick. I find it easier to set the radius relative to geometry already constructed rather than punch numbers into a calculator. Because this method creates a flat pattern relative to existing geometry, errors may snowball in geometry you construct in later steps. Of course, in this example, both the outside and inside radii are even fractional values—0.5" and 0.25". I chose this thickness to separate the inside and outside edges for clearer illustrations. With lighter gauge sheet metal, the inside radius won't be an even fractional value. Sometimes it's desirable to dimension the inside radius and set the outside radius.

When the edge view is properly constructed, both the inside and outside edges of any straight section are the same length. Verify this with the Distance command. All that remains is to draw lines from endpoint to endpoint, closing the ends (figure 2).

Use the Line command, select midpoint snap, and then click on the inside edge of a straight section. To complete the line, select Perpendicular Snap and click on the outside edge of the straight section. You should have a line drawn through and equal to the thickness of the sheet metal.

Use the Scale command and select the line just drawn through the thickness of the sheet metal. For the scale origin, select the endpoint of the line being scaled at the inside edge of the part. For the scale factor, type 0.407 (figure 1d).

Next, use the Offset command, but instead of typing a value, select Endpoint Snap and click on one end of the scaled line through the thickness of the part. Select Endpoint Snap again and click on the other end of the line. This sets the offset value to 0.407 X the metal thickness—the length of the line just scaled. Now, offset all arcs that represent inside radii (figure 1d).

Use the Polyline command on each of the arcs you just constructed. You'll see a Command line prompt:

**Object selected is not a polyline. Do you want to turn it into one? <Y>**

Press <Enter> to accept the default to convert each of the offset arcs into polylines.

Construct the flat pattern by using the Offset command and setting the offset value relative to the polyline arc lengths you created and that are relative to line segment lengths in the end view already constructed. You must start with a line segment at one end of the end view and progress in order to the adjacent polyline arc, then to the next line segment, then to the next polyline arc, and so forth.

First, draw a line that represents the width of the part. Offset the line, but instead of typing a value, select Endpoint Snap and click on the end of the
first straight line segment at one end of the end view already constructed. Then, again select Endpoint snap and click on the opposite end of the line segment. This sets the offset value to the length of the line. Next, select the line to offset and the side to offset to. End the Offset command.

Next, use the List command on the polyline arc adjacent to the line used for the previous offset. When the text box displays, note the length of the arc. Then use Offset, but instead of typing a value, type `cat`. When prompted again, type `getvar` (perimeter) and press <Enter>.

This sets the offset value to the length of the polyline arc. You can observe this on the Command line. Does it match the value reported in the text box? If it doesn't match, make sure that you converted the arc to a polyline, because the procedure works only with polylines. Offset the line previously created with the Offset command. End the Offset command.

Use the Offset command again, but set the offset value to the length of the next line segment in the end view. End the Offset command.

Continue offsetting lines relative to polyline arc lengths and line segment lengths until you reach the last line segment at the opposite end of the end view.

Draw lines to connect the ends of the lines at the extreme edges of the flat pattern. Then change the linetype of all interior lines of the flat pattern to phantom lines. To complete the flat pattern, draw a line from midpoint to midpoint of the phantom lines to define the length of a curved section on the flat pattern.

Use the Offset command with the offset value set to through, and offset a line through the midpoint of the lines just drawn. Change the linetype of these newly offset lines to center lines.

To use this procedure for different materials, perform the same steps, but use the appropriate offset factor. Manipulating the formulas found in the Machinery's Handbook returns the offset values for various materials. For example, for bends in soft brass and copper, the offset value is 0.350; for half-hard brass, copper, soft steel, and aluminum, it's 0.407; and for hard copper, bronze, cold-rolled steel, and spring steel, it's 0.452.

**COMPLETE THE FLAT PATTERN**

All that remains is to dimension the flat pattern and delete temporary lines and arcs. You use only dimensions to the center of bends. The dimensions along the top of the illustration show relationships of lengths of flat and curved sections of the part (figure 3). In the past, a close sheet-metal tolerance was 0.03215" (1/32"). With today's laser and water-jet cutting machines, parts can be cut with a tolerance of 0.005". This method is great for small quantities of custom parts or for a full load of parts.

---

*Andy Wendt (www.home.com, Andy.Wendt) is an independent contractor and consultant. He is an electrical and mechanical designer, AutoCAD manager, and technical writer and illustrator of equipment operations manuals. Send comments to editors@cadalyst.com, andywendt@bellsouth.net www.bellsouthpwp.net/a/w/awendt*