Chapter

2

Designing with Sheet Material

In this chapter, you will learn the following to World Class standards:

- **Sheet Metal and Stampings**
- **Calculating the Bend Allowance in Material**
- **Making a Drawing for a Stamped and Formed Panel**

Sheet Metal and Stampings

Sheets and plates are in use in many designs. Sometimes we have a hard recognizing the sheet material in the design, because the flat material is shaped by a press that stamps the part making the object look as if it was molded. We can bend many materials into different shapes. Some of the most common materials that are formed into usable shapes are aluminum, steel, and the plastic polycarbonate. Most likely the case that holds the mother board, power supply, hard drive and other digital devices inside our computer has outside panels that are formed sheet steel.

A characteristic of a stamped part is the thickness of the sheet itself. Most enclosures that hold electrical devices such as the computer case or a metal skin that protects us from heat like on the outside of a furnace are using sheet steel to prevent injury from accidental contact. There are many devices inside our homes and businesses that we utilize safely every day without thought of harm, because they have a protective skin which prevents us from contacting the dangerous items inside and hurting ourselves. We touch the outsides of appliances like washers and microwave ovens everyday not thinking of the dangerous machines inside them, since we are prevented from access mainly by sheet components.

If the thin material is able to protect us, another trait of sheet made products is built in rigidity, which is the feature that prevents the product from bending. Many times, designers will form the edges of a panel to remove a sharp edge and to give an amount of stiffness to an exterior cover. Appliances have motors, fans and transformers, which can cause vibration in the housing to which these machines are mounted. Since we connect the protective sheet metal panels to the same frame that holds the vibrating mechanisms, the amount of rigidity designed in the stamping is also helpful to prevent the part from creating a large amount of noise through vibration.

In Figure 2.1, we see an exterior cover made from sheet metal where this part is made easily by stamping or cutting the 24 inch square from a larger sheet. Many sheet metal parts are made from thin material that is measured in the industry using a gauge system. Let us use 18-gauge (0.048) for this cover plate, but the edges will be very sharp and the technicians or consumers will need to wear gloves when handling the steel cover, unless we change the design.

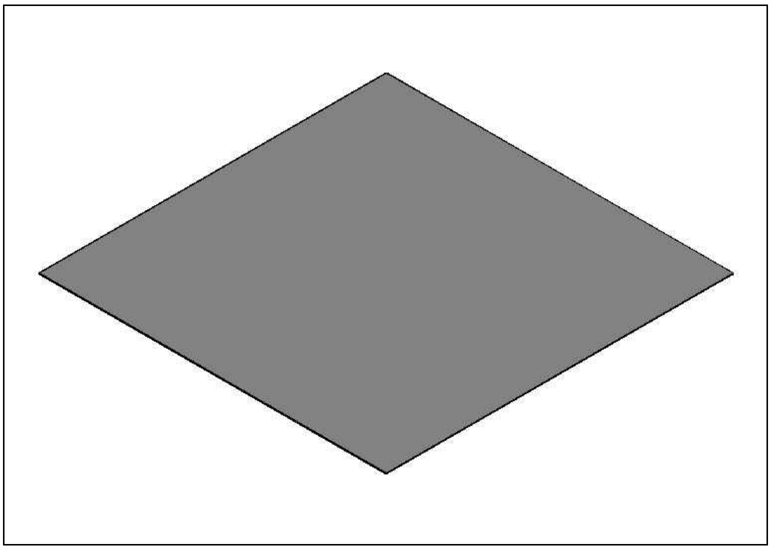

Figure 2.1 – Sheet Metal Part without any Bends

We decide to form the panel to make the sheet metal part functional. First, we as designers need to be familiar with two pieces of information, which are the gauge thickness and bend allowance. When we search for the sheet steel, we will use a gauge chart as shown in figure 2.2, which displays steel sheet thickness, galvanized steel sheet thickness and aluminum sheet thickness. When the sheet thickness approaches ¼ of an inch, individuals in the industry many times will refer to the material as plate. We will probably never see 1/4, 3/8, 1/2, 3/4 or even 1 inch plate steel in a automobile or any other consumer products, but those designers who support heavy industries will use the thicker steel plates for various designs requiring additional strength.

Gauge (ga)	Standard Steel Thickness (inches)	Galvanized Steel Thickness (inches)	Aluminum Thickness (inches)
3	0.2391		0.2294
4	0.2242		0.2043
5	0.2092		0.1819
6	0.1943		0.1620
7	0.1793		0.1443
8	0.1644		0.1285
9	0.1495	0.1532	0.1144
10	0.1345	0.1382	0.1019
11	0.1196	0.1233	0.0907
12	0.1046	0.1084	0.0808
13	0.0897	0.0934	0.0720
14	0.0747	0.0785	0.0641
15	0.0673	0.0710	0.0571
16	0.0598	0.0635	0.0508
17	0.0538	0.0575	0.0453
18	0.0478	0.0516	0.0403
19	0.0418	0.0456	0.0359
20	0.0359	0.0396	0.0320
21	0.0329	0.0366	0.0285
22	0.0299	0.0336	0.0253
23	0.0269	0.0306	0.0226
24	0.0239	0.0276	0.0201
25	0.0209	0.0247	0.0179
26	0.0179	0.0217	0.0159
27	0.0164	0.0202	0.0142
28	0.0149	0.0187	0.0126
29	0.0135	0.0172	0.0113
30	0.0120	0.0157	0.0100
31	0.0105	0.0142	0.0089
32	0.0097	0.0134	0.0080
33	0.0090		0.0071
34	0.0082		0.0063
35	0.0075		0.0056
36	0.0067		
37	0.0064		
38	0.0060		

Figure 2.2 – Sheet Metal Gauge Table

In Figure 2.3, we show material that is ½ inch in depth folded up on all four edges of the exterior panel. As a designer, we will need to design a blank, which is the flat cut out of material from the larger piece of sheet metal that allows for the lips to be bent up 90° into position as shown. Placing all four formed edges on the exterior panel will increase the rigidity of the panel and reduce the amount of vibration and noise in the sheet.

Most fabricators of sheet metal parts will require a minimum bend radius on formed objects. The minimum bend radius should be a minimum of ½ of the material thickness, so if the material is 18-gauge or 0.048 thick, then the inside radius would be 0.024, making the outside radius 0.072. By maintaining the minimum radius, the material is less likely to tear on the outside radius which would weaken the part. Also, the customer finds that cracks in the metal are not aesthetically pleasing to the overall product appearance.

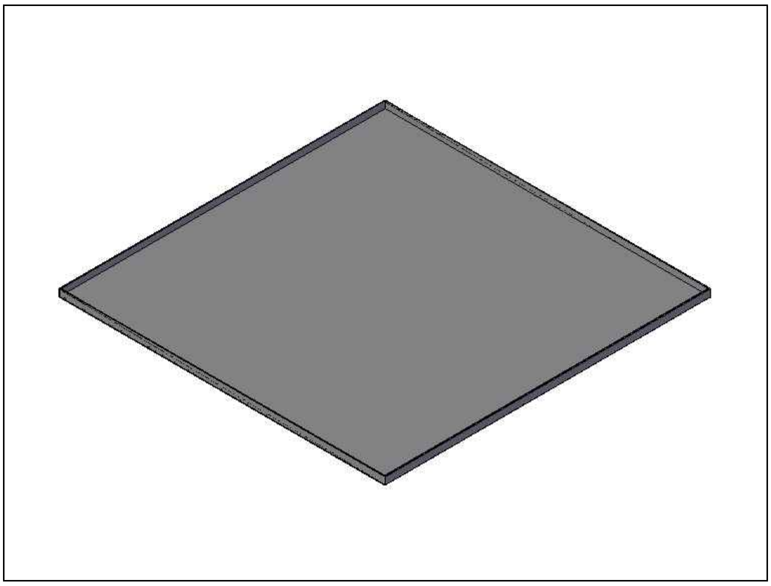

Figure 2.3 – Shaped Sheet Metal Part

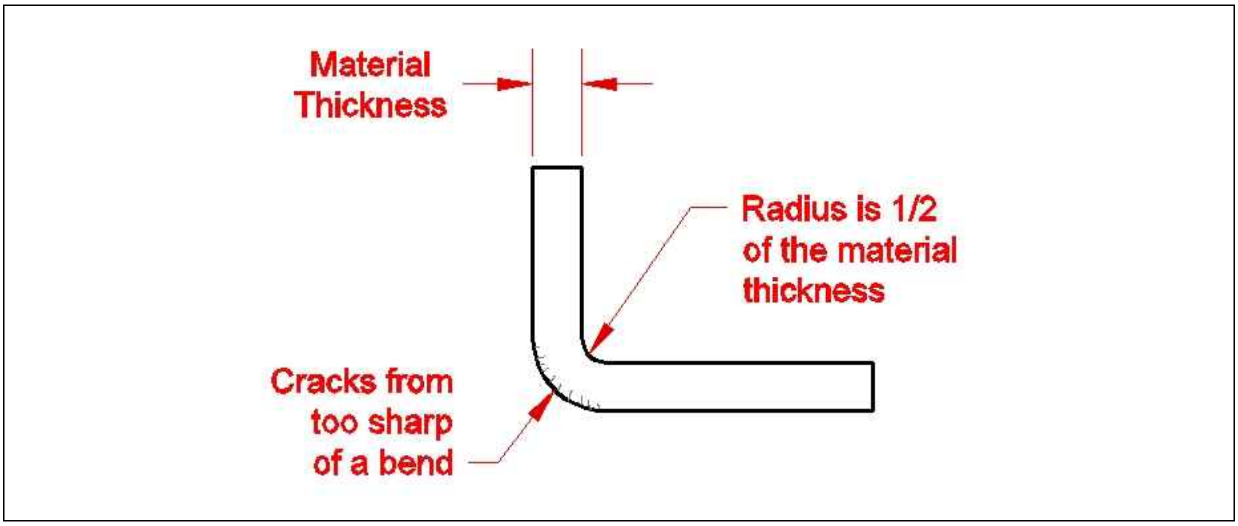

Figure 2.4 – Minimum Bend Radius

Calculating the Bend in Material

Before we calculate the material for the 24 x 24 inch square sheet metal cover, we will determine the material we need when we bend ½ inch of material up on the part shown in figure

2.5. We will place an inside bend radius of 0.025 one on this part.

When we bend metal, we think of the arc going through the bend radius where the sheet material is neither in tension nor in compression as the neutral axis. To find the true length of the material for the part in figure 2.5, we need to obtain three measurements for a part that has one bend. We will subtract the material thickness and the inside radius from the 0.5 side, which gives us 0.427 inches. We will subtract the material thickness and the inside radius from the 1.5 side, which gives us 1.427 inches.

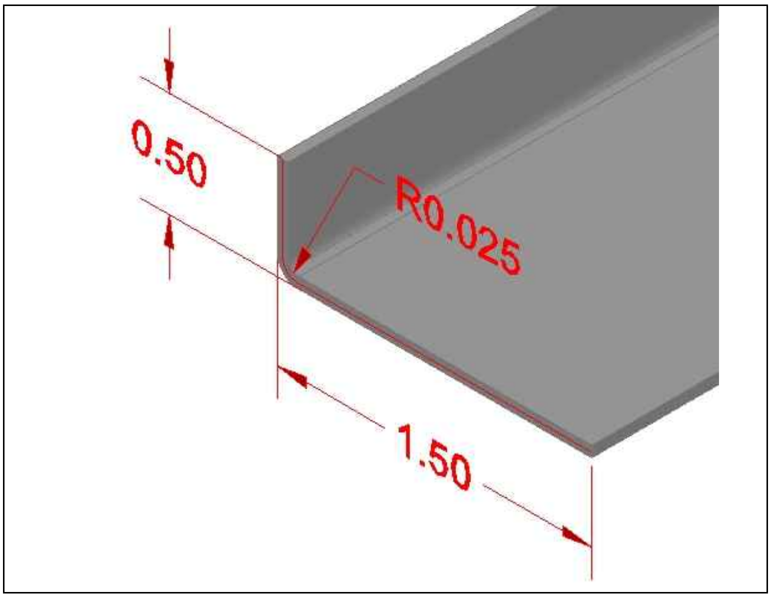

Figure 2.5 – Simple L-Bracket Part

<u>Bend Formula for a L-bracket</u>

Step 1: Length of Side 1= Bend Length – Material Thickness – Inside Bend Radius

0.50 – 0.048 – 0.025 = 0.427

Step 2: Length of Side 2 = Bend Length – Material Thickness – Inside Bend Radius

1.50 – 0.048 – 0.025 = 1.427

Step 3:
Length of Neutral Axis = Angle x (pi / 180) x (Inside Radius + K-factor x Thickness)

90 x (3.14159 / 180) x (0.024 + 0.41 x 0.048)

1.570795 x (0.024 + 0.01968)

1.570795 x 0.04368 = 0.06861

Step 4:
Total Material = Length of Side 1 + Length of Side 2 + Length of Neutral Axis

0.427 + 1.427 + 0.06861 = 1.92261 or 1.923

Figure 2.6 – Calculations for Material when Bending a L-Bracket

To discover the third number, the length of the neutral axis of the sheet metal, we will multiply the bend angle times pi (3.14159) divided by 180 to get the bend angle in radians. Then we will multiply the K-factor of 0.41 times the thickness of the material to get the radius of the neutral axis. To get the true length of the bend multiply the bend angle in radians times the radius of the neutral axis.

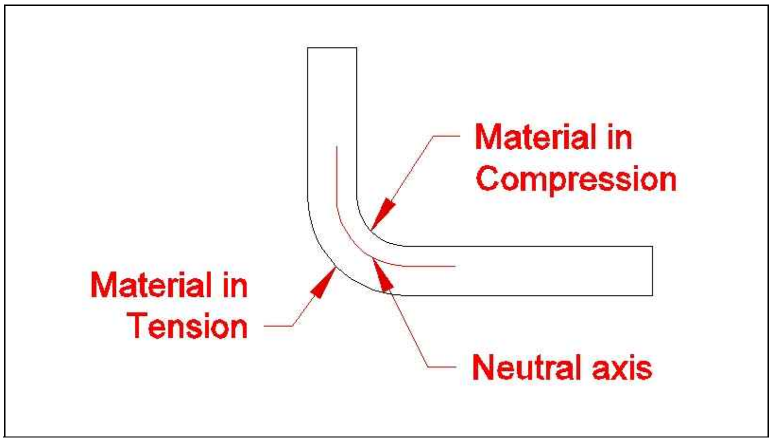

Figure 2.7 – Detail of Neutral Axis

To obtain the total length of the L-Bracket shown in figure 2.5, add the length of side 1, the length of Side 2 and the length of the neutral axis. By adding 0.427 + 1.427 + 0.06861, we get 1.923 inches. Notice, that the answer is not 2.00, which is 0.50 and 1.50. Computing the bend allowance will give us and the machinist the correct result when bending the 0.50 lip up 90 degrees. If we had a 2.00 blank and bent the material up on a 0.024 inch radius, we would have a 0.577 lip.

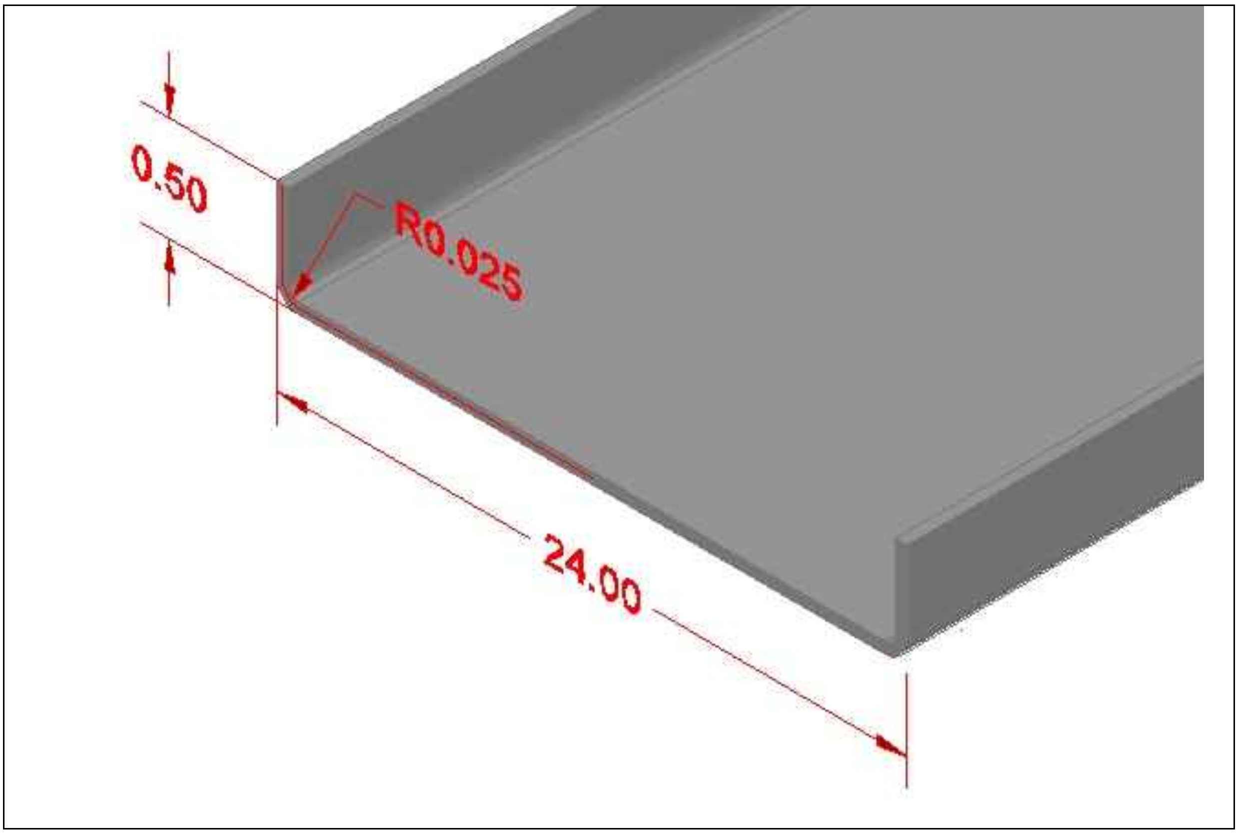

Figure 2.8 – Out of Scale Picture of the Cover Plate showing 24.0 Length and Bends

Now, the we know the process to compute the blank size for an L-bracket. To find the true length of the material for the part in figure 2.8, we need to obtain three measurements for a part that has two bends of the same distance, however this time we will multiply the bend lip times two and the length of the neutral axis times two.

To determine the length of the lip, we will subtract the material thickness and the inside radius from the 0.5 side, which gives us 0.427 inches. To get the length of the non bent section, we will add the material thickness and the inside radius, and multiply the number by two. Then subtract it from the 24.00 side, which gives us 23.854 inches.

To discover the third number, the length of the neutral axis of the sheet metal, we will multiply the bend angle times pi (3.14159) divided by 180 to get the bend angle in radians. Then we will multiply the K-factor of 0.41 times the thickness of the material to get the radius of the neutral axis. To get the true length of the bend multiply the bend angle in radians times the radius of the neutral axis. To account for the two bends, we multiply the answer by two.

To obtain the total length of the U-Bracket shown in figure 2.6, add the length of side 1 and 3, the length of Side 2 and the length of both neutral axes. By adding 0.427 + 0.427 + 23.854 + 0.137, we get 24.845 inches. Since the cover plate is square, the blank 18-gauge steel sheet will be cut at 24.845 by 24.845 inches.

Bend Formula for a U-bracket

Step 1:
Length of Sides 1 and 3 = Bend Length – Material Thickness – Inside Bend Radius

$$0.50 - 0.048 - 0.025 = 0.427$$

Step 2: Length of Side 2 = Length – 2 x (Material Thickness + Inside Bend Radius)

$$24.00 - 2\,(0.048 + 0.025) = 23.854$$

Step 3:
Length of Neutral Axis = 2 x Angle x (pi / 180) x (Inside Radius + K-factor x Thickness)

$$2 \times 90 \times (3.14159 / 180) \times (0.024 + 0.41 \times 0.048)$$

$$2 \times 1.570795 \times (0.024 + 0.01968)$$

$$2 \times 1.570795 \times 0.04368 = 0.13722$$

Step 4:
Total Material = Length of Side 1 and 3 + Length of Side 2 + Length of Neutral Axis

$$0.427 + 0.427 + 23.854 + 0.137 = 24.845$$

Figure 2.9 – Calculations for Material for a U-Bracket

Making a Drawing for a Stamped and Formed Panel

Now that we have the technical data computed, we may also have to draw a layout for the machinist to cut the blank before the sheet metal cover is formed. In our computer aided design program, draw a 24.845 by 24.845 inch square. Offset four lines 0.496 on all sides. Remove the material in the corners as shown in figure 2.10. Change the linetype representing the bending line as shown in the figure to a phantom line and label one of the phantom lines as a bending line. Dimension the blank as shown. A designer can place this top view of the cut out blank from the sheet metal off to side on the drawing sheet.

On the cover panel, we are bending the part in all four corners, so we will not label the direction of the material's grain on the drawing of the blank. When we do have the choice of indicating that direction of the material grain such as with the L-bracket, then we will label the drawing so that the grain is perpendicular to the bend line. Machinist, technicians and designers can easily see the grain in the metal, so adding this hopeful note will aid an inexperienced production worker in cutting or sheering sheet stock in the proper direction to allow the part to be formed and to reduce the chance of cracking the metal along the area of the bend.

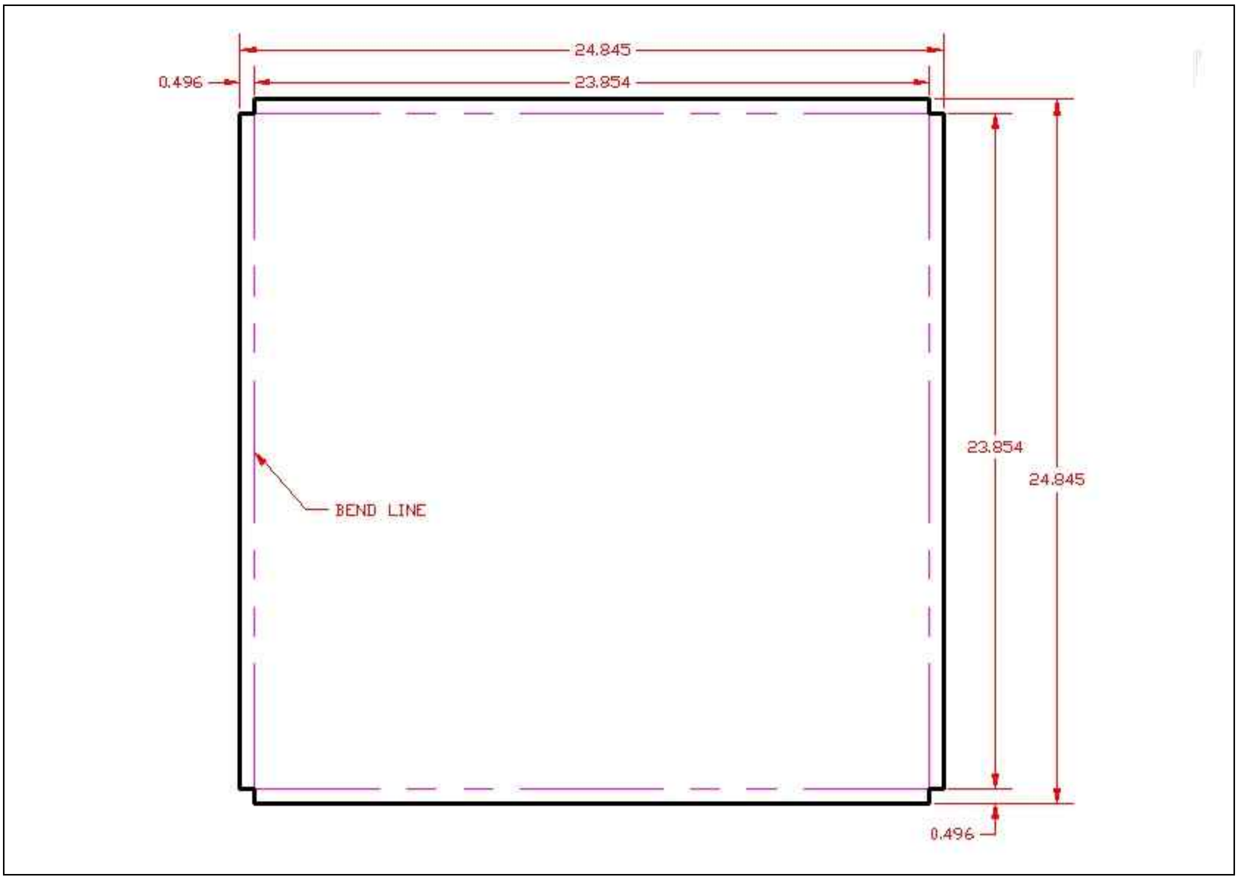

Figure 2.10 – Top View of the Blank Cutout

When the drawing of the blank cut out is complete, place two orthographic views of the cover panel on a border and dimension the views. Add the fabricating notes just as we did with the rectangular for circular problem in the World Class CAD Fundamentals of 2D Drawing textbook. The first note will call out the 18-gauge cold rolled steel. The second note will have the machinist deburring the part. The third note will call for one coat of primer and one coat of flat black enamel. The last note will call for the standard in tolerances used in previous drawings.

After we submit the drawing to the checker and then the approver, go ahead and practice these steps of making a formed part with cover plates, U-brackets and L-brackets. Each time the part drawings are submitted, include the calculations for the bends on 8.5 x 11 paper, word processing document or on a spreadsheet.

Another area where we can practice our CAD skills is to draw the neutral axis on a side view of the drawing and use the List function to retrieve the length of the components of the sheet metal allowance. If we use a graphic to show the calculation, include this information on the drawing, next to the blank cutout.

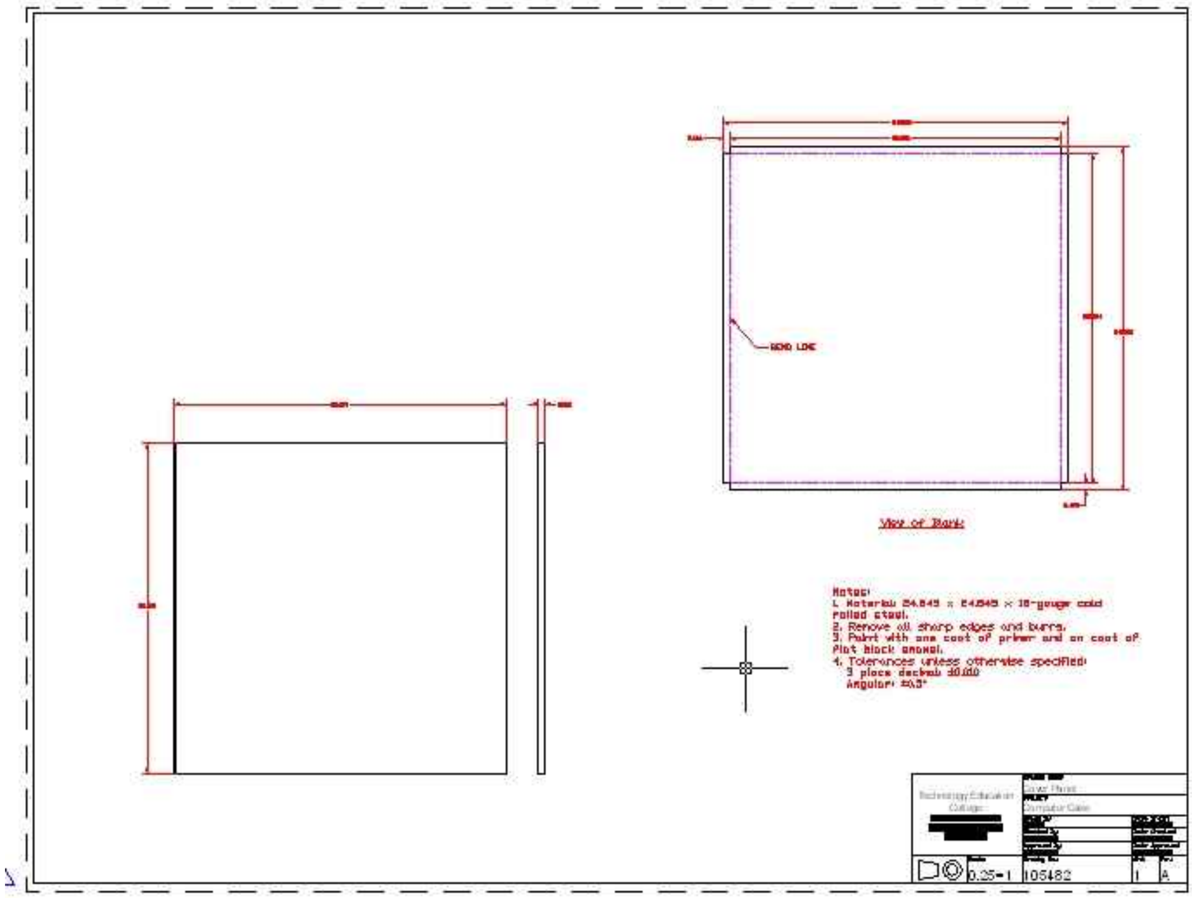

Figure 2.11 – Final Drawing of the Cover Panel

*** World Class CAD Challenge 08-1 * - Save and close the Cover Panel drawing file. Create a new file and make a part drawing for a 16-gauge x 18.0 x 18.0 cover panel with four 0.375 bends, compute the bends, draw the blank cutout, dimension, and place the border and notes in less than 60 minutes. Continue this drill three more times, for a L-Bracket and various other formed components, each time completing the drawing under 60 minutes to maintain your World Class ranking.**