Sheet Metal Forming

Slitting
In previous classes, we talked about rolling a large billet of metal into sheet metal. Rolling is done to create the desired thickness of the sheet metal.

Once we've rolled a master coil of sheet metal, which could be 6 ft. wide or more, we need to slit it to the desired width. To do this, we use a slitter. The metal is pulled through a set of slitting knives in the center…rotary cutters that shear the metal (the same idea as rotary cutters that quilters use for cutting fabric). The slit coils are wound up on a recoiler. The cartoon shown in class doesn’t show the protective barrier between the slitter and the operator, but in modern high-speed slitting lines there’s always the possibility of injury, so they tend to be caged off.

Making Blanks
Now that we’ve got a coil of metal of the right thickness and width, we need to make a blank. We can use a mechanical press or a hydraulic press.

The strips shown in this cartoon are shown in plan view…we’re looking down on a flat sheet of metal. The thin lines show where the sheet metal is cut, and the shapes on the left show the finished parts.

Cutoff
The simplest type of blank is a cutoff blank, where you shear the metal in a repeated pattern to produce blanks of the same size and shape, with no scrap. When you first start the blanking press, there is some scrap at the end of the coil, but after that there’s no more scrap.

The 6-sided shape could theoretically produce no scrap at all. This isn’t what really happens in a press, because it’s cheaper to feed in a little extra material than it is to align the shears up perfectly with the end of the coil. The material savings aren’t worth the added labor costs and downtime during changeover.

Parting
It’s not always possible to produce blanks without scrap along the strip. If your finished part shares two edges with the strip, but the other edges won’t nest, then there will be some scrap for every blank. This process is called parting.

None of the shapes shown here will nest in any direction.

The tool designer has to decide whether the tooling should remove the part or the scrap. In each of these cases, the scrap is smaller, so it is cheaper to build a tool to remove the scrap…there is less tool steel to buy and machine.

With sheet metal, the scrap doesn’t go to a landfill…it’s actually worth something as a waste product. You can sell the scrap to a scrap collector, and it will get resold to a mill to make new steel or aluminum sheet. However, you’ll only get 10¢ on the dollar for scrap, so you can save a lot of money by eliminating scrap from your process.
Blanking (Punching)

Some shapes don’t share an edge with the sheet metal, so there’s scrap all around the part. This process is called blanking or punching. You cut the blank in one stroke of the press, and the skeleton is scrap.

A skeleton is a continuous piece of scrap…so after we cut the circular blanks, another shearing operation is used to cut the skeleton into small pieces that will fit into the scrap bin.

Piercing (Perforating)

Every process we’ve looked at so far involves cutting a blank from a strip of metal. We can cut out the scrap instead, with a piercing operation. This is also called perforating. The perforated metal industry is huge.

If you look up at the sheet metal on the inside of a schoolbus roof, it’s all perforated metal. The tiny holes help dampen sound, and reduce the weight of the roof. The filter industry uses perforated metal for filters.

The light fixture industry uses perforated metal as light diffusers. The architecture industry uses perforated metal for ceilings and wall surfaces.

If you perforate a strip, you can blank out a larger shape, like the disc shown here. It’s easier to put the smaller holes in first, because it’s easier to handle a continuous strip than discrete parts.

Notching

You can remove scrap but keep the blanked part continuous with notching. By cutting away the scrap (gray), we’re left with a series of hexagons connected together. These parts will be separated further along in the press to make finished products.

You can also notch blanks. Here’s a rectangular blank where we notch the four corners. If we bend the metal at the blue dotted lines, we can make a simple tray. If we want to spot-weld the corners, we can notch some tabs into the corners.

Lancing

Lancing is a process which creates no scrap. The reason that we lance metal is to free metal for forming. The cartoon shows a lanced strip that has a tab bent down in a subsequent step. If you look inside a computer, you’ll see this process used a lot in the metal housing. Sometimes a tab will have a hole punched in it for a screw or rivet.

A tab produced by lancing saves you from having to spot-weld a bracket to the sheet metal, so it not only saves metal, it saves processing. Lancing is also used for creating louvers in sheet metal.

If you make a can by deep-drawing, the metal stretches differently in different directions, so even if you start with a perfectly circular blank, ears will form on the finished product.

We’ve got to get rid of these ears if we want a uniform height, so that’s where trimming comes in. After forming, we trim the part to final dimensions. This process produces scrap, but it’s unavoidable. What you can avoid is excessive scrap…so you design the formed part to minimize the extra material that requires trimming.
Other Sheet Metal Considerations

The Cut Edge

This is a cartoon of the cut edge created by shearing metal. You know that sheared metal doesn’t have smooth edges… just look at a cut sheet metal washer. There’s a burr on one side, and a rounded edge on the other. The diagram shows that there’s a smooth portion of the hole where the metal fails in shear, and a rougher part where the metal fails by tearing. When you get a burr, you may have to do some additional finishing operations, depending on the product.

The percent penetration before tearing depends on composition and work hardening. You’re balancing the shear strength of the metal (which allows it to shear cleanly) with the tensile strength of the material (which allows it to break). Annealed low-carbon steel and aluminum will have a small break zone, and a large burnished zone. Cold-rolled steel with a little more carbon will have a small burnish zone and a large break zone, so the burr could be more severe.

Blanking Force

Consider the force it takes to produce a blank. Since we’re shearing metal, we need to exceed the shear strength. Force = stress × area, and the area we’re shearing is the thickness of the sheet metal times the length along which it’s being cut.

Unfortunately, shear strength data is hard to come by. There’s a shortcut equation in the textbook that says you can estimate the ultimate shear strength as $\tau_{\text{ult}} = 0.7 \sigma_{\text{UTS}}$, so we’ll use this equation in our example. Let’s say we want to cut a 6” disk out of 0.030”–thick low-carbon steel with a tensile strength of 40 ksi. The length we’re cutting is the circumference of the circle: $\pi$ times the diameter. Multiply this result by the sheet metal thickness to obtain the area being sheared. The required force is

$$F = 0.7 \sigma_{\text{UTS}} \pi d = \frac{0.7 \times 40000 \text{ lb}}{\text{in.}^2} \times \frac{0.030 \text{ in.} \times \pi \times 6\text{ in.}}{15800 \text{ lb.}} \approx 8 \text{ tons.}$$

If you have a number of cuts to make, you could easily exceed the capacity of the press. This is one of the reasons that sheet metal parts are often made in more than one operation. If you’ve got two cuts to make, and each is about 8 tons, and you have a 12-ton press, you can’t make both cuts at the same time on the same press.

One example of a sheet metal part that cannot be made in one strike is a can…like a pop can, or a soup can. First, you blank a circle. Next, you draw it into a broad, shallow dish. Then, you deep-draw the dish into a can. Last, you trim the top edge, like we saw earlier. So each piece is handled separately, and it’s transferred to the next station for the next step…hence the name transfer press.

In a progressive die, the part stays connected to the strip all the way through the process. Using lancing, drawing, bending, & other operations, the part is formed to its final shape before being cut from the strip.
Springback

After we’ve blanked a part, we can deform it by bending, drawing, deep drawing, etc.

When we bend sheet metal, it stretches elastically until it exceeds the yield strength. Then it deforms plastically, and takes a permanent set (blue dot on the center diagram). When we release the load, the plastic strain is maintained, but the elastic strain is recovered. The metal “springs back” a little.

You can see springback with a paperclip. If you bend a paperclip a little bit (elastic deformation), it recovers its shape when you release the load. If you bend a paperclip 90° (plastic deformation), it doesn't stay at 90° when you release the load – it springs back a little, giving you an angle of 70° or so. If you need a 90° bend, you have to push the paperclip to 110°, so it springs back to 90°. Engineers who design metalforming tooling have to take springback into account as they design tooling and processes.

Anisotropy and Thinning

Another issue with sheet metal: it’s been rolled, and the mechanical properties are not the same in all directions. We say the material is anisotropic... “an” means “not”, and “isotropic” means “same properties in all directions.” For example, the density of iron in a cannonball is isotropic...it's the same anywhere within the cannonball. Air pressure in the atmosphere is anisotropic...it's greater at sea level than on a mountaintop.

If a solid material has different mechanical properties in different directions, then it will stretch differently in each direction. Therefore, the strain in the width direction, $e_{\text{width}}$, is different from the strain in the thickness direction, $e_{\text{thickness}}$, and different again from the strain in the length direction, $e_{\text{length}}$.

We can calculate a plastic strain ratio, $R = \frac{e_{\text{width}}}{e_{\text{thickness}}}$. Why is this important? If $R$ is small, it means that the sheet metal gets thinner more easily than it stretches in-plane...so the sheet metal will neck (just like our tensile specimen in the lab), and the sheet metal will split if we try to form it. As a result, the sheet metal forming industry pays a lot of attention to $R$.

We can cut tensile specimens from a strip of steel, and measure the tensile properties in three directions: parallel, perpendicular, and 45° to the rolling direction. Once we have the plastic strain ratio in the three directions, we can calculate the average plastic strain ratio, or “$r$ bar”, as $R_{\text{avg}} = \frac{R_{0^\circ} + 2R_{45^\circ} + R_{90^\circ}}{4}$. When this value is greater than 1, then the material is stronger in the thickness direction than in planar directions.

In my experience with deep drawing in the oil filter industry, low-carbon sheet steel will crack if $r < 1.3$, but it runs fine in the presses if $r < 1.4$. 
We can also calculate the anisotropy of the sheet, as \( \Delta R = \frac{R_0 - 2R_{45} + R_{90}}{2} \). When \( \bar{r} = 1 \) and \( \Delta R = 0 \), the properties are the same in all directions. As \( \Delta R \) gets bigger, we get earing.

If we plot the plastic strain ratio \( R \) as a function of the angle between the test direction and the rolling direction, we see this pattern for angles 0°, 45°, and 90°:

Now let’s run many samples at all angles between 0° and 90°, and plot the results for three different samples of sheet metal. The purple line matches the data from the previous graph, and has an average plastic strain ratio \( \bar{r} = 1.4 \). The cartoon figures at the right show the tallest can which can be produced for each material in one strike. We’ve all seen taller cans in the grocery store; they are made in more than one deep-drawing operation.

**Metal Spinning**

If you want to form a piece of sheet metal, but low volumes make drawing dies cost-prohibititive, you can use a process called spinning. There’s a cartoon in the textbook that shows this process. Set the sheet metal on a lathe, get it spinning, and press a tool against the sheet metal to push it against a mandrel. The mandrel can be made of cast iron, steel, aluminum, or even wood…so the tooling is pretty cheap. This work can be done by people or by CNC machines. You can spin pretty large structures. The largest machines will handle a 20-foot-diameter piece of sheet metal. As you get to that size, it gets hard to find sheet metal that’s wide enough for the blank.