

Lecture 4. Sheet metal processing

The raw material for sheet metal manufacturing processes is the output of the rolling process. Typically, sheets of metal are sold as flat, rectangular sheets of standard size. If the sheets are thin and very long, they may be in the form of rolls. Therefore the first step in any sheet metal process is to cut the correct shape and sized 'blank' from larger sheet.

Sheet metal processing is an important process for many HK industries, which are one of the largest manufacturers of products such as home appliances (fridge, washer, dryer, vacuum cleaners etc.), electronics (DVD- and CD-players, stereos, radios, amplifiers etc.), toys and PC's. Most of these products have metal casings that are made by cutting and bending sheet metal. We look at some the basic sheet metal cutting and forming processes.

4.1. Shearing and Punching

Shearing is similar to the process by which you cut a sheet of paper using scissors. However, the machinery used is a little different. The image below shows two typical machines used to cut sheet metal – the first is a shearing machine which has a long blade to make straight line cuts; it is used to cut long sheets of metal into smaller sheets. This operation is similar to that of a paper-cutting machine: the metal sheet is held on top of a hardened die, and the shearing blade cuts downward, usually driven by an electrical or hydraulic punch. The second image is a sheet metal punching machine – the cutting tool is a punch, a piece of hard tool steel which is punched down on the sheet to cut a hole. The punch in the image is a turret punch – the turret is a rotating tool holder that can hold tens of different shapes and sizes of punching dies. Typical shapes are rectangular and circular. By punching in a series of steps, a long slot can be cutout. A typical punching operation is similar to that of a paper punch that you may have used to cut holes to hold paper in a 3-ring binder.



Figure 1. (a) Hydraulic sheet metal shearing machine (b) A CNC turret-type sheet metal punching machine

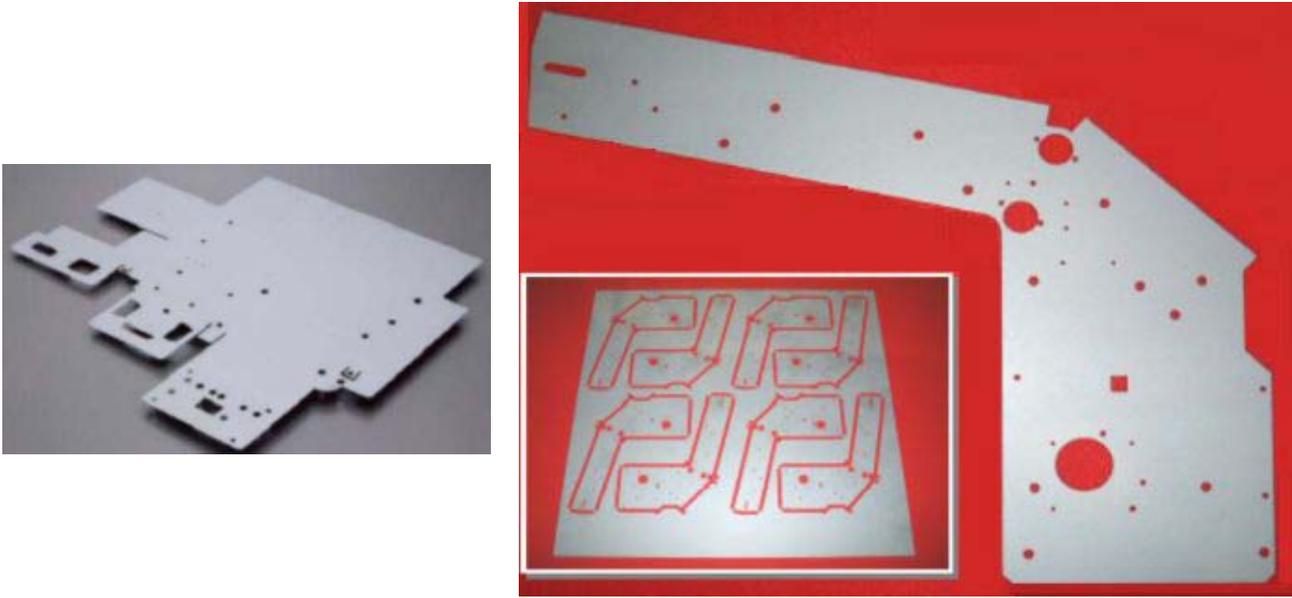


Figure 2. (a) Typical punched part. (b) A square sheet stock used to punch out 8 parts; enlarged view of part

Cutting metal by punching utilizes almost pure form of failure in shear; therefore the cutting forces and the energy required are functions of the shear strength of the material. The figure below shows a schematic of how the metal is sheared between the cutting edges of the punch and the die.

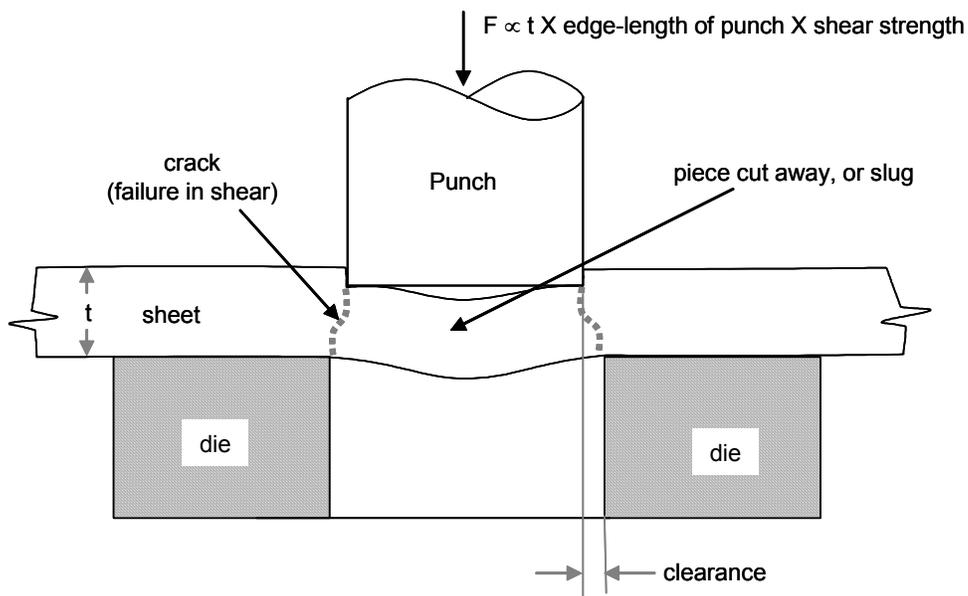


Figure 3. Schematic of the shearing process

The clearance is typically between 2% and 10% of sheet thickness, t ; if the clearance is larger, the part that is sheared away, called the slug, may just flow through the clearance without separating from the sheet.

Other methods of cutting sheets

These include flame cutting, sawing, laser cutting, and water-jet machining, some of which we shall cover later.

4.2. Bending

A large percentage of sheet metal parts are bent along some lines to get them into the desired shape for use (for example, think of the metal case for a computer). The sheet metal part before it is bent is called a blank. Here, we shall also look at a special case of bending of wires or rods – the reason this is interesting is that one of the most important mechanical component: springs.

Bending induces plastic deformation in the material, so the part retains its shape after the bending force is released. However, on studying the stress-strain curves for materials, you will notice that when a material is deformed into the plastic region and then released, some portion (the elastic part) of the strain is released. This phenomenon causes an action called spring-back in the part that we want to bend. Thus, the bending dies must account for the spring-back.

Figure 4 shows a simple bend on a rectangular blank. The top profile of the blank undergoes extension – a thin element along the top surface will be longer after the bending than the initial length; likewise, the bottom portion experiences compression. Thus, as we travel from the bottom to the top, there is some layer in the middle which retains its original length – this forms the neutral axis. The location of the neutral axis, and therefore its length, determines the length of the blank we must begin with, in order to get the final part with the correct geometry.

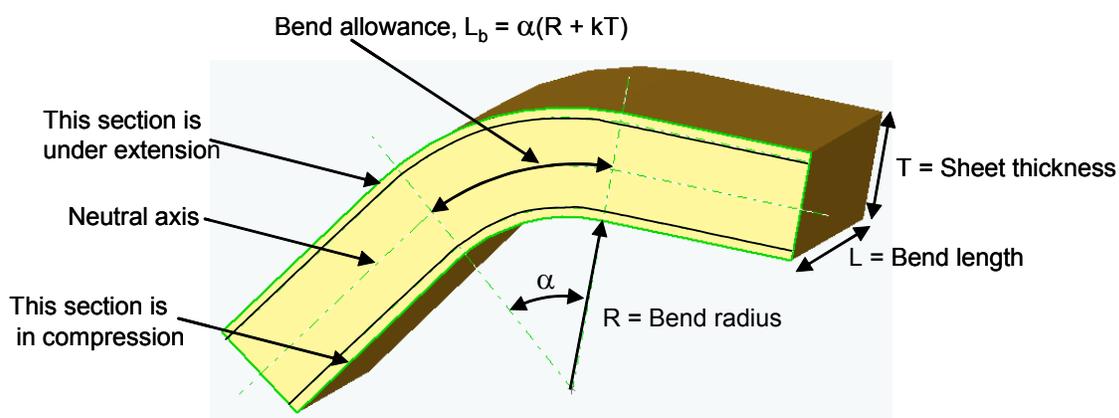


Figure 4. Schematic of bending mechanics

4.2.1. Some issues in bending

(i) Minimum bending radius and cracking

When the bending radius is too small, the strain level on the outer layers is too high, and usually the top layer will undergo plastic deformation or cracking. If you keep bending, there will also be failure in buckling at the bottom. The engineering strain during bending is approximated as:

$$\text{Engineering strain in bending} = e = 1/(1 + 2R/T)$$

Thus, as R/T decreases, stress increases and cracking begins. The minimum bend radius is the radius at which cracking begins; it is expressed as a multiple of T ; i.e. a minimum bend radius = $3T$ for a sheet of 1mm means that the bend radius should be larger than 3mm to avoid cracking.

(ii) Anisotropic properties

Since most sheets used as blanks are formed by rolling, they have anisotropic properties (different yield strength along different directions). Thus, the orientation in which you cut the blank from the raw sheet may depend on the bending operations you will perform. Figure 5 demonstrates this.

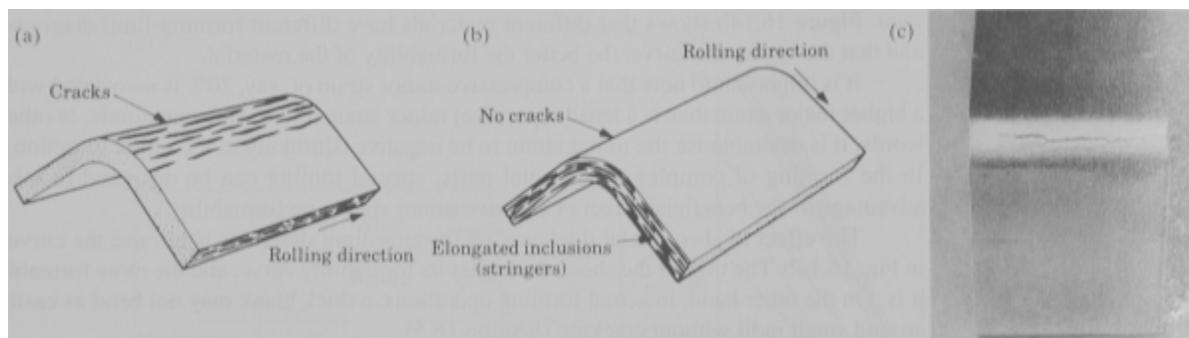


Figure 5. Bending outcome is dependent on anisotropic structure [source: Kalpakjiam & Schmid]

In the above figure, you also see the Poisson effect. When a material is subjected to tensile stress, it elongates; however, due to the law of conservation of mass, the total volume remains the same, and therefore its cross section area decreases. This effect, called the Poisson effect, can be seen in the two ends of the bent metal piece in the figure.

(iii) Springback

Recall from your stress-strain curves that when the stress is released, the all materials experience some elastic recovery. An approximate formula for springback is given terms of the relation between the initial bending radius, R_i , and the final bending radius (after springback), R_f :

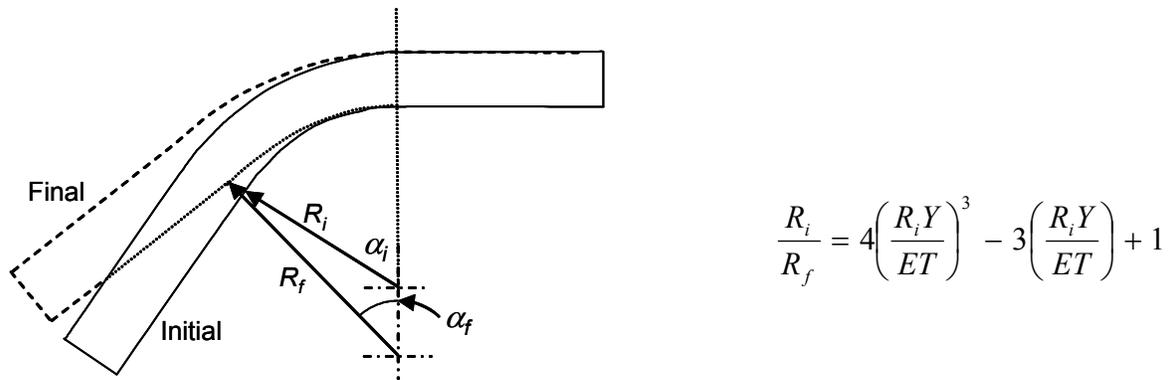


Figure 6. Springback in bending

There are several methods to counter the effect of springback:

(a) Compensation method: here, the metal is bent by a larger angle, such that it springs back to the desired value; namely, the desired angle of bending is set to R_f , and then, using the above formula, R_i is computed; then the blank is bent to R_i . In most companies, some trial and error is required before the exact bending angle that gives the desired result is obtained.

(b) Coining the bend: In this method, the bending is achieved by pushing with a punch, and letting the metal bend into a die; at the end of the cycle, a relatively large squeezing force is exerted, which creates a permanent bend angle.

Figure 7(a) below shows some typical bending operations. Another bending operation is flanging – the flange is usually a 90° bend of uniform width running along a border of a sheet. Take a look at the box covering your computer to see a practical example. Some typical flanges are shown in figure 7(b).

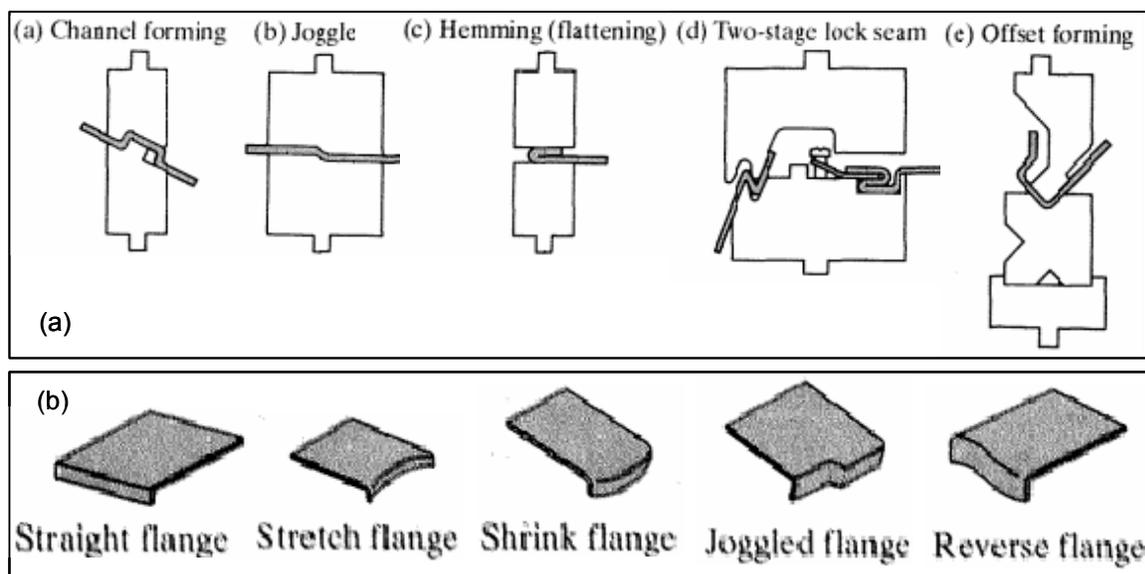
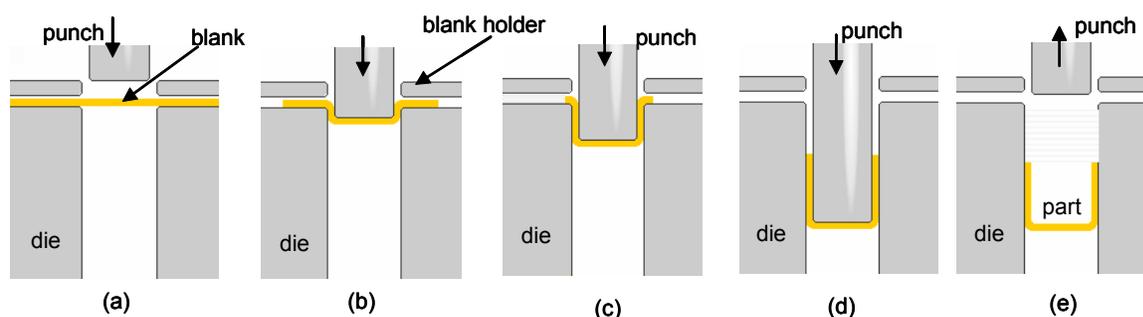


Figure 7.(a) Typical Sheet-metal bending operations (b) Typical flanges [source: Kalpakjian & Schmid]

4.3. Deep Drawing

Another method of great importance is deep drawing, which is commonly used to manufacture cooking utensils and other containers made from metal (and also cans of soda drinks such as Coke etc.) Figure ?? shows the deep drawing process in several steps, as the punch pushes the blank down into the die cavity, and finally retracts; the part is finally ejected out of the cavity by an ejection pin (not shown in the figure). The blank is a piece of sheet-metal cut to the required shape. The die has a cavity in the shape that is required (the most common shape is cylindrical). The punch is of the same shape, but the difference in the size of the punch and the cavity is just sufficient to allow the sheet to be pushed by the punch into the die. As the punch pushes the sheet into the cavity, the upper portions of the sheet will tend to deform in wrinkled shapes – this is avoided by keeping the top part of the sheet pressed down by a blank holder. The punch is of the same shape, but the difference in the size of the punch and the cavity is just sufficient to allow the sheet to be pushed by the punch into the die. As the punch pushes the sheet into the cavity, the upper portions of the sheet will tend to deform in wrinkled shapes – this is avoided by keeping the top part of the sheet pressed down by a blank holder.



Examples of deep drawn parts

Figure 8. (top) stages of the deep drawing process and (bottom) some example parts

There are a variety of other sheet metal processes that have specialized uses in some industries. These include spinning, explosive forming, etc. You can read some more about them from your text book.