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. 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 industries, producing 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 of the basic sheet metal cutting and forming processes.

The operations are performed on relatively thin sheets of metal:

- Thickness of sheet metal = 0.4 mm to 6 mm
- Thickness of plate stock > 6 mm
- Operations usually performed as cold working
Advantages of Sheet Metal Parts

- High strength
- Good dimensional accuracy
- Good surface finish
- Relatively low cost
- Economical mass production for large quantities

Sheet Metal Cutting & Forming Processes

Classification

1. Cutting Operations
2. Bending Operations
3. Drawing
Basic Types of Sheet Metal Processes

1. Cutting
   - Shearing to separate large sheets
   - Blanking to cut part perimeters out of sheet metal
   - Punching to make holes in sheet metal

2. Bending
   - Straining sheet around a straight axis

3. Drawing
   - Forming of sheet into convex or concave shapes

1. Cutting Operations

Three principal operations in pressworking that cut sheet metal:

- Shearing
- Blanking
- Punching
1.1 Shearing

Sheet metal cutting operation along a straight line between two cutting edges. Typically used to cut large sheets.

Shearing of sheet metal between two cutting edges:
(1) just before the punch contacts work;
(2) punch begins to push into work, causing plastic deformation;
Shearing of sheet metal between two cutting edges:
(3) punch compresses and penetrates into work causing a smooth cut surface;
(4) fracture is initiated at the opposing cutting edges which separates the sheet.

1.2 Punching – 1.3 Blanking

Punching - sheet metal cutting operation where the cut piece is scrap.
Blanking - sheet metal cutting to separate piece (called a *blank*) from surrounding stock.
Die size determines blank size $D_b$

Punch size determines hole size $D_h$

$c$ = clearance

---

**Clearance in Sheet Metal Cutting**

Distance between punch cutting edge and die cutting edge

Typical values range between 4% and 8% of stock thickness
Clearance in Sheet Metal Cutting

- Recommended clearance is calculated by:
  \[ c = at \]
  where,  
  - \( c \) = clearance  
  - \( a \) = allowance  
  - \( t \) = stock thickness

- Allowance \( a \) is determined according to type of metal

Punch and Die Sizes

- **For a round blank of diameter \( D_b \):**
  - Blanking punch diameter = \( D_b - 2c \)
  - Blanking die diameter = \( D_b \)
  where \( c \) = clearance

- **For a round hole of diameter \( D_h \):**
  - Hole punch diameter = \( D_h \)
  - Hole die diameter = \( D_h + 2c \)
  where \( c \) = clearance
Cutting Forces

Important for determining press size (tonnage)

\[ F = S \cdot t \cdot L \]

where,
\( S \) = shear strength of metal
\( t \) = stock thickness
\( L \) = length of cut edge

2. Sheet Metal Bending

Straining sheetmetal around a straight axis to take a permanent bend

(a) Bending of sheet metal
2. Sheet Metal Bending

Metal on inside of neutral plane is compressed, while metal on outside of neutral plane is stretched.

(b) both compression and tensile elongation of the metal occur in bending.

Types of Sheet Metal Bending

- **V-bending** - performed with a V-shaped die
- **Edge bending** - performed with a wiping die
- For low production
- Performed on a press brake
- V-dies are simple and inexpensive

(a) V-bending:
**Edge Bending**

- For high production
- Pressure pad required
- Dies are more complicated and costly

(b) edge bending.

**CD: Shearing / Bending**
3. Drawing

Sheet metal forming to make cup-shaped, box-shaped, or other complex-curved, hollow-shaped parts

- Sheet metal blank is positioned over die cavity and then punch pushes metal into opening.

- Products: beverage cans, ammunition shells, automobile body panels.

- Also known as **deep drawing** (to distinguish it from wire and bar drawing)

---

(a) Drawing of cup-shaped part:

1. before punch contacts work
2. near end of stroke.

(b) workpart:

1. starting blank
2. drawn part.
Clearance in Drawing

- Sides of punch and die separated by a clearance $c$ given by:
  \[ c = 1.1 \, t \]
  where $t$ = stock thickness
- In other words, clearance is about 10% greater than stock thickness

Tests of Drawing Feasibility

- Drawing ratio
- Reduction
- Thickness-to-diameter ratio
**Drawing Ratio DR**

Most easily defined for cylindrical shape:

\[ DR = \frac{D_b}{D_p} \]

where \( D_b = \) blank diameter
\( D_p = \) punch diameter

- Upper limit: \( DR \leq 2.0 \)

---

**Reduction r**

- Defined for cylindrical shape:

\[ r = \frac{D_b - D_p}{D_b} \]

- Value of \( r \) should be less than 0.50
**Thickness-to-Diameter Ratio** $t/D_b$

Thickness of starting blank divided by blank diameter:

\[ \frac{t}{D_b} \]

- Desirable for $t/D_b$ ratio to be greater than 1%

- As $t/D_b$ decreases, tendency for wrinkling increases

**CD: Drawing**
Dies and Presses
for Sheet Metal Processes

Gap frame press for sheet metalworking
capacity = 1350 kN (150 tons)
Press brake
bed width = 9.15 m
and capacity = 11,200 kN (1250 tons).

Sheet metal parts produced on a turret press,
showing variety of hole shapes possible
Computer numerical control turret press