Metal Forming Processes (ME5807)

Dr. Yogesh Kumar
Assistant Professor
Mechanical Engineering Department
National Instiute of Technology Patna
Bihar - 800 005, India
yogesh.me@nitp.ac.in

Sheet Metal Forming Processes

- Forming and related operations performed on metal sheets, strips, and coils
- High surface area-to-volume ratio of starting metal, which distinguishes these from bulk deformation
- Often called pressworking because presses perform these operations.
  - Parts are called stampings
  - Usual tooling: punch and die

Sheet Metalworking Defined

Cutting and forming operations performed on relatively thin sheets of metal.

- Thickness of sheet metal = 0.4 mm (1/64 in) to 6 mm (1/4 in)
- Thickness of plate stock > 6 mm
- Operations usually performed as cold working

Sheet and Plate Metal Products

Sheet and plate metal parts for consumer and industrial products such as:

- Automobiles and trucks
- Airplanes
- Railway cars and locomotives
- Farm and construction equipment
- Small and large appliances
- Office furniture
- Computers and office equipment

Advantages of Sheet Metal Parts

- High strength
- Good dimensional accuracy
- Good surface finish
- Relatively low cost
- For large quantities, economical mass production operations are available

Sheet Metal Forming Processes

- Bending
- Stamping
- Deep drawing
- Stretch forming
- Spinning and incremental forming

Bending

Bending

- Bending is the forming of sheet metal by application of force, which exerts pressure on a certain length of material either at a certain point or linear as an evenly distributed weight.
- This applied force is also called the bending moment. The force of the bending moment determines the degree of deformation.
- Bending of sheet metals can be done with press brakes, roll bending machines and embossing/coining machines.

Bending Operation

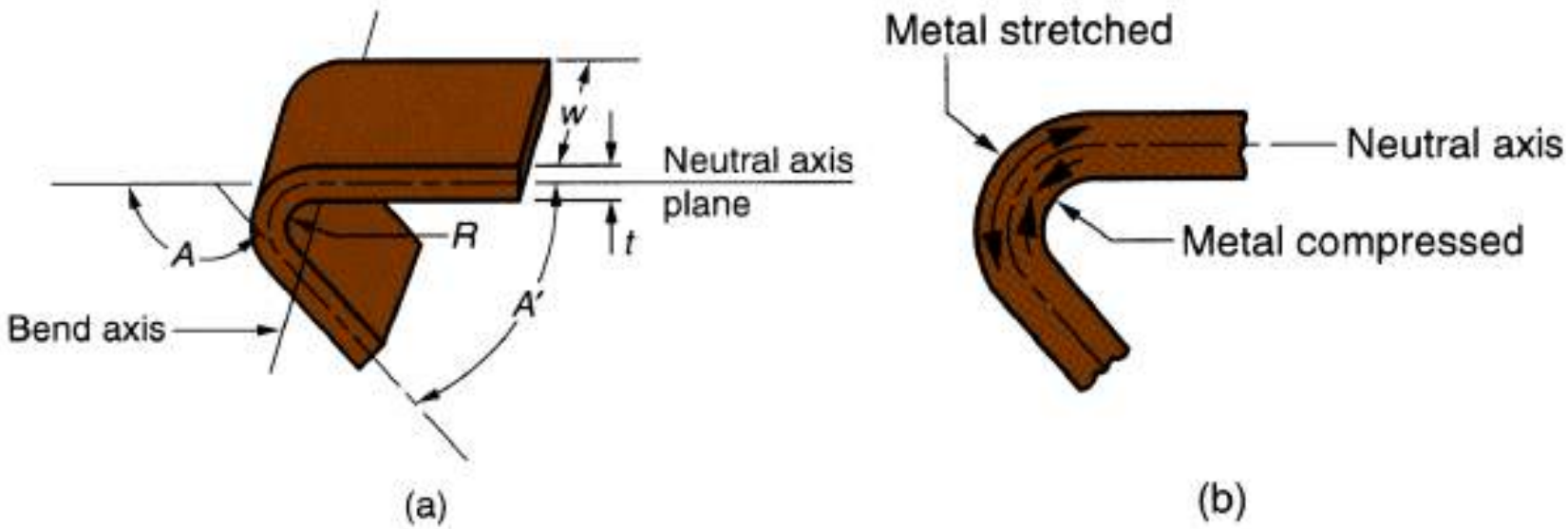

Figure: Bending

Types of Sheet Metal Bending

- V-bending - performed with a V-shaped die
- Edge bending - performed with a wiping die

V-Bending

- For low production
- Performed on a press brake
- V-dies are simple and inexpensive

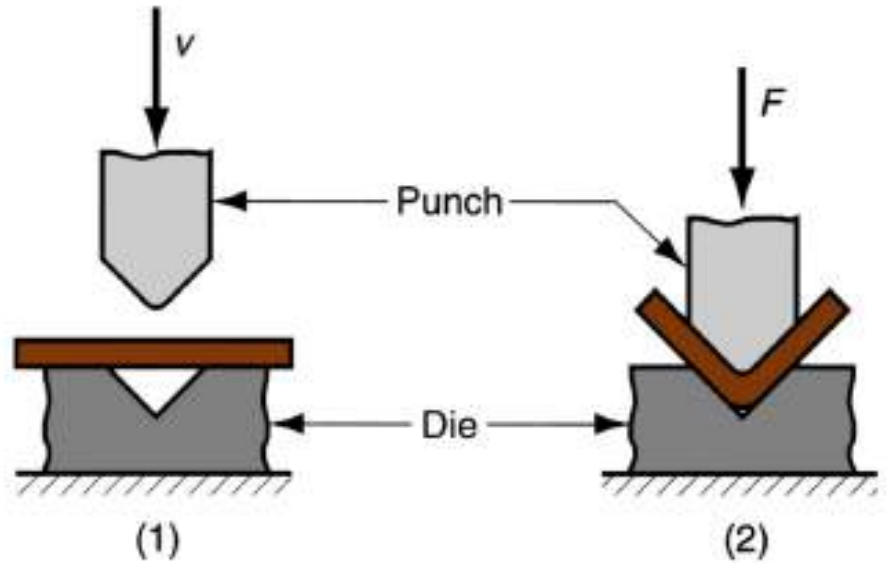

Figure: V-Bending

Edge Bending

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

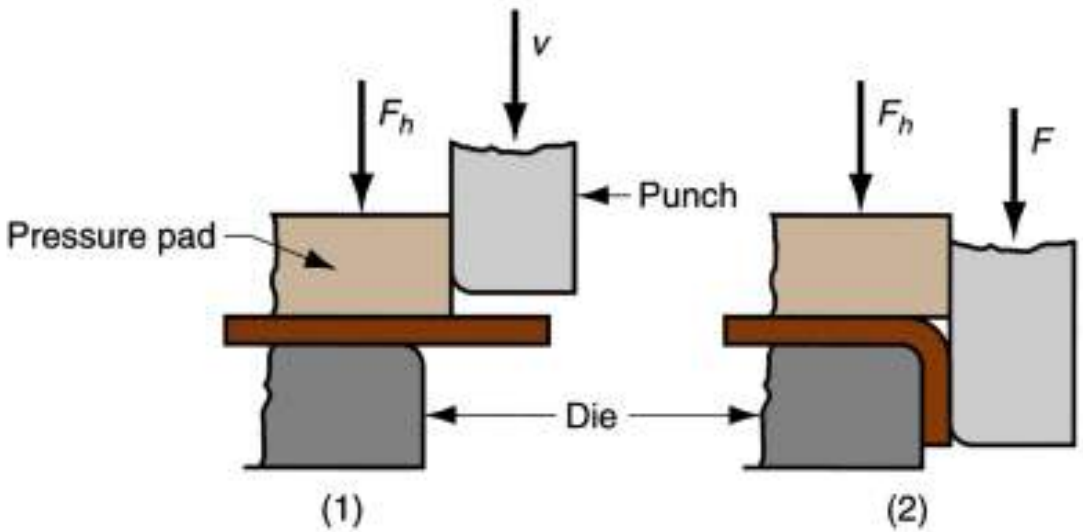

Figure: Edge Bending

Springback in Bending

Springback = increase in included angle of bent part relative to included angle of forming tool after tool is removed.

- Reason for springback:
  - When bending pressure is removed, elastic energy remains in bent part, causing it to recover partially toward its original shape.

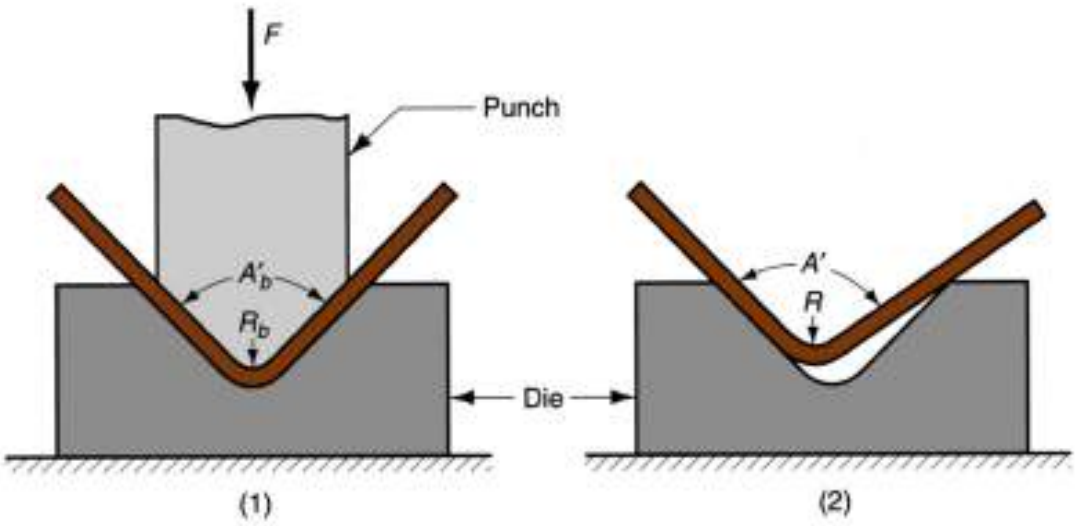

Figure: Springback in Bending

Thank You