Lecture 12: Cutting (Shearing) and Bending
Sheet Metalworking Terminology

- **Punch-and-die** - tooling to perform cutting, bending, and drawing
- **Stamping press** - machine tool that performs most sheet metal operations
- **Stampings** - sheet metal products

(1) Just before punch; (2) punch pushes into work, causing plastic deformation; (3) punch penetrates into work; and (4) fracture is initiated
Shearing, Blanking, and Punching

- Three principal operations in pressworking that cut sheet metal:
  - Shearing (cutting)
  - Blanking
  - Punching
Blanking and Punching

- Blanking (a) - sheet metal cutting to separate piece (called a *blank*) from surrounding stock
- Punching (b) - similar to blanking except cut piece is scrap, called a *slug*
Clearance in Sheet Metal Cutting

Distance between punch cutting edge and die cutting edge

- Typical values range between 4% and 8% of stock thickness
  - If too small, fracture lines pass each other, causing double burnishing and larger force
  - If too large, metal is pinched between cutting edges and excessive burr results

\[ C = \text{clearance} \]
Clearance in Sheet Metal Cutting

- Recommended clearance is calculated by:

\[ c = at \]

where \( c = \) clearance; \( a = \) allowance; and \( t = \) stock thickness

- Allowance \( a \) is determined according to type of metal
# Sheet Metal Groups Allowances

| Metal group                                                                 |  
|------------------------------------------------------------------------------|---
| 1100S and 5052S aluminum alloys, all tempers                                | 0.045  
| 2024ST and 6061ST aluminum alloys; brass, soft cold rolled steel, soft stainless steel | 0.060  
| Cold rolled steel, half hard; stainless steel, half hard and full hard       | 0.075  

Angular Clearance

- **Purpose:** allows slug or blank to drop through die
- **Typical values:** 0.25° to 1.5° on each side
Cutting Forces

- Important for determining press size (tonnage)

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

where \( S \) = shear strength of metal; \( t \) = stock thickness, and \( L \) = length of cut edge
Sheet Metal Bending

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

- If bend radius is small relative to stock thickness, neutral axis becomes longer
- Problem: to determine the length of neutral axis of the part before bending

\[ A_b = 2\pi \frac{\alpha}{360} (R + K_{ba}t) \]

where \( A_b \) = bend allowance; \( \alpha \) = bend angle; \( R \) = bend radius; \( t \) = stock thickness; and \( K_{ba} \) is factor to estimate stretching
  - If \( R < 2t \), \( K_{ba} = 0.33 \)
  - If \( R \geq 2t \), \( K_{ba} = 0.50 \)
V-Bending

• (1) Before bending
• (2) After bending

Application notes:
– Low production
– Performed on a press brake
– V-dies are simple and inexpensive

http://www.youtube.com/watch?v=qTUMbVNg1Xc
Edge Bending

- (1) Before bending
- (2) After bending
- Application notes:
  - High production
  - Pressure pad required
  - Dies are more complicated and costly

http://www.youtube.com/watch?v=W6usvOZ9ylw
Springback

When bending pressure is removed, elastic energy remains in bent part, causing it to recover partially toward its original shape

\[ SB = \frac{\alpha' - \alpha'_b}{\alpha'_b} \]

Need overbending to correct
Bending Force

• Maximum bending force estimated as follows:

\[ F = \frac{K_{bf}TSwt^2}{D} \]

where \( F = \) bending force; \( TS = \) tensile strength of sheet metal; \( w = \) part width in direction of bend axis; and \( t = \) stock thickness. For V-bending, \( K_{bf} = 1.33 \); for edge bending, \( K_{bf} = 0.33 \)
Roll Bending

- Large metal sheets and plates are formed into curved sections using rolls

http://www.youtube.com/watch?v=J09E_IuT1pM
HW assignment

- Reading assignment: Chapters 15, 16
- Review Questions: 14.3, 14.6, 14.7, 14.8,
- Problems: 14.1, 14.2, 14.3, 14.8, 14.10,