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 (rubbing) and larger force
  – If too large, metal is pinched between cutting edges and produces excessive burr (rough edge)

http://www.youtube.com/watch?v=VMu7_W0QE3Y
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 (4-8%)
# Sheet Metal Groups Allowances

<table>
<thead>
<tr>
<th>Metal group</th>
<th>Allowance</th>
</tr>
</thead>
<tbody>
<tr>
<td>1100S and 5052S aluminum alloys, all tempers</td>
<td>0.045</td>
</tr>
<tr>
<td>2024ST and 6061ST aluminum alloys; brass, soft cold rolled steel, soft stainless steel</td>
<td>0.060</td>
</tr>
<tr>
<td>Cold rolled steel, half hard; stainless steel, half hard and full hard</td>
<td>0.075</td>
</tr>
</tbody>
</table>

Softer metals have smaller allowances
Angular Clearance

- **Purpose:** allows slug or blank to drop through die
- **Typical values:** $0.25^\circ$ to $1.5^\circ$ on each side

![Diagram of angular clearance between two dies](image)
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

- 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 \( R \geq 2t \), \( K_{ba} = 0.50 \) (middle line)
V-Bending

- (1) Before bending
- (2) After bending
- Application notes:
  - Low production
  - Performed on a *press brake*
  - V-dies are simple and inexpensive
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.

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

Plastic + elastic deformation

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_JuT1pM
HW assignment

• Reading assignment: Chapters 15, 16

• Review Questions: 14.3, 14.6, 14.7, 14.8,

• Problems: 14.1, 14.2, 14.3,