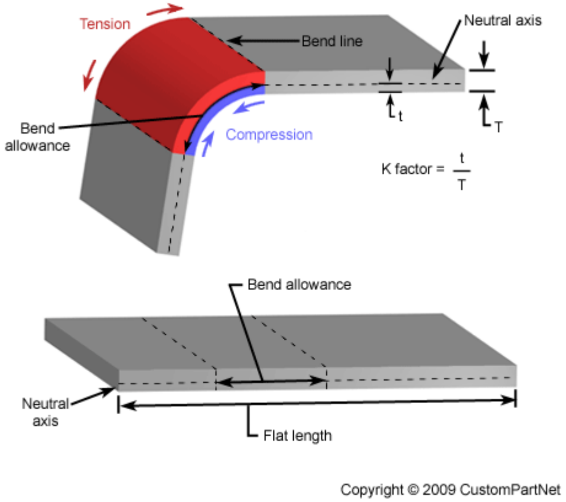
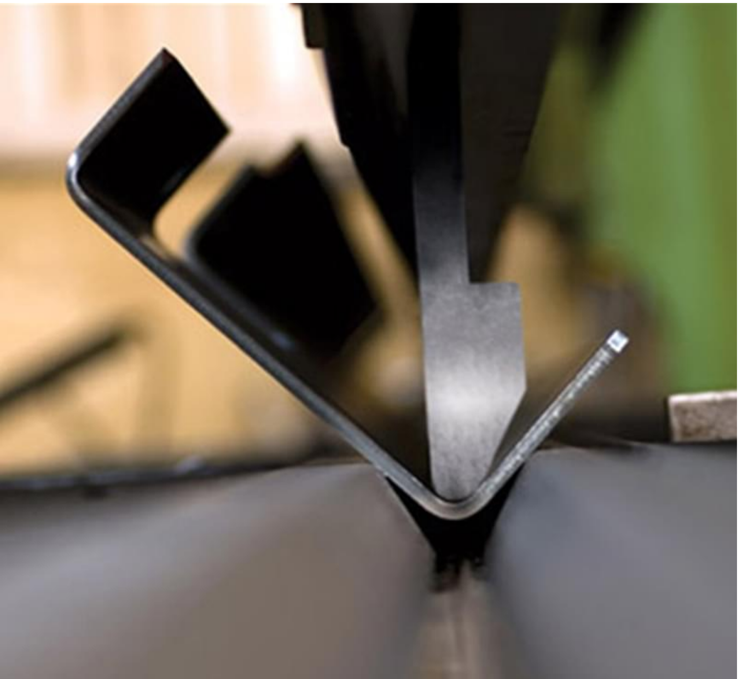
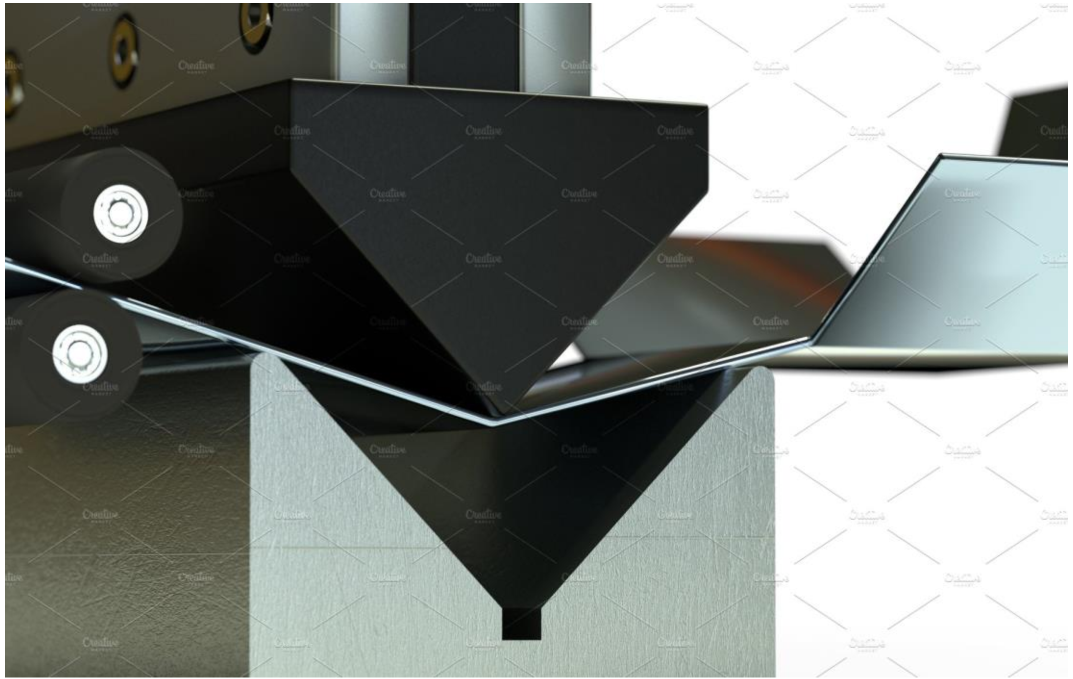
Sheet Bending Process and Die Design

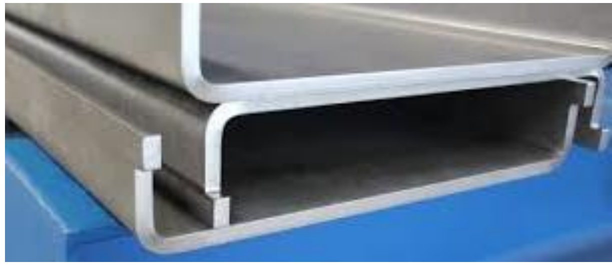
Dr. Dinesh Kumar,

PIE, NITJSR



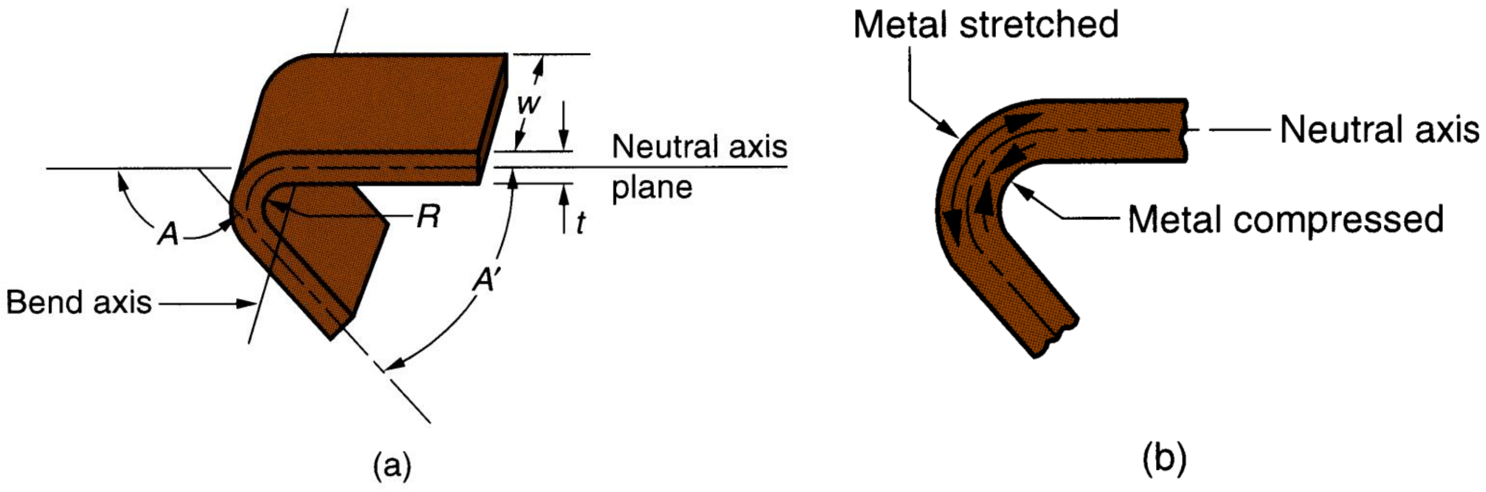






Bending

Straining sheet metal around a straight axis to take a permanent bend



(a) Bending of sheet metal

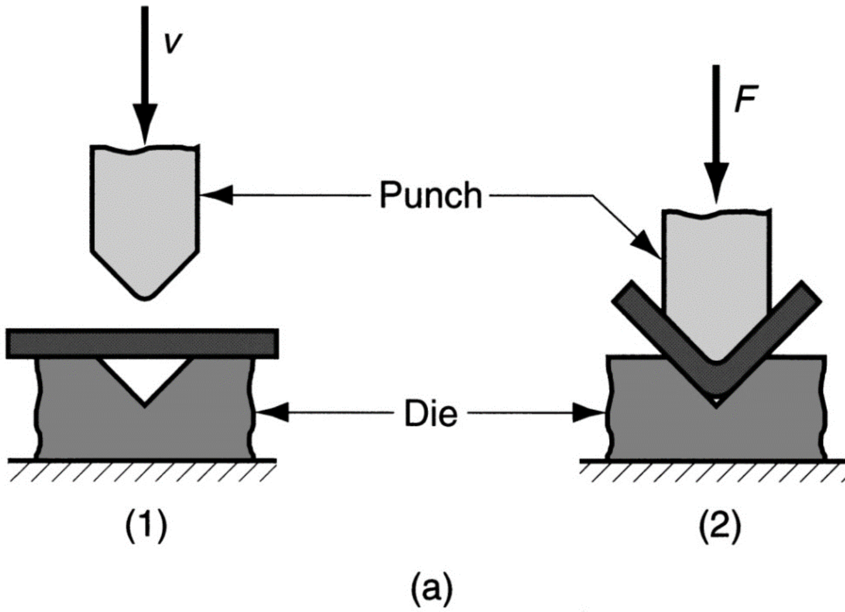
(b) both compression and tensile elongation of the metal occur in bending

Types of Sheetmetal Bending

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

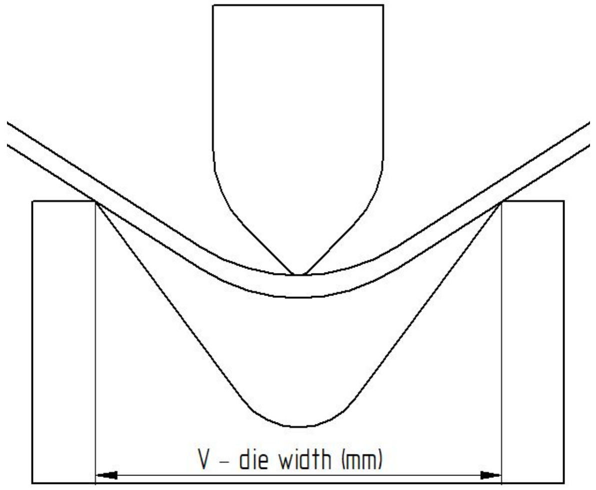
V-Bending

- V-bending is the most common bending method using a punch and die. It has three subgroups –
  - bottoming,
  - air bending and
  - coining.
- Air bending and bottoming account for around 90% of all bending jobs.
- For low production
- Performed on a *press brake*
- V-dies are simple and inexpensive



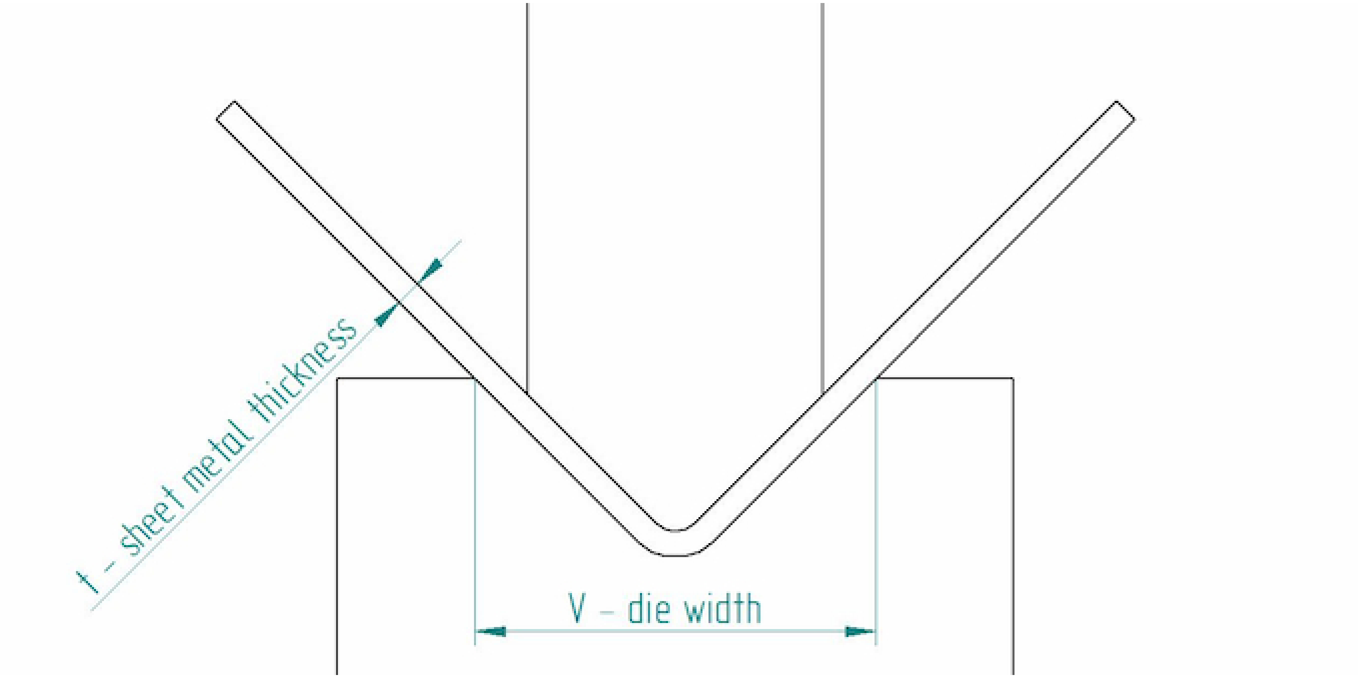
Air Bending

- Partial bending, or air bending, derives its name from the fact that the work piece does not actually touch the tooling parts entirely. In partial bending, the workpiece rests on 2 points and the punch pushes the bend.
- Air bending gives much flexibility. Let’s say you have a 90° die and punch. With this method, you can get a result anywhere between 90 and 180 degrees. Though less accurate than bottoming or coining.



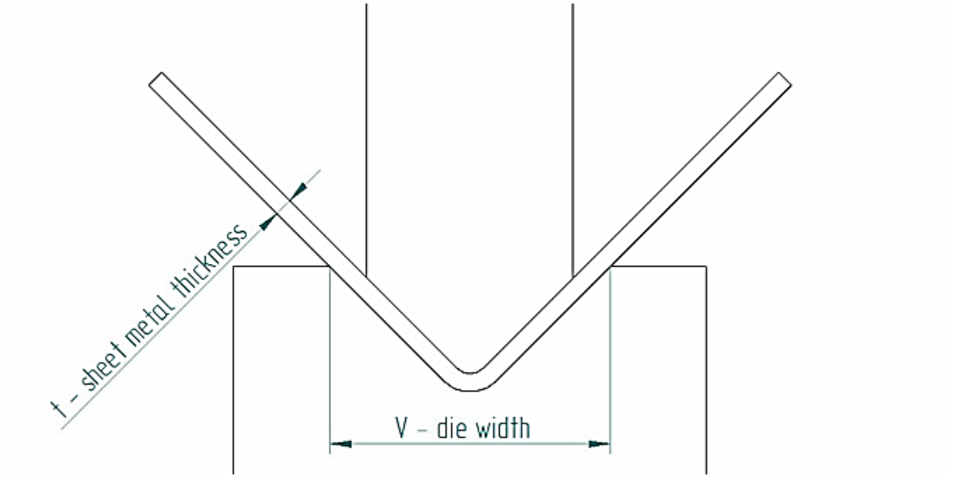
Coining

- Coining used to be far more widely spread. It was pretty much the only way to get accurate results. Today, machinery is so well controllable and precise, that such methods are not widely used any more.



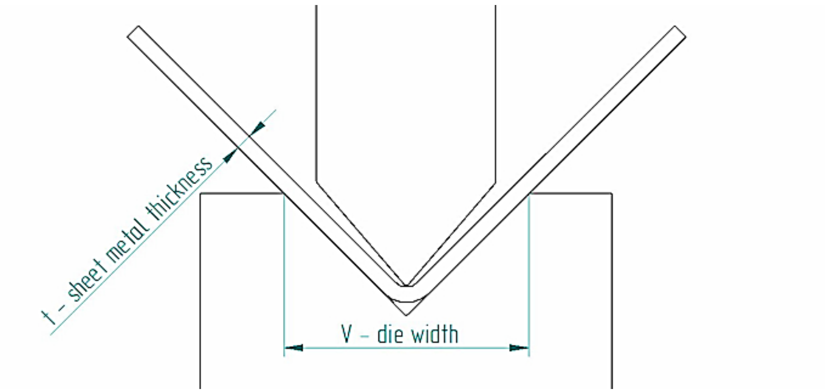
Coining

- Coining derives it name from coins, as they have to be identical to make fake money distinguishable from the real one. Coining, in bending, gives similarly precise results. For instance, if you want to get a 45 degree angle, you need a punch and a die with the exact same angle. There is no spring back to worry about.
- Why? Because the die penetrates into the sheet, pressing a dent into the workpiece. This, along with the high forces used (about 5-8 times as much as in partial bending), guarantee high precision. The penetrating effect also ensures a very small inside radius for the bend.



Bottoming

- Bottoming is also known as bottom pressing or bottom striking. As the name “bottom pressing” suggests, the punch presses the metal sheet onto the surface of the die, so the die’s angle determines the final angle of the workpiece. With bottoming, the inner radius of the angled sheet depends on the die’s radius.
- As the inner line gets compressed, it needs more and more force to further manipulate it. Bottoming makes exerting this force possible, as the final angle is preset. The possibility to use more force lessens the spring back effect and provides good precision.



Opening Width V (mm)

Method/Thickness (mm)	0.5... 2.6	2.7...8	8.1...10	Over 10
Bottoming	6t	8t	10t	12t
Air bending	12...15t			
Coining	5t			

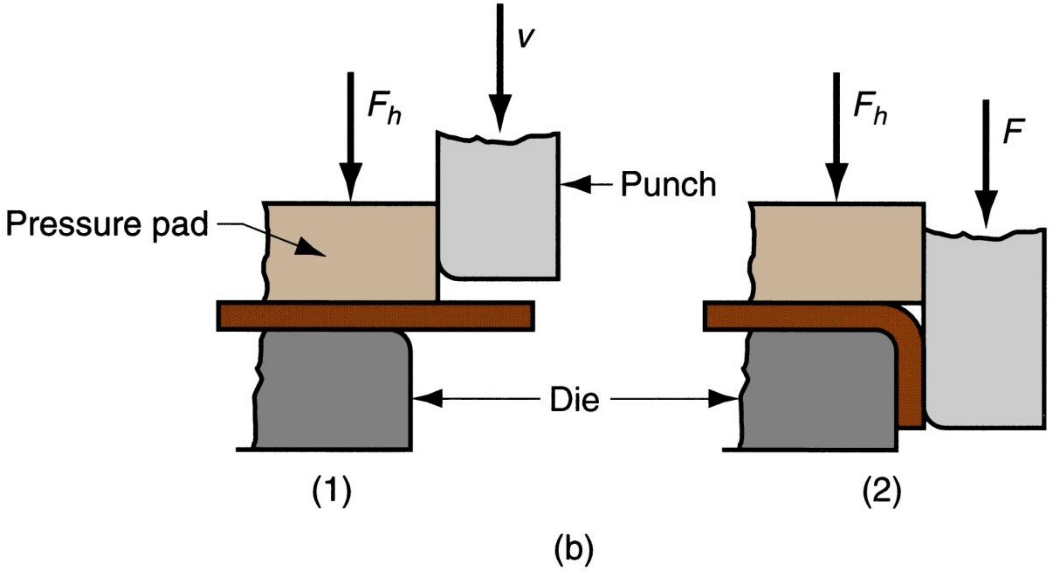
The inner radius has been experimentally proven to be around 1/6 of the opening width, meaning the equation looks like this: ir=V/6.

Bottoming

- Higher degree of accuracy
- Sharper inside corners are required
- Less spring back is achieved
- Requires 3-8 time more pressure than the air bending die
- Costlier equipment

Edge Bending/Wiping

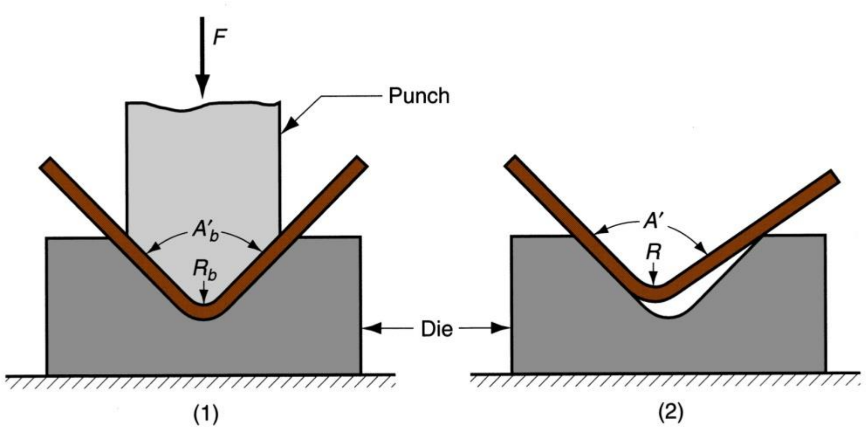
- For high production
- Pressure pad required
- Dies are more complicated and costly



Spring back 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



Stretching during Bending

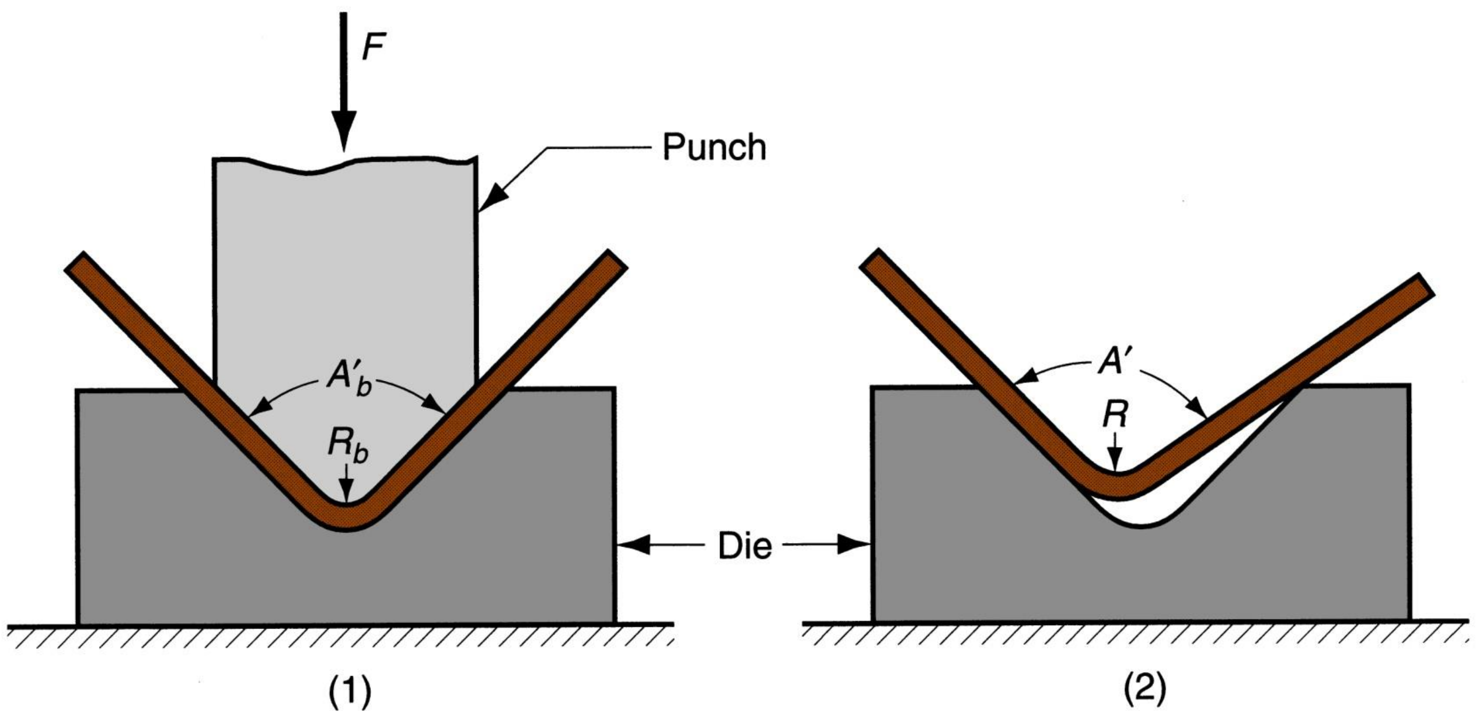
- If bend radius is small relative to stock thickness, metal tends to stretch during bending
- Important to estimate amount of stretching, so that final part length = specified dimension
- Problem: to determine the length of neutral axis of the part before bending

Bend Allowance Formula

$$BA = 2\pi \frac{A}{360}(R + K_{ba}t)$$

where *BA* = bend allowance; *A* = bend angle; *R*= bend radius; *t* = stock thickness; and K_{ba} is factor to estimate stretching

- If R < 2*t*, K_{ba} = 0.33
- If R $\geq$ 2*t*, K_{ba} = 0.50



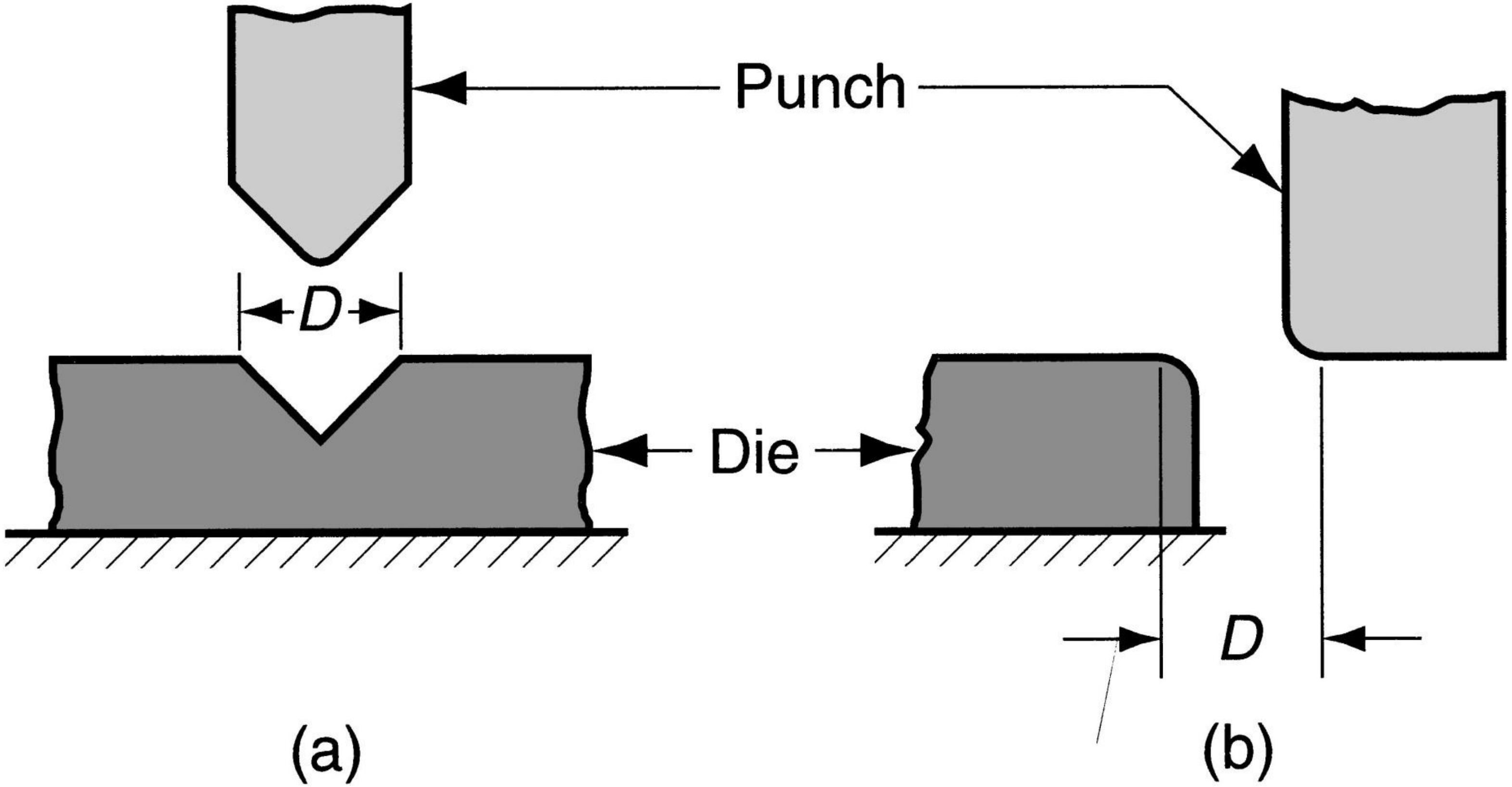
Springback in bending shows itself as a decrease in bend angle and an increase in bend radius: (1) during bending, the work is forced to take the radius R_b and included angle A_b' of the bending tool (punch in V-bending), (2) after punch is removed, the work springs back to radius R and angle A'

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


Punch
D
Die
D
(a)
(b)