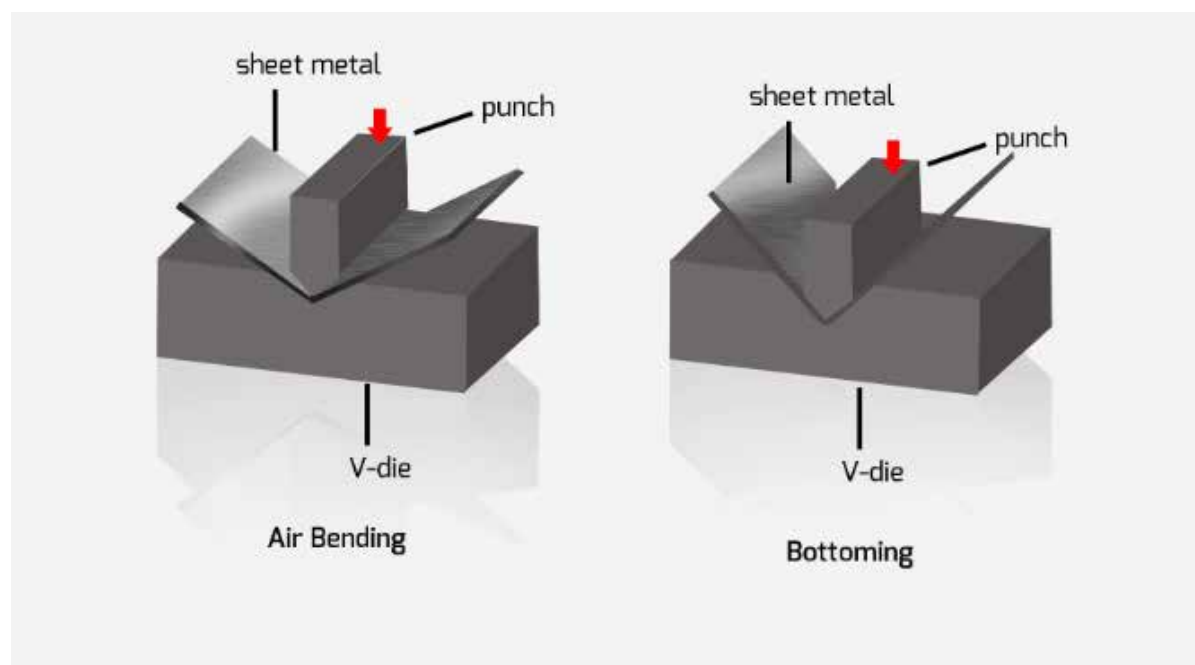


Differentiating Air Bending and Bottom Bending for Sheet Metal Fabrication

Air bending and bottom bending are two of the most common ways to create a bend during sheet metal fabrication. Both can be achieved using a manual or CNC controlled press brake.



	Air bending	Bottom bending
Custom tooling	Not required Single die can accommodate multiple angle radii	May be required Unique die is required for each angle radius
Angle accuracy	>± 45°	±15-30°
Best used for	Parts where bend accuracy isn't vital	Parts where bend accuracy is important
Pricing	No tooling investment / higher part price	Tooling investment / lower part price

What is air bending

With air bending, a press brake is equipped with a punching tool and commonly a v-opening bottom die. The punch presses the material into the bottom die, creating three contact points: one on each side of the top of the v opening of the die, and one at the tip of the punch. This defines the bend inside radius. The metal does not get pressed to the bottom of the die but rather, the depth is used to define the angle bend. Generally, the same die can be used to accommodate different materials, bend radii, and bend angles based on the punch profile and depth.

Air bent pieces will experience some spring back and angles are not highly accurate.



Advantages

- No retooling is required to create different bend angles, saving time and expense with the process
- Able to bend most material types and thicknesses
- Faster than bottom bending
- Less pressure is required to perform the bend
- Less contact with tooling results in less surface damage



Disadvantages

- Not well suited to parts requiring highly accurate bend angles
- Material will have a spring back effect which needs to be considered in the bend angle*
- Older press brake equipment often results in inaccurate bends*

**Unless your fabricator has the latest equipment with laser measuring technology*

What is bottom bending

Bottom bending also uses a punch and bottom v-shaped die but bends the metal by bringing the die and punch together. Because the material is pressed into the bottom of the die, the desired bend angle determines the specific die to be used. Bottom bending requires more pressure, generates less spring back, and creates more accurate angles. However, each bend radius will require a different bottom die.



Advantages

- More accurate angles than air bending
- Less spring back than air bending



Disadvantages

- Unique tools are required for different bend angles, creating a more complicated process
- Requires more pressure on the press brake than air bending
- Contact with punch and die can leave tooling marks