

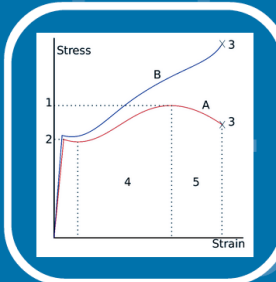


A bend is the permanent deformation of sheet metal under the effect of an external force, greater than the yield strength of the material.

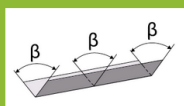
This force shapes the sheet metal beyond its elastic phase, until it reaches the plastic deformation zone (strain hardening region).

This force shapes the sheet metal beyond its elastic phase, until it reaches the plastic deformation zone (strain hardening region).

Different bending processes are used in a wide range of fields: automotive, furniture, doors, trains, construction, aeronautics, electronics, cranes, ships, etc.



- 1: Ultimate strength
3: Rupture
5: Necking region
B: Actual stress (F/A)
2: Yield strength (yield point)
4: Strain hardening region
A: Apparent stress (F/A_0)



- Accurate bending angle (β)
- Constant bending angle along the entire piece
- Constant bending angle regardless of length

SPRINGBACK

Springback is the elastic recovery of the material after forming load is removed, which tends to open the angle a bit. It can be compensating with overbending and special tools.

Read more on goo.gl/mIDvqf

CROWNING ERRORS

Crowning compensates for the press brake ram deformation. Sheet metal characteristics are highly variable, and cannot be accurately foreseen. For this reason it's important to use real-time crowning systems.



This is the most frequent V-bending method, commonly used in a wide range of productions.

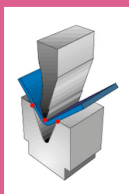
The punch forces the sheet metal down into a V-shaped die. The angle is defined by the 3 contact points with tool and die, and depends on the depth reached by the punch.

PROs:

- simplicity
- low bending force required
- possibility to bend very thick sheets
- possibility to obtain different angles with the same tools
- less expensive
- more flexible

CONS:

- precise punch depth positioning required
- difficulty in finding the correct bending angle due to sheet springback
- material characteristics have a greater final effect



The punch presses the sheet metal completely into the die, so that the punch, the sheet metal, and the die are coupled together. Punch and die have to fit together exactly. Sheet metal is forced to exactly follow the tool shape.

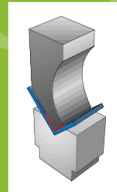
Coining is mainly used for producing 90-degree angles on thin sheet metal parts where small bend radius is required.

PROs:

- no springback
- higher precision
- smaller bending radius

CONS:

- different tool set for each angle/shape
- higher tonnage (5 times more than air bending)



PROs:

CONS:

