

BENDING FAQ 2

This time, following on from Technical Guidance Vol-PB.13, we will pick up and explain the content you inquired about.

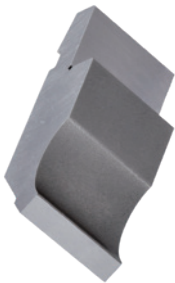
ABOUT TOOL SELECTION

Q: Is there a way to make tool selection easy?

A: When selecting a tool, it is necessary to consider the machine specifications, V-width, combination, bending order, and required pressure.

This time, we will summarize and explain the guidelines. For the basic tool selection procedure, see Technical Guidance Vol-PB.6.

[PUNCH SELECTION CRITERIA]



1. Interference: Use the “press brake tool cross-sectional shape sheet” and “bending limit graph” to check whether the tool work interferes and determine the tool shape. (See Technical Guidance Vol-PB.13)

2. Tip R: R of the product is determined by “ $r \cong V/6$ ”.
For the tip R, select an R that is slightly smaller than the inner R.
For the tip R, R0.2 is often selected, but...

- Advances in machinery have enabled highly accurate alignment.
- I want to make the tip angle gentle and minimize wear on the cutting edge.
For these reasons, there are cases where the tip R0.6 is used even when processing thin sheets.

3. Punch angle: (In case of 90° bending with bottoming)

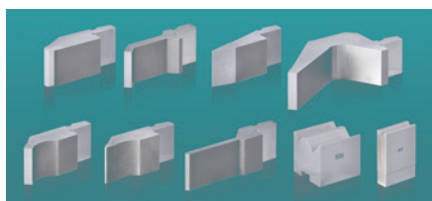
Material	Thickness	Angle	Effect
Mild steel plate / Thin plate	~2.0mm	88° · 90°	Small effect of spring-back
SUS / AL / Thickness plate		88° → 84° → 82°	Since the effect of spring-back is large, select a slightly sharp tip angle.

[DIE SELECTION CRITERIA]



Selection of 88° die and 90° die

Material tensile strength	88° for high material, 90° for low material
Spring-back value	88° for large, 90° for small
When performing coining	90°



The above is a basic guideline for selecting standard punches and dies. When processing, be sure to perform preliminary verification such as trial bending to confirm sheet metal quality and safety.

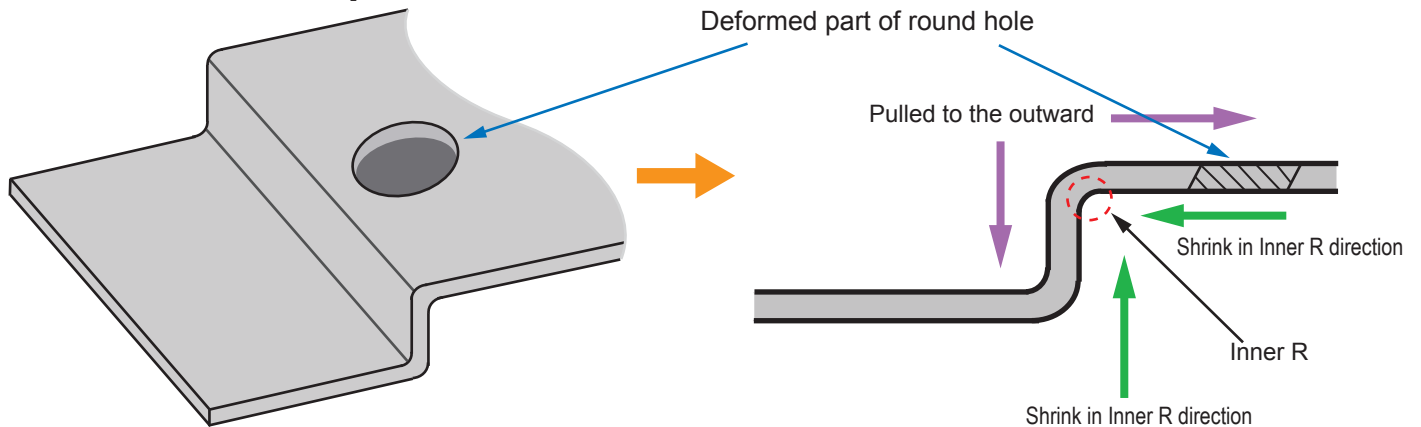
[On the hole deformation near the bending line]

Q: I want to perform burring near the bending line, but the pre-hole has been deformed.

How far should it be from the bending line?

A: The hole near the bending line is easily deformed.

[Hole deformation mechanism]

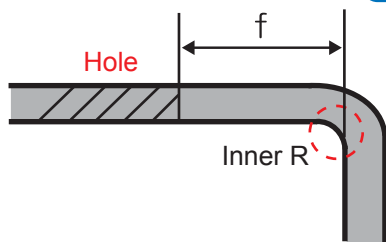


The hole near the bending line is deformed by the force shown above.

The smaller the inner radius, the greater the force that extends outward on the outer radius and the greater the degree of deformation.

The distance (f) that is not affected by hole deformation can be obtained by the following formula.

$$f = \text{material thickness} \times 1.5 + R$$



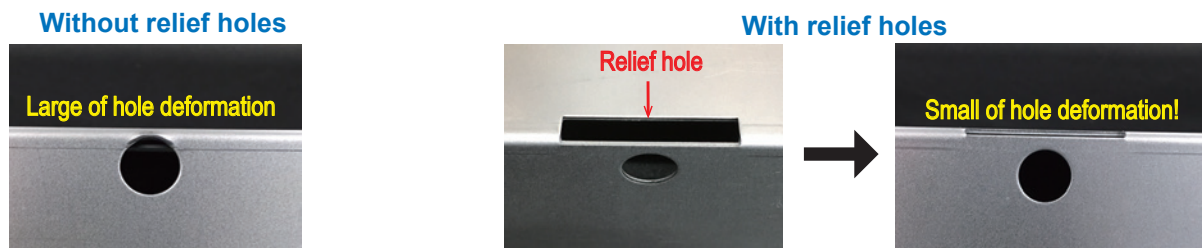
f: Distance from end of hole to inside of bend

Inner R: Inner R of bending

- ◆ You can calculate the distance that is not affected by the above formula, but it is a reference value because there are errors depending on the thickness, material, and hole size.
- ◆ For hole dimensions with tight dimensional tolerances, allow a margin of +1.0mm or more for the plate thickness from the f-value obtained by the above formula.
- ◆ “Sagging” and “burr” also affect the finished dimensions of holes. “Consider bending direction” and “Finish the cut surface nicely” are also effective.

COUNTERMEASURE [Add relief holes]

If you need to drill a hole near the bend line, you can reduce the deformation of the hole by forming a relief hole on the bend line.



Since the pulling force of the board is not transmitted to the hole, deformation of the hole can be suppressed. When making relief holes, consider the product appearance, relief hole shape and distance in advance.

For MORE information,
please contact
CONIC tool sales desk.

CONIC Co., Ltd.
10-5 Taiheidai, Shoo-cho, Katsuta-gun,
Okayama 709-4321 Japan
Email: tools@conic.co.jp
https://www.conic.co.jp

CONIC PRECISION Co., Ltd.
55/22 Moo 4, Buengkumphroy, Lumlukka,
Phatumthani 12150 Thailand
TEL: (662) 159-9870 FAX: (662) 159-9872
Email: conic_thai@conic.co.jp