

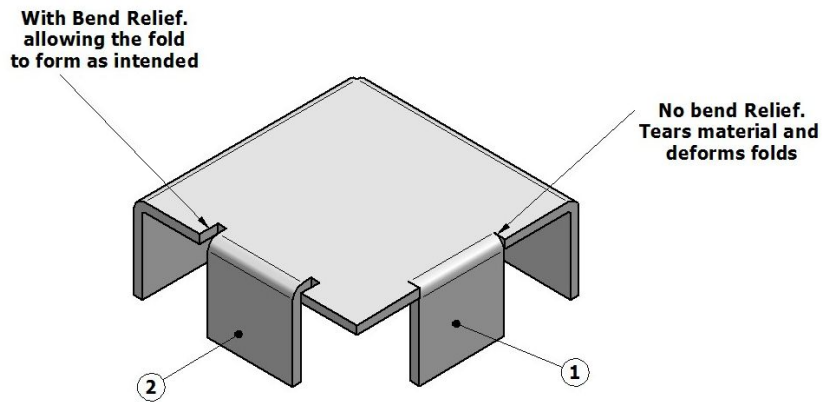
Sheet Metal Design Considerations

Tolerances

The punched feature-to-feature tolerance for Sheetmetal parts is +/-0.1mm, with a general folding tolerance of +/- 0.25mm

Bend Relief

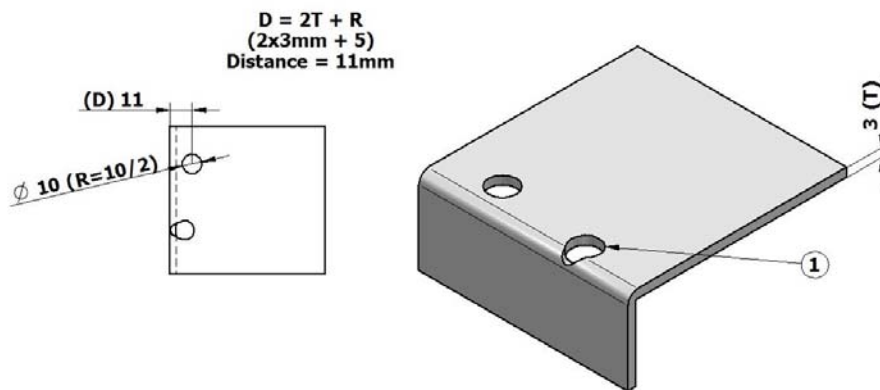
When a bend is made close to an edge the material may tear unless bend relief is given. Part 1 shows a torn part. Part 2 shows a bend relief cut into the part, the depth of the relief should be greater than the radius of the bend. The width of the relief should be a material thickness or greater.



Forming and Bending Near Holes

When a bend is made too close to a hole the hole may become deformed. "Hole 1" shows a hole that has become teardrop shaped because of this problem. To save the cost of punching or drilling in a secondary operation the following formulas can be used to calculate the minimum distance required:

For a slot or hole < 25mm in diameter the minimum distance "D" = 2T + R (see below)



Send us an e-mail: inquires@smds.co.nz



Give us a Call: 0064 (3) 388 3187 or Fax 0064 (3) 3993183



Corner Finishing

Type A

Standard Open Corner. A general-purpose corner for internal components where no finishing is required.

Type B

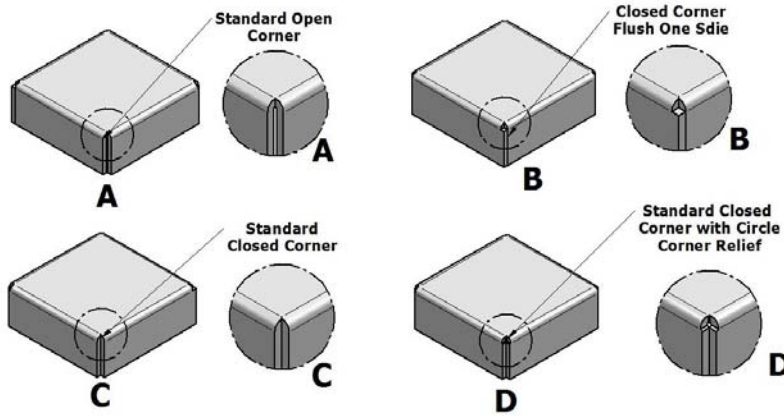
Closed Corner Flush One Side. For external finished corners where no welding is required but a clean tight corner is needed.

Type C

Standard Closed Corner. A general-purpose corner for component that requires a clean finish or fully welded finishing.

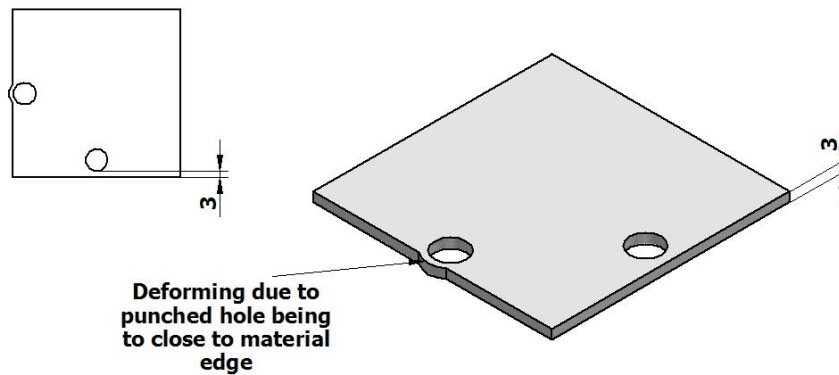
Type D

Standard Closed Corner with Relief. A general-purpose corner for component that requires a clean finish and folding relief to achieve tolerance and squariness



Hole to Raw Edge Clearance

A good rule of thumb for hole placement is to keep the hole at least one material thickness away from any raw edge. If the hole gets too close to an edge a bulge can form as shown below.



Send us an e-mail: inquires@smds.co.nz



Give us a Call: 0064 (3) 388 3187 or Fax 0064 (3) 3993183



Hole Taper

When a hole is punched, it does not have a constant radius through the entire thickness of the part. The taper in the bottom side of the hole is relative to the die clearance. Die clearance is the difference between the punch diameter and die diameter. It is usually about 10% of the material thickness. To get a constant diameter through the entire material thickness the part must be drilled or machined at far more costly operation.

Hole Diameter to Thickness Ratio

For most materials hole diameter should not be less than material thickness. As tensile strength increases the punch diameter must also increase.

Turret Punch Considerations

Special forms - The turret punch press offers many versatile solutions for putting special features into a flat sheet. Some of the options are counter sinks, offsets, card guides, Dimples, part Numbering, lettering, and Partial-shears. Virtually any form can be made as long as it falls within the following constraints. The form depth must be less than 8.0mm and it must fit into a 89.0mm diameter circle. Progressive tools can also be made for the turret press. These tools can punch and form small parts as efficiently as a progressive punch press tool with a lower initial cost.

Send us an e-mail: inquires@smds.co.nz

∇

Give us a Call: 0064 (3) 388 3187 or Fax 0064 (3) 3993183