



# Design Criteria Guidelines

Closer tolerances can be provided when necessary in the function of the part, but remember, extra operations required can increase your cost.

These tips are based on tooling methods developed by most metal stamping companies and provide quality stampings which meet all commercially accepted standards.

## Standard Tolerances

### **Blank dimensions**

Allow  $\pm 0.15\text{mm}$  on all decimal dimensions up to 150mm in length, with an additional  $\pm 0.08\text{mm}$  for each additional 75mm. All dimensions should be measured on the die side of the blank.

### **Flatness**

Standard tolerance is within 0.1mm per linear 25mm of surface. If a greater degree of flatness is required, specify it on the print as an additional operation.

### **Pierced hole tolerance**

Specifications on size of holes should be held to not less than  $\pm 0.05\text{mm}$  or its equal, as determined by the punch side of the opening. For most economical tooling and production, location of related holes should be determined from a centre line of the blank and with a tolerance of  $+ 0.05\text{mm}$  between hole centres.

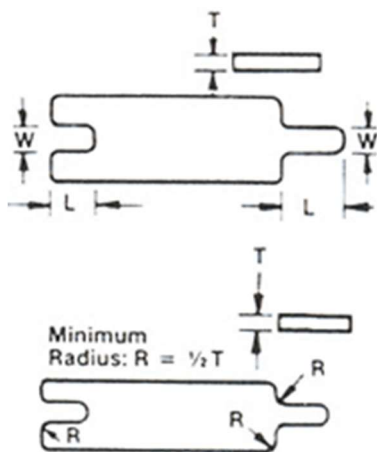
### **Finish on holes**

Always specify the degree of finish desired for the holes, such as reaming and deburring.

### **Tolerances on hole diameters**

Unless otherwise specified, tolerances are considered to apply to the punch side only. All punched holes have breakage caused by clearance between the punch and die. This will vary with temper and material differences. Where a finished hole is required through entire stock thickness, indicate this on the print so hole can be punched undersize and reamed to proper size and finish.

# Blank Design



## Width

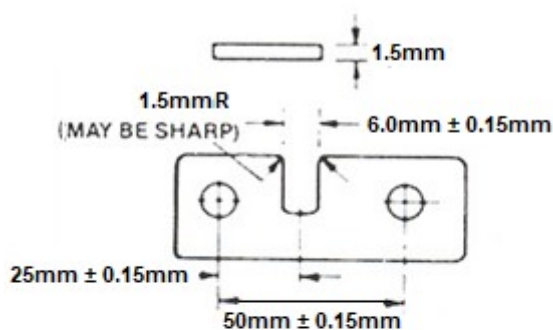
Width of minimum blank sections - never less than  $1\frac{1}{2}$  to 2 times material thickness; minimum of 0.8mm on materials under 0.5mm thick.

## Length

Length of minimum section - not more than 5 times width.

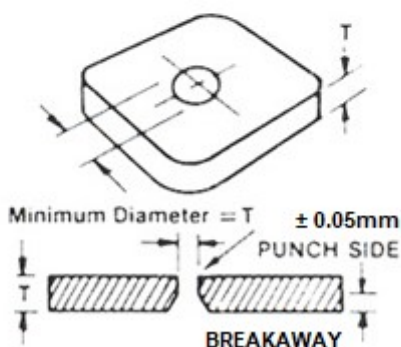
## Corner Radii

Material thickness of 1.6mm or less will require a 0.38mm radius minimum. For thicker materials, allow a minimum of  $\frac{1}{2}$  material thickness.



## Notches

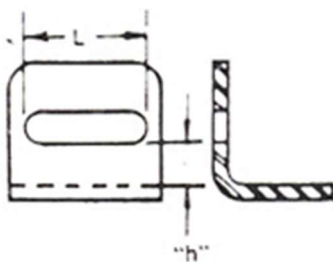
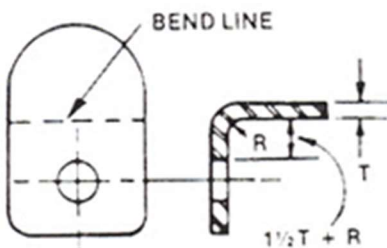
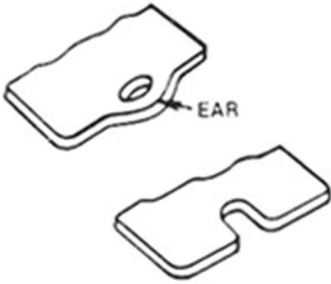
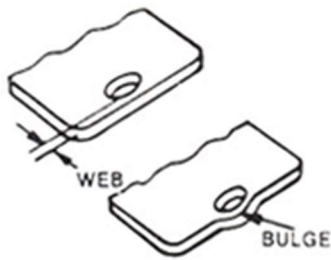
Where a notch requires close size and/or location tolerances, a maximum radius should be allowed if included in the blanking operation. A sharp corner would be economical where the notching is a separate operation.



## Minimum Hole Diameters

Holes can be punched most economically when the hole diameter is  $1\frac{1}{2}$  times greater than the stock thickness. Minimum diameter is related to the shear strength of the material. In softer materials this can be equal to or less than stock thickness. Stainless steel, on the other hand, would normally require a hole diameter equal to 2 times stock thickness. If the hole diameter is less than material thickness (or less than 1.0mm dia.), it must normally be drilled and the burr removed.

# Punching Holes and Openings



## Web Allowance

For punched holes, the web should be a minimum of the stock thickness; a narrower web will result in progressive bulging. This is hardly visible until the web is reduced to less than  $\frac{1}{2}$  the stock thickness. These conditions also apply to the web between holes.

## Alternative Methods

If bulging will be a problem, try adding an ear; or change the blank profile to include the hole as a notched area. Where the above adjustments cannot be made, separate drilling and deburring, punching or notching operations may be necessary to eliminate a measurable bulge.

## Adding Slots

Slots are dimensioned for size by overall length and width dimensions and for locations by dimensions to their centerlines. The web allowance should be provided as for holes.

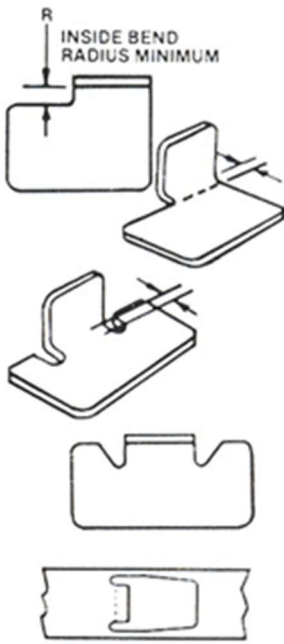
## Opening Adjacent to Bend

Minimum inside distance from edge of a hole to bend is  $1\frac{1}{2}$  times material thickness plus the bend radius.

## Opening Parallel to Bend

The more elongated the slot, the greater the required distance between edge of slot and the bend. Where an opening is parallel to a bend, the following allowances should be made for economical tooling.

# Bending



## Allowance for 90° Bends

To avoid tearing of the material in forming a right-angle bend, design the blank profile to allow offset relief where possible. This will eliminate the chance of fatigue under stress and increase the possibility of using stock tooling - not possible when the form is inside the blank profile.

## Notches for Bend Relief

Notches can be included in the blanking operation. Their width should be  $1\frac{1}{2}$  to 2 times material thickness. Design notches either as a shallow loop or angled one side.

## Internal Tabs

Relief notches around an internal tab when possible will reduce costs. Relief notch width should be  $1\frac{1}{2}$  to 2 times material thickness, 0.5mm minimum.

## Allowance for Height/Square Edges

Minimum inside height for a form with a 90° bend should equal 2 times material thickness plus the required bend radius. Closer dimensions will require adding extra stock followed by an extra cutting operation after forming.

## Allowance for Distortion

When the normal distortion caused by bending will interfere with a mating part, the print should specify a secondary operation for its removal. A simple method for eliminating the distortion resulting from compression is to narrow the part at the bend line.

## Allowance for Fracturing

On difficult or heavy forming material where the burr side of the blank must be on the outside of the bend, ample inside bend radii should be allowed to avoid fracturing. When slight fractures are permissible, the print should be so marked.

