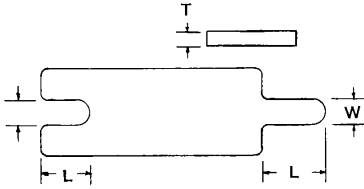


Tonnard Manufacturing Corporation

STAMPING DESIGN TIPS FOR USE IN SHORT RUNS

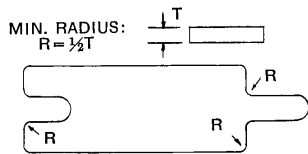
The designing tips shown below are for your use in ordering parts with the greatest economy based on Tonnard's Short Run Tooling Methods. They are commercially accepted to Tolerance Standards. Closer Tolerances are available, and in so doing it will be necessary to perform extra operations, which will in turn increase your cost.

BLANK DESIGN

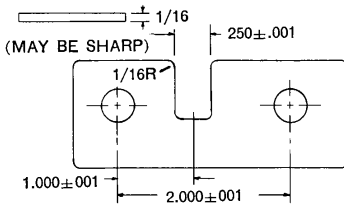


WIDTH of minimum blank sections—never less than $1\frac{1}{2}$ to 2 times material thickness; minimum of $\frac{1}{32}$ " on materials under .020 thick.

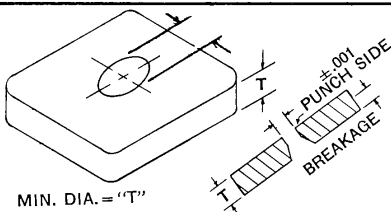
LENGTH of minimum section—not more than 5 times width.



CORNER RADII—Radii most always preferable. Corners may be sharp if material thickness is $\frac{1}{16}$ " or less. For thicker materials allow a minimum of $\frac{1}{2}$ material thickness.

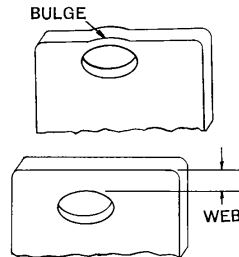


NOTCHES—Where a notch requires close size and/or location tolerances, a maximum radius should be allowed if included in the blanking operation. A sharp corner would be more economical where the notching is a separate operation.

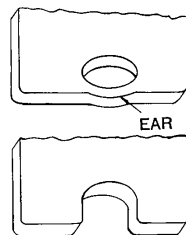


MINIMUM HOLE DIAMETERS—Holes can be punched most economically where the hole diameter is $1\frac{1}{2}$ times or greater than the stock thickness. Minimum diameter is related to the shear strength of the material. In softer materials this can be equal to or less than stock thickness. Stainless steel, 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 .032 dia.) it must normally be drilled and the burr removed.

PUNCHING HOLES AND OPENINGS



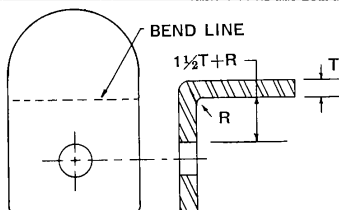
WEB ALLOWANCE—For punched holes the web should be a minimum of stock thickness; a narrower web will result in progressive bulging. This is hardly visible until the web is reduced to less than $\frac{1}{2}$ the stock thickness. (These conditions also apply to the web between holes.)



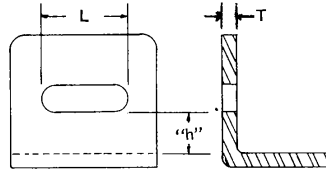
ALTERNATIVE METHODS—If bulging will be a problem, try adding an ear; or change the blank profile to include the hole as a notched area. Where the above adjustments cannot be made, separate drilling and deburring, punching or notching operations may be necessary to eliminate a measurable bulge.



ADDING SLOTS—Slots are dimensioned for size overall length and width dimensions and for location by dimensions to their centerlines. The web allowance should be provided as for holes. Use standard fraction slots where possible. Special slots cost extra.



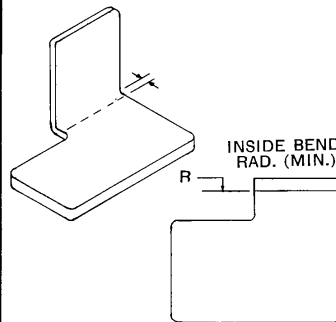
OPENING ADJACENT TO BEND—Minimum inside distance from edge of a hole to a bend is $1\frac{1}{2}$ times material thickness plus the bend radius.



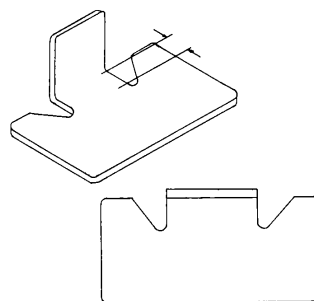
OPENING PARALLEL TO A BEND—The longer the slot the greater the allowance which must be made in the distance between edge of slot and the bend. Where an opening is parallel to a bend, the following allowances should be made for economical tooling:

MINIMUM REQUIREMENTS—
When "L" is up to 1" "h" = $2T + R$
When "L" is 1" to 2", "h" = $2 \cdot 1/2 T \times R$
When "L" is 2" or more, "h" = $3T$ or $3\frac{1}{2} T \times R$

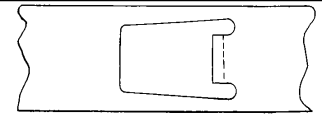
BENDING



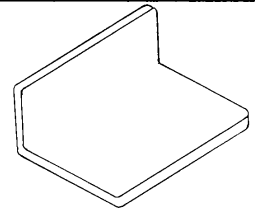
ALLOWANCE FOR 90° BENDS—To avoid tearing of material in forming a right angle bend, design the blank profile to allow offset relief where possible. This will eliminate the chance of fatigue under stress and increase the possibility of using stock tooling. This is not possible when the form is inside the blank profile.



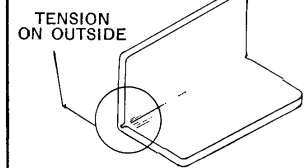
NOTCHES FOR BEND RELIEF—Notches can be included in the blanking operation at very little extra cost. Their width should be $1\frac{1}{2}$ to 2 times material thickness. Design notches either as a shallow loop or slanted one side.



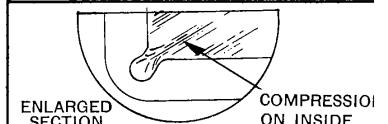
INTERNAL TABS—The slot can be punched around the entire tab to permit bend relief but this requires an additional operation. If other punched openings are required, the top slot can be punched at the same time.



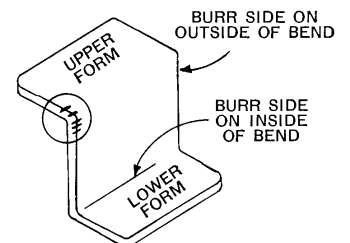
ALLOWANCE FOR HEIGHT—Minimum inside height for a form with a 90 degree bend should equal $2\frac{1}{2}$ times material thickness plus the required bend radius. A closer tolerance will usually require adding extra stock followed by an extra cutting operation after forming.



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.



SQUARE EDGES—to maintain a 90 degree bend with satisfactory square edges, the minimum inside formed height should equal $1\frac{1}{2}$ times the material thickness plus the inside forming radius ($1\frac{1}{2} T + R$).



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.