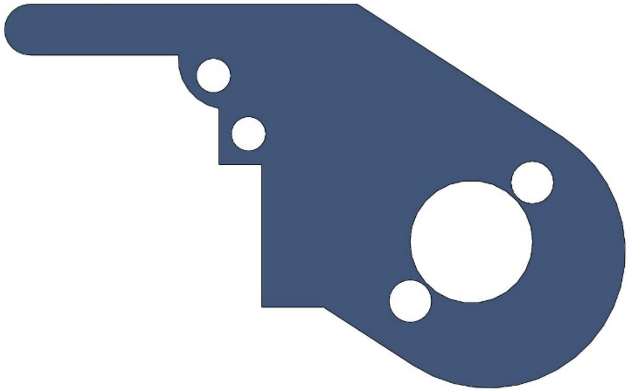
Metalforming Design Handbook

Cost Effective Design Principles

Dayton Rogers

est. 1929 MANUFACTURING CO.



Things Learned from over 90 Year making Metal Stampings

TABLE OF CONTENTS

Dayton Rogers Mfg. Co. acknowledges, with gratitude, selected illustrations and editorial material supplied by Precision Metalforming Association.

Note: All dimensions are in English unless otherwise specified.

Problem Solving is the First Step Toward Better Metal Stampings & Fabrications

Dayton Rogers views its business as a problem solving service, in which metal stampings and fabrications are components or end products that fulfill specific requirements. Our professional staff help design and manufacture quality parts that will give you a competitive edge.

Our approach to every job includes:

1. Design Assistance — using the latest computer assisted equipment

2. Inspection to your level of need, including statistical process control during manufacturing process

3. Cost-effective tooling process

4. Continued link between you and all levels of our facilities through an experienced sales engineer

5. Problem solving since 1929 (we wrote the book)

Technical Seminars Conducted At Your Place of Business

In addition to the above, Dayton Rogers also offers free Technical Seminars conducted at your facility.

These informative presentations are staffed by our technical engineering experts and augmented with videos, photos, actual samples of stamped and fabricated parts. Ample time is allowed for questions and answers. We encourage you to bring your drawings and specific design problems for discussion.

Contact us today at 800-677-8881 or **www.daytonrogers.com** to learn more! You may also follow us on **LinkedIn**, **Twitter** and **Facebook**

Short Run Stampings Can Save — Because:

TOOLING COSTS are kept to a minimum. Our standard shape punches meet many demands. Standard forming tools make a wide variety of shapes. Other standards allow us to build your parts quickly at a minimum cost.

DESIGN CHANGES are most easily incorporated with short run tooling. This minimizes costly scrap and re-tooling.

TOOLING TECHNIQUES for the short run process at Dayton Rogers are unique. Our large tool rooms use design software, CNC Milling and Wire E.D.M. machines and other special tools to speed the production of our dies.

QUANTITY Short runs do not cause you an unnecessary penalty.

QUALITY of stamped parts are more reliable than most other methods. Our Quality Assurance Departments are well-equipped to assure your parts meet print specifications.

PRICE Only a comparison of quotes will prove our point. Let us quote your projects.

DELIVERY is assured. We have presses up to 300 tons, 4000 watt laser equipment, turret presses to 30 tons and press brakes to 10´. Our large metal inventory makes two million pounds of stock immediately available. A great way to get both delivery and price. Production control methods include a daily check of job status to ensure that orders are shipped "on time" or ahead of schedule.

1. *How to Use Short Run Stampings*

Short run stampings offer an economical way to produce parts in quantities from prototype to 100,000 pieces with short lead times. It's an ideal method for checking your design, assembly process and market acceptance of your new product . . . all with minimum investment.

Let Us Help

We review thousands of prints each year searching for ways to save our customers money. Usually our suggestions involve material specifications, dimensions and tolerances.

Tolerances

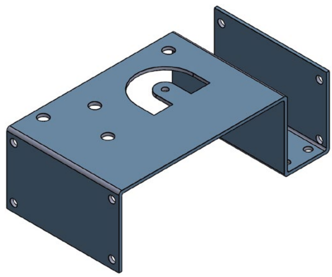
Consider the necessity of specific tempers or closer material tolerances. If a part requires a specific temper or closer material tolerances, strip steel is desirable. However, if temper or close material tolerances are not a factor, sheet steel is available at a lesser cost. Tolerances should be no tighter than necessary to make a part functional. Close or tight tolerances result in **high cost** to build tools and **additional operations** to hold required tolerances.

Typical Fabrication Methods Compared to Short Run Stamping

METHOD	COMMENT	SUGGESTION
Progressive	For parts where long runs with no design changes exist. High tooling, low piece price.	Consider short run tools to prove design and start up quantiles. Short run tools also provide vital backup.
Fine Blanking	Flatness and full shear edge condition. Costly tooling that can wear.	We can tool for a fraction of the fine blanking cost on many shapes, less the full shear edge.
Casting	Complex shapes are readily produced. Part cost is not as low as stamping and holes and other dimensions are not as easy to control.	Careful engineering or re-design can allow stampings to replace more costly investment castings.
Machining	Very complex shapes. Extremely expensive for more than a few parts. Waste of material.	Depending upon your application, short run tooling can give you alternate design at a much lower cost.
Laser Cutting	Great method for small lots and proving your design.	With complex forming short run tooling can utilize laser blanks.
Turret	For certain parts the Turret is an ideal method. Short run typically holds closer tolerance and there is no scalloping on the curved edges.	Short run tools can be competitive at an early stage in your production. These tools can further reduce the cost of tool up and provide back up.

Typical Parts Price Comparison of Variations in Materials and Design/CAD

Comparing cost impact of the several variables is difficult. To help explain the impact, we have selected a sample part and shown the price range that you could reach depending on how you control your design.



LOW COST DESIGN

Tolerance	+ .010 on centers — + .005 on hole diameters
Material	16 ga. 1010 sheet — standard tolerance .059 + .004
Bends	Radius inside = to material thickness
Hole distortion	Print should note “distortion permissible” if hole is too close to a form (see page 16.)
Outside Corners	1/16” radius allowed
Deburr	Break sharp edges

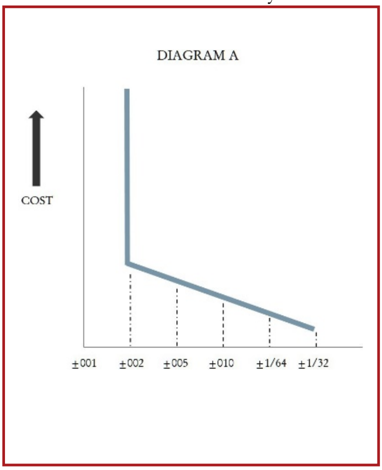
HIGH COST DESIGN

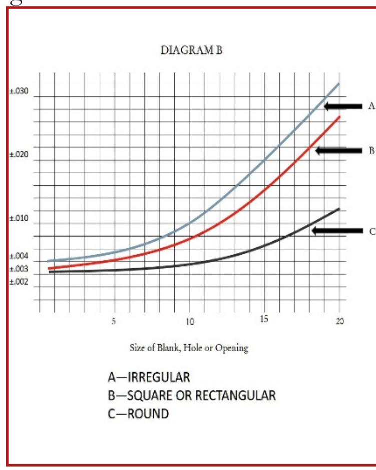
Tolerance	+ .002 on centers — + .001 on hole diameters
Material	1010 strip steel — .059 + .003 thickness
Bends	Inside .000 to .015 radius
Hole distortion	Not allowed
Outside corners	0 -.015 radius allowed
Deburr	.010 Max/Min

NOTES

2. *Tolerances*

Practical tolerances for our short run metal Stampings vary according to the design of the part. Generally, all dimensional tolerances compared to costs can be illustrated by the following diagram “A”.





Reading from right to left, we illustrate that within the more generous limitations, the cost increases slowly—but, as the permissible variance is reduced, the cost increases more rapidly—and as the limits are still further reduced, the cost increases sharply to the extent that it might be impractical to stamp. There is a definite breakoff for stamping operations beyond which machining, grinding, lapping, honing, and other precision operations must be employed.

On diagram “B”, we indicate the relation between tolerance and size for flats, or holes and openings in flat blanks, for the most economical tools and production. These tolerances can be held on the die side of the part and the punch side of the holes and openings.

For the most economical tooling and production of related holes, the **hole location** is referred to as the **dimension between holes** and hole location is determined from a centerline of the blank.

Using these interpretations, we can hold plus or minus .005 tolerance between hole centers with the lowest priced tooling. For plus or minus .002 tolerance between hole centers, slightly higher priced tooling would be used.

In contrast, close hole locations measured to the blank contour could require expensive dies and additional operations.

3. *Edge Conditions*

On ferrous materials, the clearance between punch and die should be about 8% to 10% of the material thickness, which may vary due to hardness and thickness. The punch and die clearance will differ on various non-ferrous materials.

Example #1 illustrates the results that will be expected from material like cold rolled sheet steel, commercial quality, that is less than 1/4 hard temper; or with a shear strength of approximately 45,000 psi.

A slight pulldown "A" of the blanked edge and a straight sheared section "B" for about 25% to 30% of the material thickness will occur on the die side.

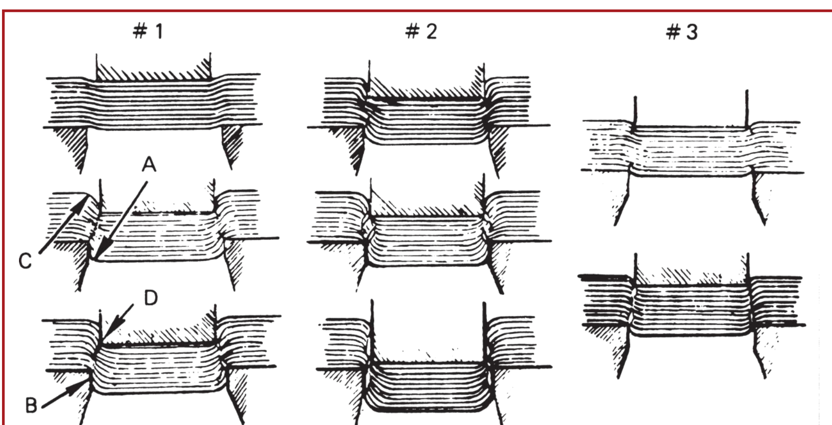
The opposite will occur upon piercing as indicated by "C" and "D" on the punch side.

The balance of the thickness will have breakage.

More pulldown and a greater sheared edge will occur for softer material.

Example #2 indicates the results of less than the usual clearance. Increased blanking pressure will be required and double breakage on the blanketed edge will occur. At times double breakage on thick parts might be more desirable than 8% to 10% taper.

Example #3 indicates that on very hard materials, the pulldown is negligible and the sheared edge would possibly amount to only 10% of the thickness of the material.



4. *Blank Design*

a. Minimum practical section should never be less than material thickness or .060. A minimum section must be one and one half times material thickness for high shear strength material for the most practical stamping.

b. Radii on blank corners — Corners can be sharp if material thickness is 1/16 or less — over 1/16 allow corner radii (R) equal to 1/2 material thickness. See illustration.

c. Practical Design for Economy Manufacture.

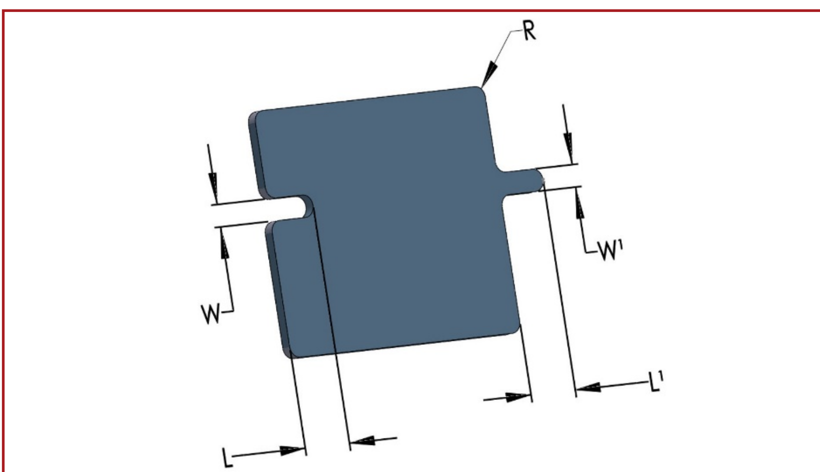
W = .060 minimum for materials thinner than .060 wider if possible.

W_1 = Never less than material thickness, wider if possible.

L = 5 x W is maximum depth, should be less if possible.

L_1 = 5 x W is maximum length, should be less if possible.

The above rules (**a** and **b**) apply for maximum economy. If followed, all blank periphery can be included in the blanking die, eliminating secondary tooling and operations.



5. *Piercing Round Holes*

To pierce holes with economical tools and operations, the hole diameter must not be less than the stock thickness. If the hole diameter is less than the stock thickness (or less than .060) it usually must be drilled and deburred and each of these operations is slower than punching.

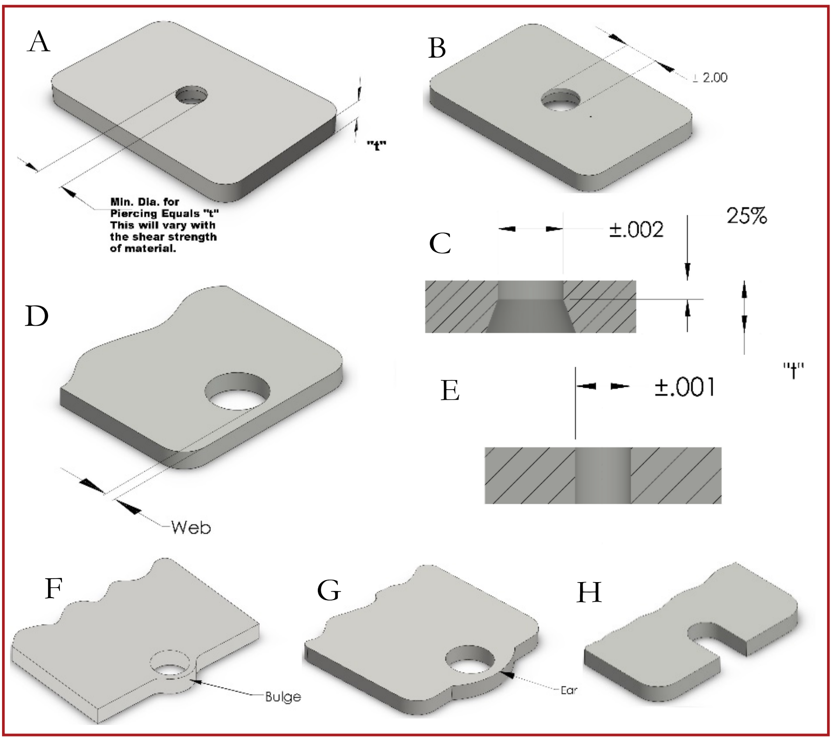


Illustration “B” indicates a hole diameter with a tolerance of plus or minus .002”. We can pierce a hole within these limits on the **punch side** for approximately 25% to 30% of the material thickness as indicated in illustration “C”. The percent of thickness varies with the shear strength of the materials.

On holes where a machine finish is required, they can be punched undersized, drilled and reamed to size eliminating die breakage as shown. (See Illustration “E”.)

If the web (distance between the hole and edge of material) is a minimum of the stock thickness, the hole can be punched which is less expensive than drilling and deburring. (See Illustration “D”.)

A web that is less than the stock thickness will result in a bulge on the blank. Bulge conditions would increase progressively as the web decreases, until there would be a complete break-through. However, the bulge is hardly visible until the web is reduced to less than 1/2 the stock thickness. These examples would also apply to a web between holes. (See Illustration “F”.)

If a measurable bulge is not permitted, a drilling and deburring operation may be necessary.

As a suggestion, if the web is too narrow, the profile of the blank could be changed by adding an ear of sufficient dimensions and shape to eliminate the problem. (See Illustration “G”.)

Another suggestion would be to change the contour of the blank to include the hole as a notch. (See Illustration “H”.) The notch could either be pierced or be wide enough so it could be included in the blank without a piercing or notching operation. (See Blank Design section, page 10)

NOTES

6. *Definition of Punched Holes*

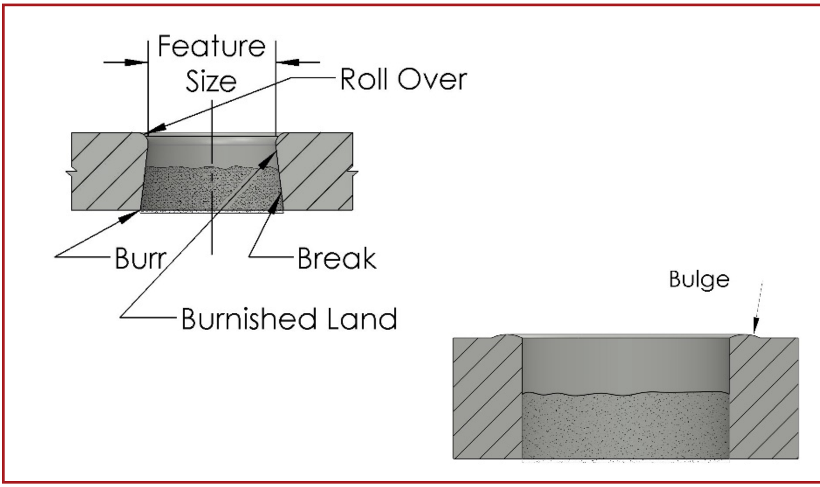
Identification of these punched holes can provide the basis for determining the inspection parameters.

Roll-over is the natural consequence of:

1. The punching process.

2. The mechanical properties of the material being punched.

3. The die application of techniques employed.



7. *Specification and Measurement of Punched Holes*

1. Dimension feature size limits only.

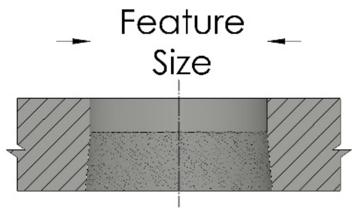
2. Feature size is to be measured only in the burnished land area or punch side.

3. Shape deviations within the feature size limits are permissible.

Special Purpose Holes

1. Dimension only those elements that affect part function.

2. When dimensioned:
   a. Burnished land to be specified as minimum only.
   b. Burr height to be specified as maximum only.



8. *Recommended Minimum Ratios of Punched Hole Diameters to Stock Thickness*

Limitations are established in common practice for most economical production. Recommended ratios are applicable to all common metals.

P = Punched Hole Diameter
(0.062 min. dia.)
T = Stock Thickness

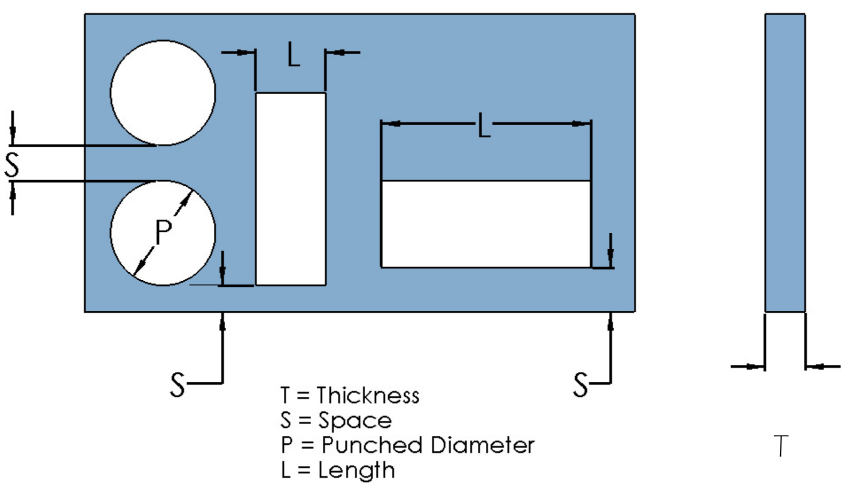
Material Ultimate Tensile Strength	**Ratio P to T**
32,000	P = 1.0T
50,000	P = 1.5T
95,000	P = 2.0T

9. *Spacing of Hole Edge to Part Edge and Adjacent Holes*

Economical minimum spacing of punched holes to a straight edge or other hole is established. Distortion is not more than 5 percent of metal thickness.

T = Thickness
S = Space
P = Punched Diameter
L = Length

Avoid minimum values of "S" where possible.

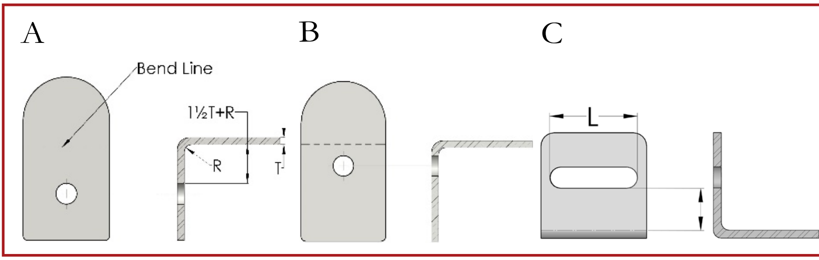


Avoid Minimum Values of "S" Where Possible.

Length "L" is less Than	Punch Dia. "P" is less than	Space "S" is Minimum
	5T	1.5T
	10T	2T
10T		2T
25T		4T

10. *Piercing – Adjacent to Bends*

Illustration "A" indicates that the minimum inside distance required from the edge of a hole to a bend is 1-1/2 times the material thickness (T) plus the bend radius (R).



Otherwise, distortion will occur as indicated in Illustration "B" — or piercing after form must be considered.

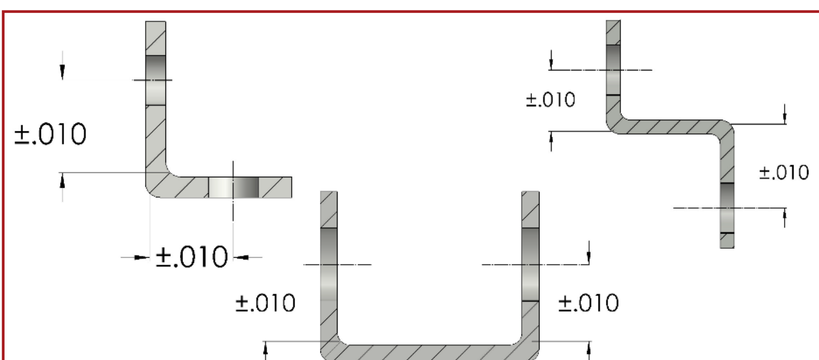
Illustration "C" indicates a similar condition to "A", except for openings with an edge parallel to bend. In this case the following requirements apply for economical tooling and production:

When "L" = up to 1 — 2T + R (minimum)

When "L" = 1 to 2 — 2-1/2T + R (minimum)

When "L" = 2 or more — 3T to 3-1/2T+ R (minimum)

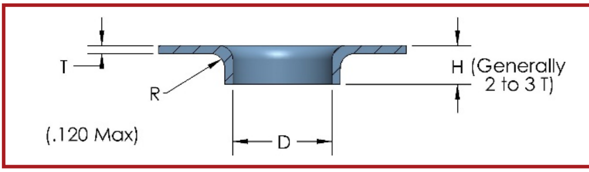
Minimum recommended tolerances are shown below: (on multiple bends, each bend is made separately).



11. *Definition and Limits of Extruded Holes*

An extruded hole is formed by punching a smaller hole and then flanging the sides.

R = Inner radius
H = Flange height
D = Inner diameter
T = Material thickness

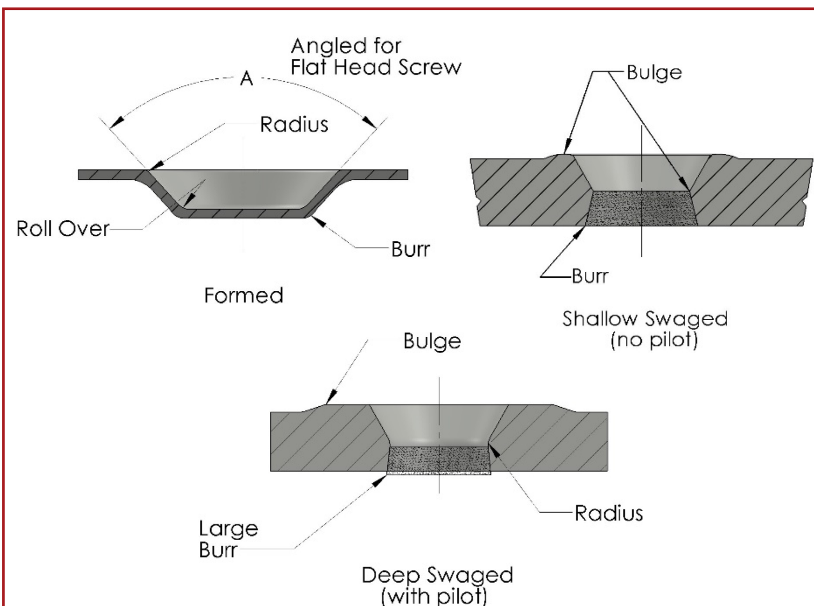


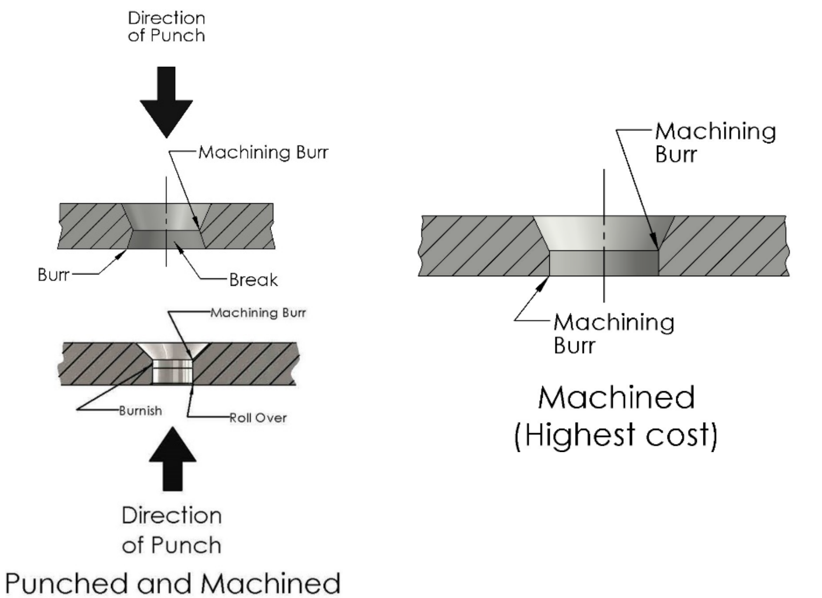
12. *Common Methods of Countersinking*

Various methods of countersinking produce certain characteristics.

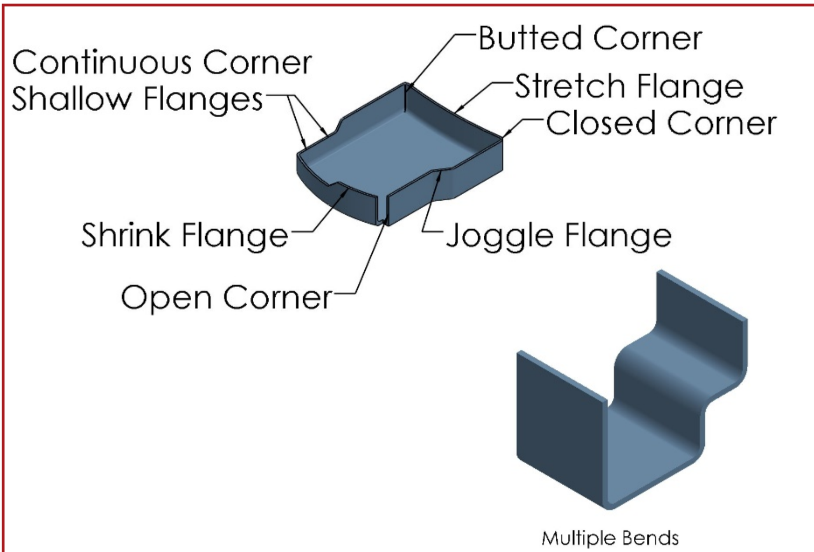
Countersinking produces a bevel or flare providing a surface or a recess for the head of a fastener.

Most common methods used are:





13. *Elements of Formed Stampings*



14. *Specifications and Measurement of Formed Parts*

Preferred dimensioning and points to measure:

L = Linear dimensions

R = Radii

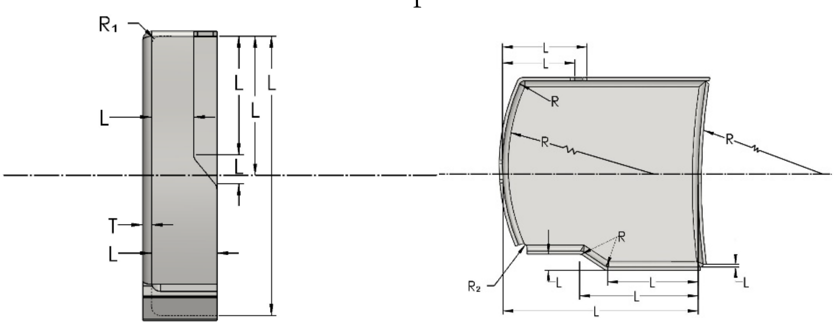
R1 = Typical inside bend or

R2 = Radius in flat blank

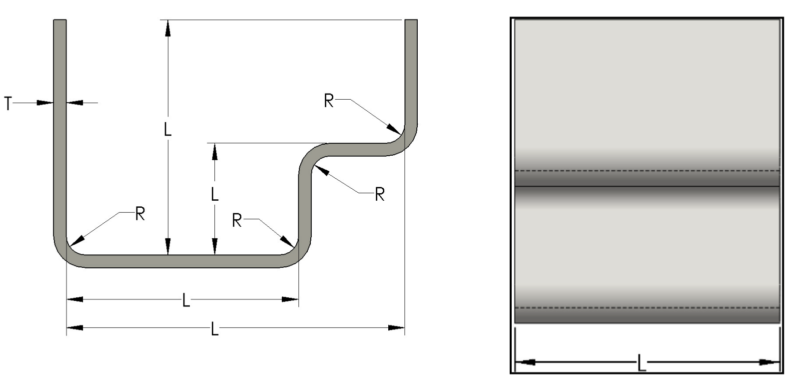
T = Material thickness

See two typical examples below.

Example 1



Example 2



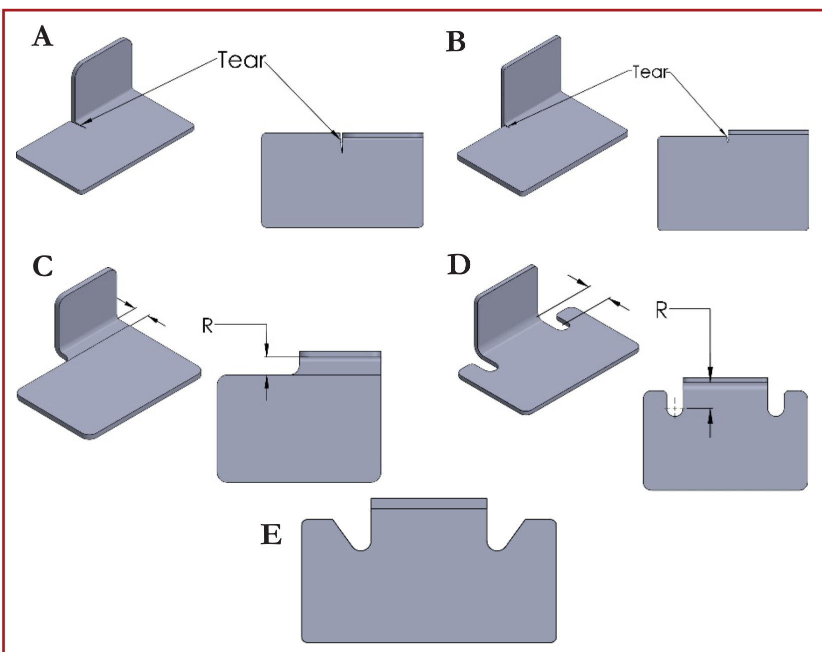
15. *Position of the Form*

Diagram "A" illustrates a design that is not desirable for quality or economy. When the form is inside the blank profile, as shown, the material must be torn through the stock thickness and the bend radius. If the part is under stress, this tear will likely cause fatigue failures. In addition, stock tooling cannot be adapted because the flat area adjacent to the form must be held in position during forming, which means extra tooling expense.

Diagram "B" illustrates a similar condition but with the form just outside the blank profile. In this case the tear extends to the center of the required bend radius.

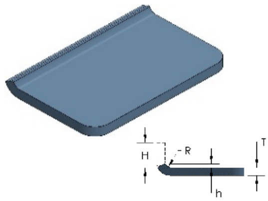
Diagram "C" and "D" illustrate a possible solution by changing the blank profile to provide relief for the bend. Besides eliminating the chance of fatigue under stress, there is a possibility of using stock 90 degree vee punches and dies. The results are better quality and less expensive engineering charges.

If the relief notches in illustration "D" are wide enough compared to the material thickness and the shear strength, or are designed like the relief in illustration "E", they can be included in the blanking operation for very little engineering cost and **no extra operation.**



16. *Height of the Form*

A



B

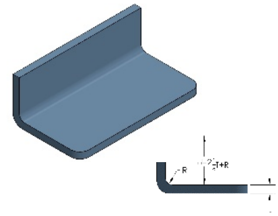


Diagram "A" illustrates a 90 degree bend with insufficient height (H) to form properly.

Consequently, stock must be added so the form is high enough (H), stock is then cut-off, which means additional tooling and an additional operation.

If (H) is not high enough, the cut-off tool may not have sufficient strength to stand up for a particular material or thickness. This may result in a higher cost secondary operation such as milling.

Illustration "B" indicates how to determine the minimum inside height (H), which in this case equals 2-1/2 times the material thickness (T) plus the required bend radius (R).

The concept illustrated by "B" above is converted to chart form below for convenience. These recommended minimum formed height dimensions are general to cover most variables of design, size material types, tempers and thicknesses. Properly designed small parts and easily formed materials such as aluminum, brass, copper, and mild steel may be formed with a slightly lower minimum inside formed height (roughly 20% less).

MINIMUM INSIDE HEIGHT OF FORM

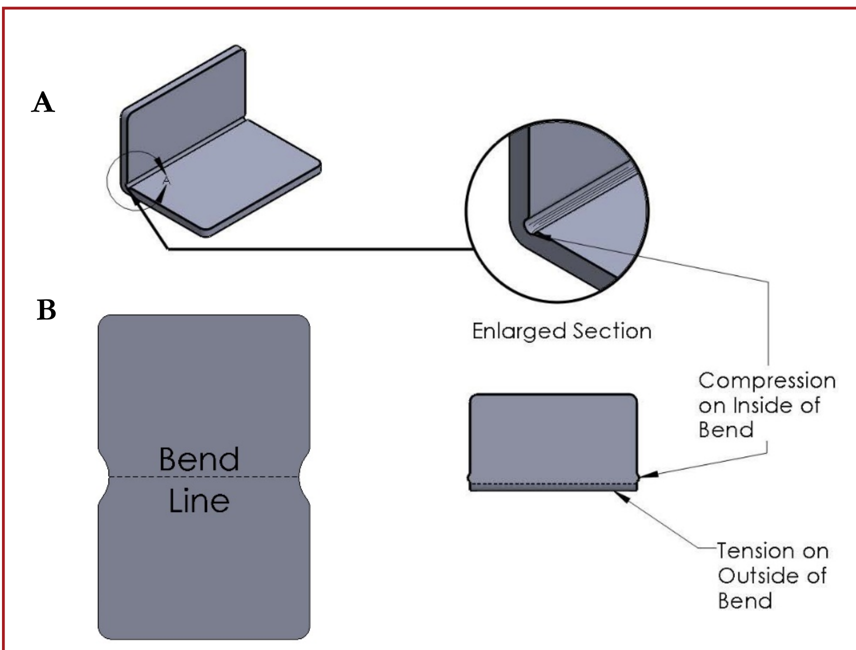
Stock Thickness	Inside Bend Radius				
	Sharp	1/32"	1/16"	3/32"	1/8"
.031	.078	.109	.140	.171	.203
.062	.156	.187	.218	.250	.281
.093	.234	.265	.296	.328	.359
.125	.312	.343	.375	.406	.437
.156	.390	.421	.453	.484	.515
.187	.468	.500	.531	.562	.593

17. *Distortion and Interference of Forming*

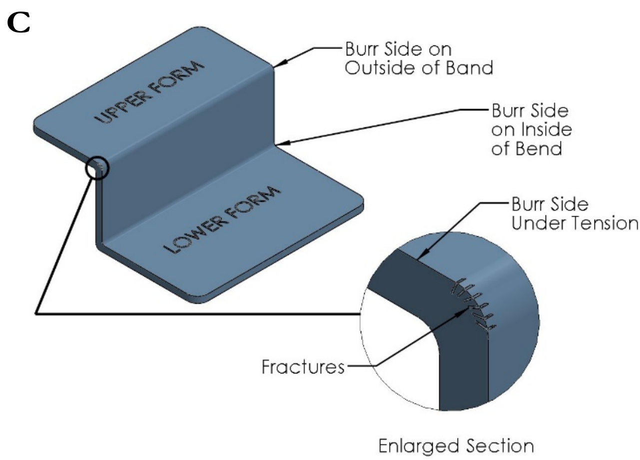
"A" illustrates a distortion condition that occurs in forming. It is a noticeable distortion when heavy material is bent with a sharp inside bend radius. On material thicknesses less than 1/16, or when the inside forming radius is large in comparison to the material thickness, distortion is barely noticeable.

The material on the inside of the bend is under compression which results in this bulge condition on the edges. In addition, the edges on the outside of the bend are under tension and tend to pull inside.

This bulge or distorted condition is usually of no concern and is accepted as standard practice unless the bulging will cause any interference with a mating part. This interference should be referred to on the print so a secondary operation can be considered to remove it. The extra operation may not require tooling but it will add to the cost of production.



"B" illustrates a blank developed to present interference resulting from the bulge without extra production cost.



The upper left hand form (enlarged section) in Illustration "C" indicates a fracture condition that occurs when the burr side of the blank is on the outside of the bend.

This fracture condition occurs because the burr side of the blank on the outside of the bend is under tension and causes the minute fractures on the sharp edge to open up and in extreme cases become visible.

Blanks should be produced so the burr side will be on the inside of the bend which is under compression like the lower form. However, when print requirements prevent this, or when a bend is in an opposite direction, like the upper form, fractures may occur.

Tumbling or deburring before forming can minimize the fracture in most cases. On extra heavy material with a very sharp inside bend radius, or on materials difficult to form, like SAE4130, tumbling before forming may not be adequate. It may be necessary to hand file or disc sand a radius on the sharp edges. Such secondary operations will add to the cost of production.

Therefore, for the most economical production and if design will permit, ample inside bend radii should be permitted for heavy and difficult forming material when the burr side of the blank must be on the outside of the bend.

If slight fractures are permissible it should be indicated on the print or inquiry.

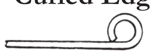
18. *Edge Conditions of Formed Parts*

1. "**As Formed**" — This is most common and is formed from a controlled size and shape blank.

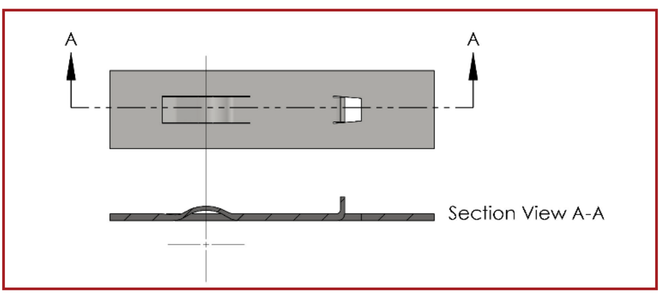
2. "**Die Trimmed**" — This is usually simple die trimming and parts have edges similar to flat stampings.

3. "**Hemmed Edge**" — This is formed for smoothness and stiffness.

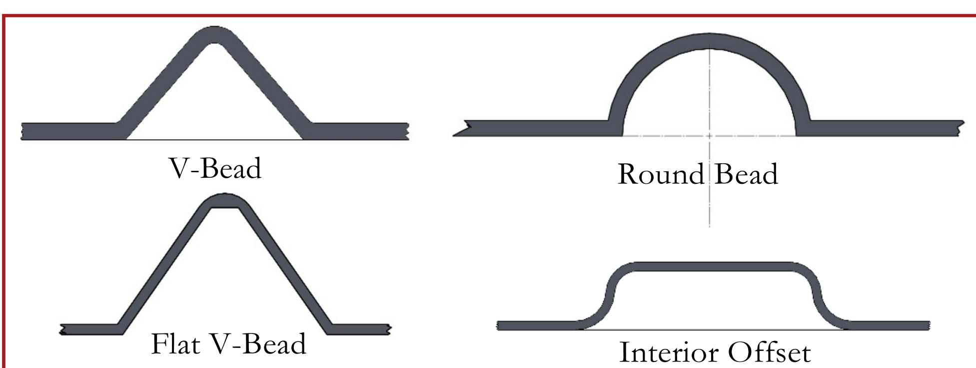
4. "**Curled Edge**" — Commonly used for hinges.

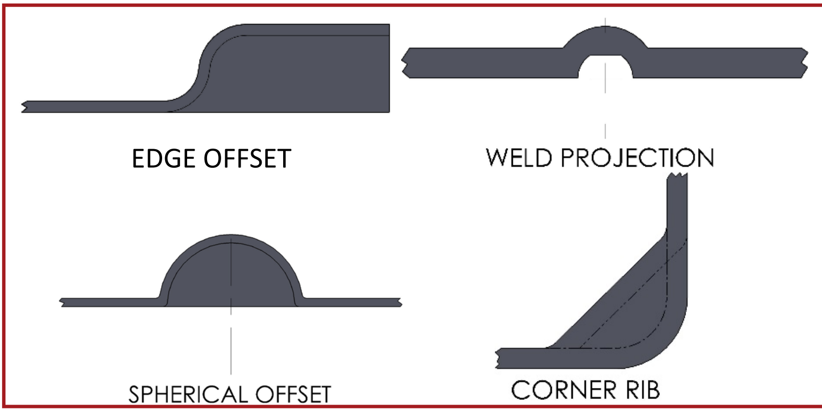


5. "**Lance-Formed**" — Edges are likely to have more burr than flat stampings.



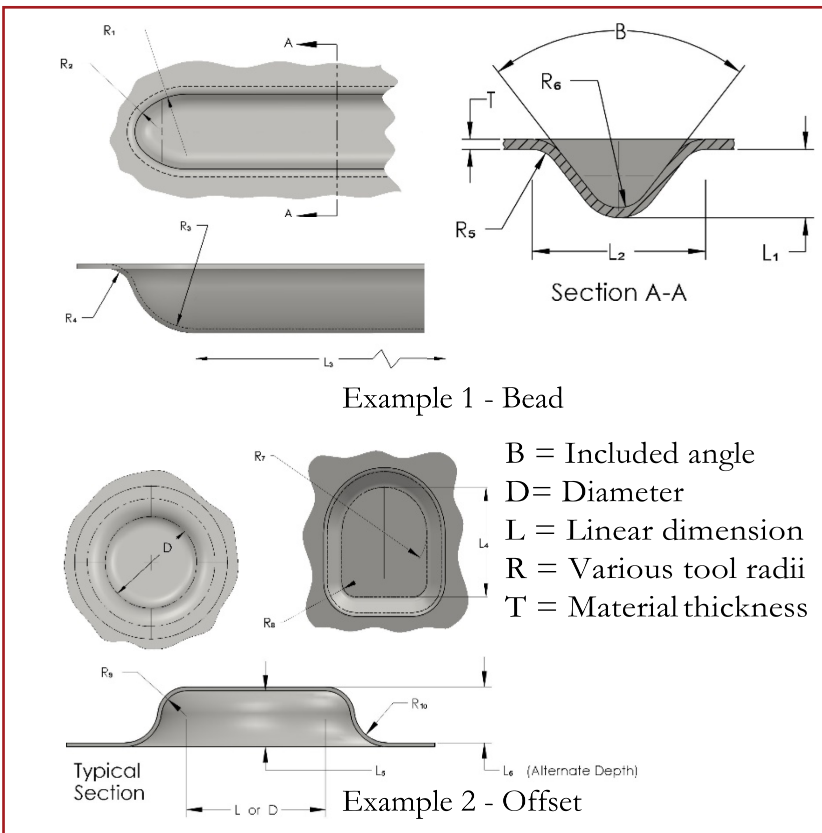
19. *Styles of Embossing*





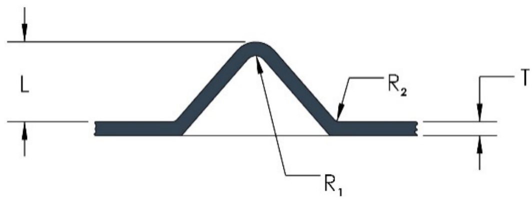
20. *Specification and Measurement of Embossed Parts*

Preferred dimensioning and points to measure on embossed parts.

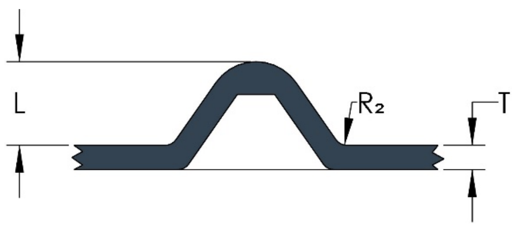


21. *Recommended Limits of Embossed Parts*

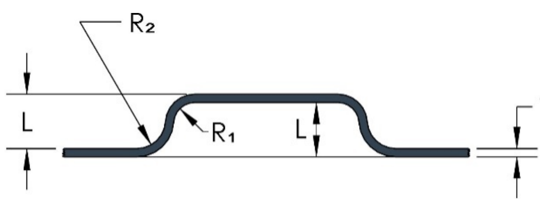
Limits for depths of embossments to minimize fracturing



Flat V-Bead L(Min)=3T*



Flat V-Bend L(Min)=3T*



Offsets

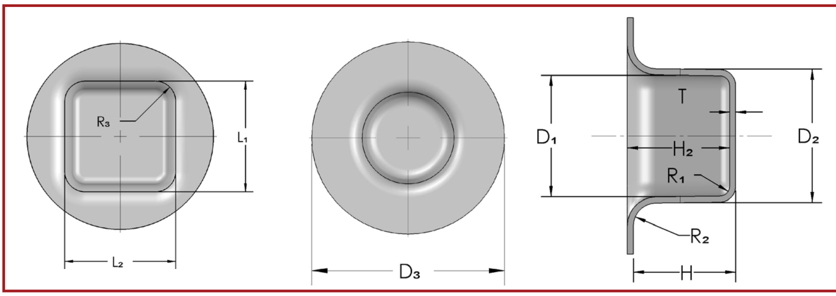
$L(Min) = R_1 + R_2$

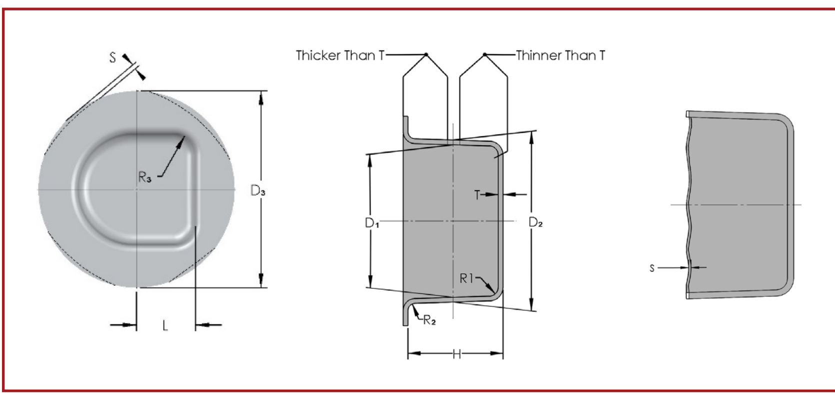
* Reduce to 2T for half hard tempers and alloys of aluminum.

22. *Specifications and Characteristics of Drawn Parts*

The specification should show the form of the part, state the material, specify dimensions and condition of symmetry.

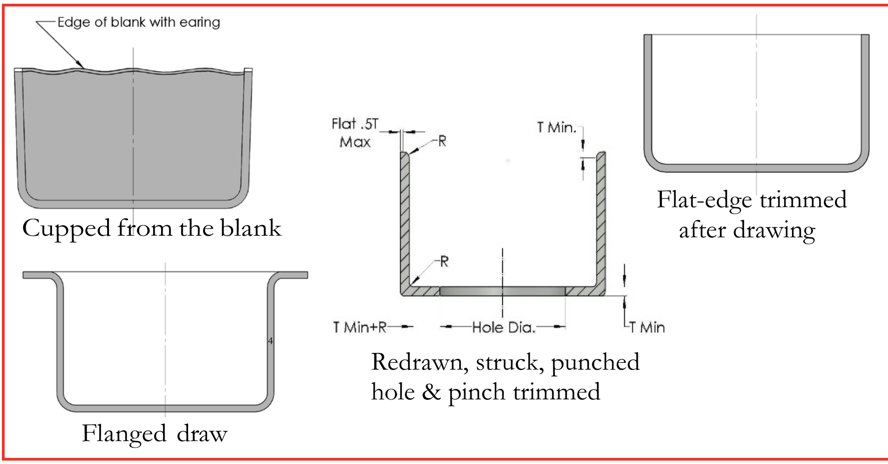
D[1] or **D**[2] (not both)





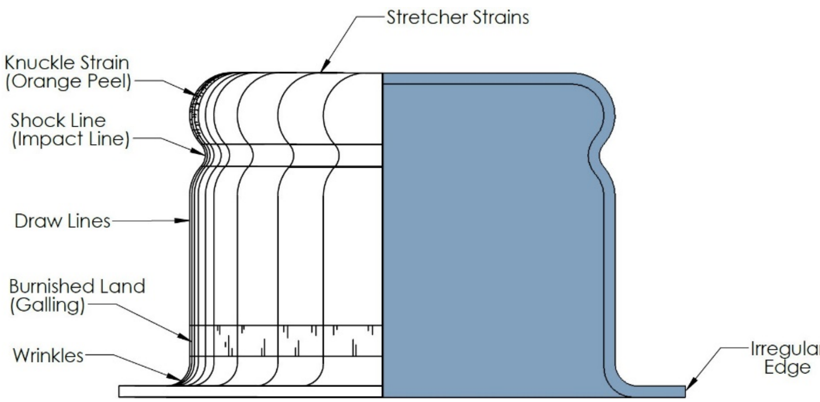
D1 = Inside diameter at a plane of intersection
D2 = Outside diameter at a plane of intersection
D3 = Flange diameter at a plane of intersection
H = Depth of Draw
L = Linear dimension
R1 = Punch radius
R2 = Die radius
R3 = Corner radius
S = 'Earing' due to directional property of material
T = Material thickness

23. *Edge Condition of Drawn Parts*

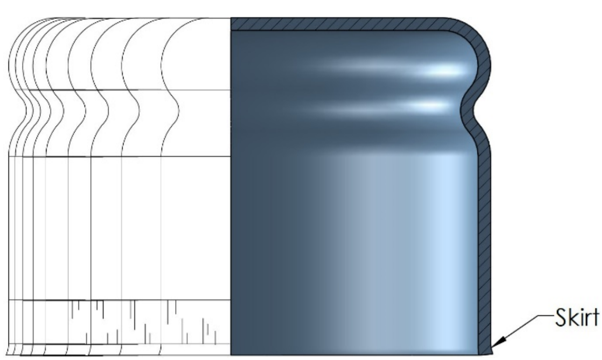


NOTES

24. *Surface and Characteristic Conditions of Drawn Shapes*



Flanged Part
(Straight Opening)



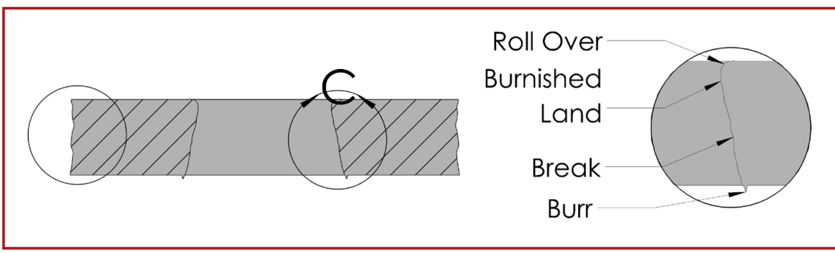
No Flange
Irregular Edge
Condition

25. *Limits for Burrs*

Burrs are ragged, usually sharp, protrusions on edges of metal stampings.

Common methods of specifying permissible burr (least expensive to most expensive):

1. Unless otherwise specified, an acceptable burr can be 10% of stock thickness.
2. A note "Conditioned for Handling" is interpreted to mean that normal stamping burrs are to be refined as necessary for average handling.
3. Remove burrs on specified edges.
4. Remove all burrs.
5. Break sharp edges or corners where specified.
6. Break all sharp edges and corners.



26. *Measurement and Limits of Flatness*

Perfect flatness is that condition which exists when all points on a surface lie in the same plane.

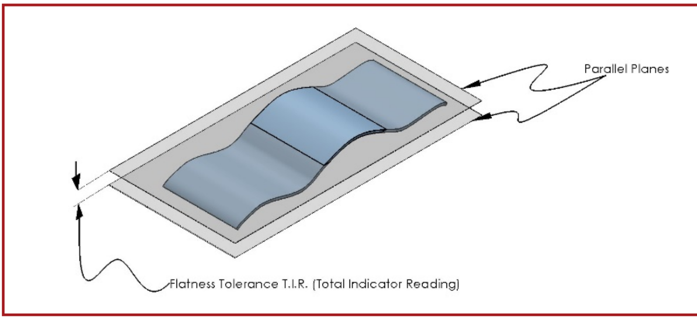
Flatness tolerance is the permissible distance between two parallel planes within which all irregularities of a given surface must lie.

Parts may be measured in either unrestrained or restrained conditions. (Recommended limits are applicable to parts measured in the unrestrained condition.)

Bottom surface irregularities are illustrated.

Commonly used flatness tolerances for metal stamping, laser and turret are shown below.

Surface Length	Flatness Tolerances
From 0" to 1"	.005 T.I.R.
Over 1" to 4"	.005/inch T.I.R.
Over 4"	.020+ .004/inch of additional length, T.I.R.



27. *Dimensioning Practices for Turret Press*

If there is a single area where the designer can accomplish the greatest benefit in reducibility and economy of manufacture, it is perhaps in communicating effectively with the supplier, and using appropriate detailing practices on drawings. Following are a few basic guidelines which can make an enormous difference.

First, select a meaningful datum in the body of the part—passing through the hole center, if possible—rather than using an edge or corner of the part. (See #28-Dimensioning Practices for Press Brake). There are several reasons for this suggestion.

It avoids problems of possible misalignment of the part, distortion from clamping, etc. It allows for more precise measurement by avoiding measurements from edges, which may be tapered and therefore dimensionally uncertain. It facilitates accurate inspection and it avoids unnecessary accumulation of tolerances.

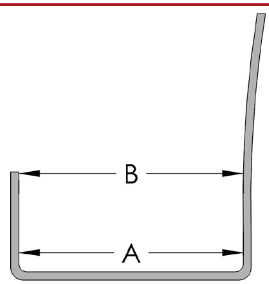
Second, on related hole patterns, dimensioning and tolerances should be within this pattern with only one dimension linking to the general datum. Better quality control and function of the product can be expected.

Third, highlight the truly significant dimensions. Critical dimensional relationships can be protected, if they are known.

28. *Dimensioning Practices for Press Brake*

Practical experience has proven that dimensioning and measuring practices must both be understood and agreed on by all parties to achieve a workable, mutual standard. Formed sheet metal parts present a unique problem in that angular tolerances as well as the flatness conditions interact with single plane dimensions because of the flexibility of sheet metal, especially the thinner gauges. To achieve consistent results when measuring formed parts, a standard has to be established on where and how dimensions are to be taken.

Form dimensions should be measured immediately adjacent to the bend radius in order not to include any angular and flatness discrepancy. See illustration below.



A-true bend dimension B-A + angular and flatness tolerance

Condition shown exaggerated for clarity

Feature-to-feature dimensions on formed legs of any length on flexible parts will be assumed to be measured in constrained condition, holding the part fixed to the prints' angularity specification. This standard is appropriate for the majority of thin sheet metal parts and results in a functional product. This is always true when the assembled part is, by design, held in constrained condition.

For the most economical production, dimension the part in a single direction wherever possible. Because of the sequential nature of the forming process, and the fact that dimensional variation is introduced at each bend, dimensioning in a single direction parallels the process and helps to control tolerance accumulation.

It is generally recommended that dimensioning be done from a feature to an edge. Feature-to-feature dimensions in two planes should be avoided. Feature-to bend dimensions may require special fixtures or gauging.

This also means that tolerances in the title block of a drawing may be unnecessarily restrictive for certain dimensions and angles, while very appropriate for others.

29. *Laser Cutting*

Current trends toward just-in-time (JIT) manufacturing, shorter part runs, and limited product life cycles have increased the use of laser cutting machines in production and prototype fabrication. Laser cutters are constantly evolving, as manufacturers find new and innovative ways to apply this basic technology.

Often the capabilities of lasers and turret presses can be combined. Turret presses are very fast and generate acceptable accuracy when punching many holes of the same or different diameters. Lasers are particularly accurate and economical for profiling irregular exterior contours.

These capabilities can be combined to produce accurate, complex parts at acceptable production rates by using each machine to perform that part of the cutting operation for which it is best qualified.

30. *Laser Considerations*

In addition to production economics, precision and edge condition, the knowledgeable designer considers these characteristics of laser produced parts when designing for lasers:

- **Localized Hardening.** Lasers cut by melting or vaporizing metal. This can create problems when cutting heat treatable materials as the area around the part will become case hardened.

  Laser cut holes in stainless steel or heat treatable steel alloys which require machining (tapping, countersinking or reaming) can be particularly troublesome. By the same token, designers can employ this characteristic to their benefit when a product must be case hardened for wear resistance.

- **Edge Taper.** The laser is most accurate where the coherent light beam enters the work piece. As the beam penetrates the part, the light scatters creating an edge taper condition similar but opposite from "breakout" in a sheared or pierced part. (The hole on the side of the work piece from which the laser beam exits is generally smaller in diameter than the entrance side).

  Thus the designer must carefully consider the final use of the part and, in some cases, may have to specify from which side the part should be cut.

• **Minimum Through-Feature Size.** The cutting laser beam is focused down to approximately 0.010 in. (0.2 mm) and is therefore capable of cutting holes and features with radii approximating 0.030 in. (0.76 mm). The limits applicable to piercing or blanking with a punch and die, such as the relationship between minimum hole size and material thickness, or the minimum distance between features to avoid distortion, do not apply when laser cutting.

• **Micro-tab.** Holds part in the sheet while processing, generally removed after processing. Can be strategically located so removal not necessary, resulting in cost savings. General size of micro-tab would be .25mm-.5mm.

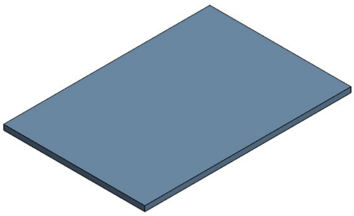
Since no mechanical force is applied, the width of material remaining between cutout features may be very narrow without distortion occurring during metal removal. A typical application would be tight spaced venting slots on a visually important surface.

Note: Minimum Features equal to material thickness.

It should also be recognized that the laser, like any other CNC servo driven machine, accumulates mechanical, thermal and electro-mechanical tolerances during the production cycle. For economy and quality, critical dimensions should be highlighted and functional dimensions should be detailed in accordance with their function.

31. *Carbon Steel Strip*

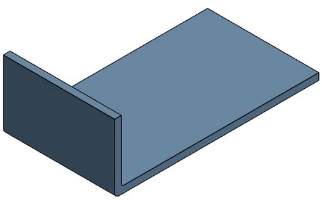
Cold Rolled Tempers



No. 1 —

Hard

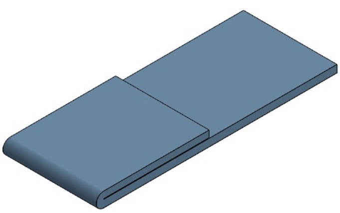
For stamping or punching flat pieces requiring rigidity and strength.



No. 2 —

Half Hard

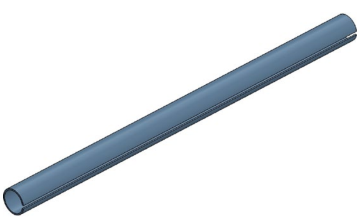
For simple blanking operations. Will bend at sharp right angle across the grain (direction of rolling).



No. 3 —

Quarter Hard

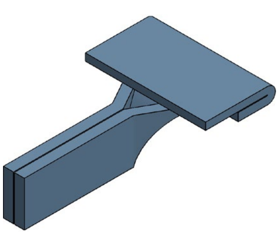
Will bend flat on itself across the grain. Takes some bending with the grain.



No. 4 —

Pinch Pass or Skin Rolled

For tubing, molding, some deep drawing. Will bend both directions of the grain.



No. 5 —

Dead Soft

For deep drawing and difficult forming. Extremely soft, it will bend flat on itself both directions of the grain.

32. *Minimum Bend Radii for Aluminum*

Minimum permissible radius varies with the nature of forming the operation, type of forming equipment, and the design and condition of tools. Minimum working radius for given material or hardest alloy and temper for a given radius can be ascertained only by actual trial under contemplated conditions of fabrication.

Radii Required for 90° bend in terms of thickness (T) approximate thickness						
TYPE	.016	.032	.064	.125	.187	.250
1100-0, 3003-0 5052-0	0	0	0	0	½T	1T
1100-H14 6061-0	0	0	0	1T	1T	1½T
3003-H14 5052-H32 , 2024-0	0	0	0	1T	1T	1½T
5052-H34	0	0	1T	2T	2T	3T
6061-T4	0	0	1T	1½T	2½T	3T
6061-T6	1T	1T	1½T	2½T	3T	3½T
2024-T3	2½T	3T	4T	5T	5T	6T

[1]The radii listed are the minimum recommended for bending sheets without fracturing.
[2]Alclad sheet in the heat-treatable alloys can be bent over slightly smaller radii than the corresponding tempers of the bare alloy.
[3]Heat-treatable alloys can be formed over appreciably smaller radii immediately after solution heat treatment.

NOTES

33. *Tap Drill Sizes Machine Screw Threads*

TAP NO.	THDS PER INCH	TAP DRILL SIZE		O.D. OF SCREW	TAP NO.	THDS PER INCH	TAP DRILL SIZE		O.D. OF SCREW
		NO.	DEC.				NO.	DEC.	
0	80	56	.046	.0578	1/4	20	7	.201	.250
1	64	54	.055	.0710	1/4	28	3	.213	.250
1	72	54	.055	.0710	5/16	18	F	.257	.3125
2	56	51	.067	.0842	5/16	24	I	.272	.3125
2	64	50	.070	.0842	3/8	16	5/16	.312	.375
3	48	48	.076	.0973	3/8	24	Q	.332	.375
3	56	47	.078	.0973	7/16	14	U	.368	.4375
4	36	45	.082	.1105	7/16	20	W	.386	.4375
4	40	44	.086	.1105	1/2	13	27/64	.421	.500
5	40	39	.099	.1236	1/2	20	29/64	.453	.500
6	32	36	.106	.1368					
6	36*	35	.110	.1368					
6	40	34	.111	.1368					
7	32*	31	.120	.1500					
8	32	30	.128	.1631					
8	36	29	.136	.1631					
9	32*	25	.149	.1763					
10	24	25	.149	.1894					
10	32	21	.159	.1894					
12	24	17	.173	.2158					
12	28	15	.180	.2158					

Formula to find Tap Drill Size on Odd Sizes.

$$\text{Tap Drill Size} = (\text{O.D. of screw}) - \frac{.97425}{\text{No. Threads Per Inch}}$$

Example - 1/4 - 20

$$\frac{.97425}{20} = .049 \quad .250 - .049 = .201$$

The above tap drill sizes will produce a 70-75% thread.
*Non-standard thread.

NOTES

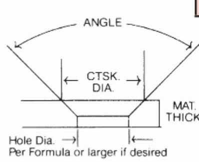
34. *Pierced Hole Size Ctsk. Diameter for 82° Screws*

SCREW SIZE	0	1	2	3	4	5	6	8	10	12	1/4	5/16	3/8
CTSK. DIAM	.119	.146	.172	.199	.225	.252	.279	.332	.385	.438	.507	.636	.762
MATERIAL THICKNESS	SIZE HOLE TO BE PIERCED ± .002												
.015	.098	.125	.151	.178	.204	.231	.258	.311	.364	.417	.486	.615	.741
.018	.093	.120	.146	.173	.199	.226	.253	.306	.359	.412	.481	.610	.736
.020	.089	.116	.142	.169	.195	.222	.249	.302	.355	.408	.477	.606	.732
.025	.080	.107	.133	.160	.186	.213	.240	.293	.346	.399	.468	.597	.723
.031	.070	.097	.123	.150	.176	.203	.230	.283	.336	.389	.458	.587	.713
.036	.061	.088	.114	.141	.167	.194	.221	.274	.327	.380	.449	.578	.704
.042		.078	.104	.131	.157	.184	.211	.264	.317	.370	.439	.568	.694
.050			.090	.117	.143	.170	.197	.250	.303	.356	.425	.554	.680
.062					.122	.149	.176	.229	.282	.335	.404	.533	.659
.072						.132	.159	.212	.265	.318	.387	.516	.642
.083							.140	.193	.246	.299	.368	.497	.623
.094								.173	.226	.279	.348	.477	.603
.109									.200	.253	.322	.451	.577
.125										.226	.295	.424	.550
.140											.269	.398	.524
.156												.370	.496
.187												.316	.442

H - Pierced Hole Size
C - CTSK. Diam. T. Thickness of Mat.

Sizes shown are minimum sizes to produce a satisfactory CTSK.

60° Formula - H = C-1.155 (T-.005)
70° Formula - H = C-1.620 (T-.005)
80° Formula - H = C-1.739 (T-.005)
90° Formula - H = C-2 (T-.005)
100° Formula - H = C-2.383 (T-.005)



NOTES

35. *Tonnage Formula*

Pressure required in tons to cut a blank or contour using a flat faced punch with no shear.

Formula

T = P x Th x C

T = Pressure required in tons

P = Perimeter of blank in inches

Th = Thickness of material

C = Constant (see common ones below)

Constant = The material shear strength/2000 pounds

Example

.050 CR Steel, 1/2 Hd; Cutting edge of 12 linear inches

T = 12 x .050 x 32

T = 20 tons required

Constants

Aluminum — Soft-11
T4/T6-15

Steel — HR/Cold Rolled-27
1/2 Hard-32

Stainless Steel — Annealed-37
1/2 Hard-50

4130 AQ — 40

Brass— 1/2 Hard-22
Hard-25

Dayton Rogers

est. 1929 MANUFACTURING CO.

Building on Experience . . . Unequaled Craftsmanship . . .
Stamping & Forming an Impact Now and into the Future . . .

Corporate Offices

8401 West 35W Service Drive
Minneapolis, Minnesota 55449

With locations in Minnesota, Ohio, South Carolina and Texas

(800) 677-8881

www.daytonrogers.com