APPLIED MATERIALS®

Sheet Metal Handbook

BASIC

APPLIED MATERIALS®

CONTENTS

DESIGN

DRAFTING

MANUFACTURING

REFERENCE

DESIGN TIPS

Design

MATERIAL

Following are the preferred materials and sizes to be used for sheet metal parts

Material		Thickness Sizes					
Aluminum 5052-H32	Decimal	.063	.090	.125	.190		
CRS	Gage #	18	16	14	12	11	10
	Decimal	.048	.060	.075	.105	.120	.135
HRS	Gage #	7					
	Decimal	.179					
SST 304	Gage #	18	16	14	12	11	10
	Decimal	.048	.060	.075	.105	.120	.135

Aluminum 6061(in any temper) **IS NOT** to be used for sheet metal parts (enclosures, panels, brackets etc)

Material CALLOUT on drawings:

Aluminum 5052-H32, thickness to 3 decimal places
CRS, thickness to 3 decimal places (Gage #)

HRS, thickness to 3 decimal places (Gage #)

SST 304, thickness to 3 decimal places (Gage #)

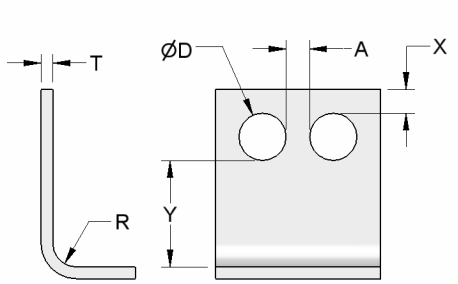
No other description like 'Sheet' or length and width dimensions should be specified.

For the above material callouts the supplier will provide following materials:

Material Callout	Material provided by supplier	Conforming to Specification
Aluminum 5052-H32	Aluminum 5052-H32	ASTM B 209
CRS	Cold Rolled Steel, Commercial Quality	ASTM A 1008
HRS	Hot Rolled Steel, Pickled and Oiled	ASTM A 1011
SST 304	Stainless Steel 304, #2B finish	ASTM A 240

Refer to the Reference section for complete list of sheet sizes, tolerances on thicknesses and material properties.

DESIGN CONSIDERATIONS: PUNCHING



HOLES

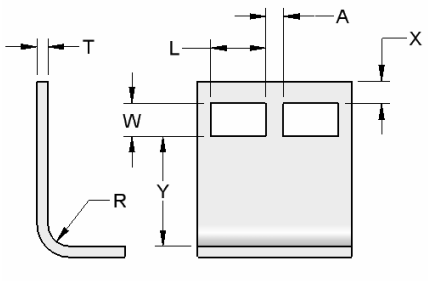
ØD =1.2 T MIN. (Steel & Aluminum)
ØD =2T MIN. (SST)
For ØD < 5T
X = 1.5T (FROM EDGE)
Y = 2.5T+R (FROM INSIDE BEND)
A = 1.5T (BETWEEN EDGES)
For ØD =5T TO 10T
X = 2T (FROM EDGE)
Y = 2.5T+R (FROM INSIDE BEND)
A = 2T (BETWEEN EDGES)
For ØD > 10T
X = 4T (FROM EDGE)
Y = 4T+R (FROM INSIDE BEND)
A = 4T (BETWEEN EDGES)

HARDWARE: Holes for hardware such as self-clinching nuts are to be located per the manufacturers recommendations, without violating the above minimum edge distances.

HOLE SIZES: Up to 1 inch diameter use preferred drill sizes. The available sizes are tabulated in Reference section.

Holes smaller than 1.2 T are possible using LASER cutting or machining in a secondary operation.

Material thickness range	Minimum hole diameter or slot width achievable
<.075	.010
.075 - .090	.015
.090 - .125	.020
.125 - .156	.025
.156 -.187	.030

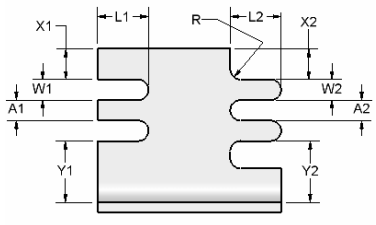


SQUARES & RECTANGLES

W =1.2 T MIN. (Steel & Aluminum)
W =2T MIN. (SST)
For L < 10T
X = 2T (FROM EDGE)
Y = 2.5T+R (FROM INSIDE BEND)
A = 2T (BETWEEN EDGES)
For L > 10T
X = 4T (FROM EDGE)
Y = 4T+R (FROM INSIDE BEND)
A = 4T (BETWEEN EDGES)

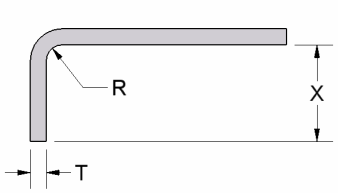
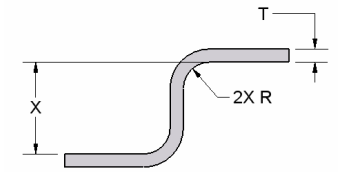
SLOTS

SAME AS SQUARES AND RECTANGLES

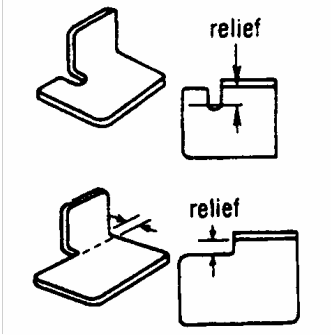
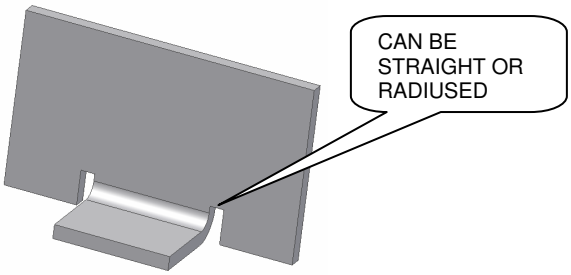


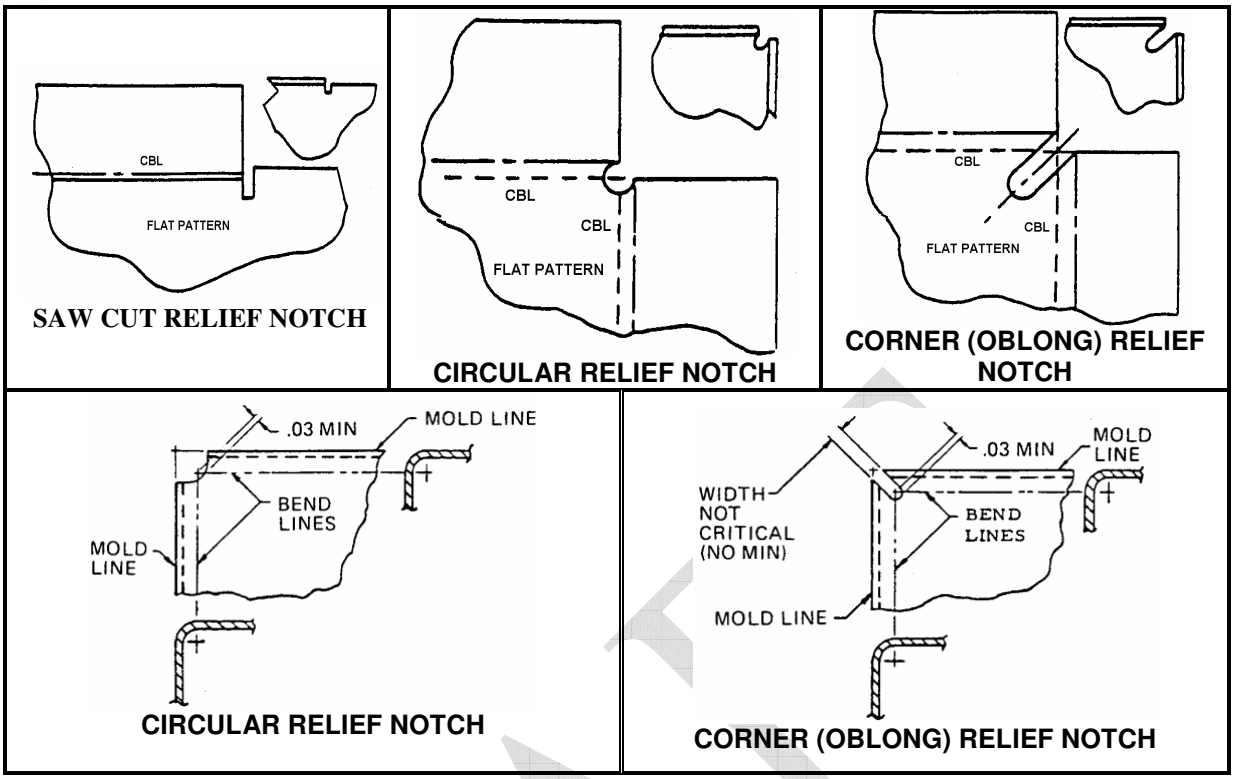
NOTCHES & TABS

NOTCHES:
W1 = 1.5T
L1 < = 5W1
A1 = 4T
TABS:
W2 = 1.5T
L2 < = 5W1
A2 = 4T
R > = .5T

DESIGN CONSIDERATIONS: BENDING	
 **MINIMUM BEND HEIGHT/ MINIMUM FLANGE WIDTH**	**FOR FLANGE LENGTH < 12** **X = 2.5T+R** **FOR FLANGE LENGTH = 12 TO 36** **X= 4T+R** **FOR FLANGE LENGTH > 36 CONTACT SUPPLIER.** **IF SPOT WELDING ON THE FLANGE** **X= .62 MIN (TO ACCOMMODATE TOOL)**
 **MINIMUM BEND STEP**	**X = 2T+2R**

BEND RADII
UOS Bend Radii are minimum and depend on the material and thickness. REFER to Manufacturing section for minimum bends radii values for various materials. When other than minimum bend radius is needed choose from the following values: **.063, .094, .125, .188, .250, .375, .500**
MINIMUM BEND RADII ARE NOT TO BE DIMENSIONED.

BEND RELIEFS	
Whenever sheet metal bends intersect one another, it is necessary to remove material from the intersection area to prevent interference and is known as bend relief. There are many possible configurations for bend reliefs and few of them are illustrated below. Supplier is to provide minimum sizes. Refer to manufacturing section for some values.	
BEND RELIEFS ARE NOT TO BE DIMENSIONED.	
 **RELIEF NOTCHES FOR L BENDS**	 NOTCH WIDTH = 2T (MIN .060) NOTCH DEPTH = T + BEND RADIUS



TOLERANCES

Following table shows economically attainable tolerances for various situations. With title block tolerances a (.XX = .02 & .XXX = .010), it may not be necessary to tolerance the dimensions, as the two and three decimal place dimensioning will meet the tolerances in the table.

TOLERANCES (Economically Attainable)					
Condition	**Plus**	**Minus**	**Condition**	**Plus**	**Minus**
Hole to Hole	.010	.010	Edge to Edge	.02	.02
Hole to Edge	.010	.010			
Edge to Fold (Bend)	.010	.010	Title Block **.XX**	.02	.02
Hole to Fold (Bend)	.02	.02	Title Block **.XXX**	.010	.010
Fold (Bend) to Fold (Bend)	.02	.02	Title Block Angular	1°	1°
FEATURE SIZE TOLERANCES					
Holes: SINGLE HIT, Up to 1" diameter (See Reference section for standard punch sizes up to 1" diameter)	.003	.003	Holes: NON-SINGLE HIT, NIBBLED (Some sizes are available for single hit, when tolerance is =/- .003	.010	.010
All other features (Multiple hits)	.010	.010			
THICKNESS OF COPY (XEROX) PAPER IS .005					

GEOMETRIC TOLERANCES
RULE #1
Where only a tolerance of size is specified, the limits of size of an individual feature prescribe the extent to which variations in its geometric form, as well as size, are allowed.
Explanation: When **NO geometric tolerance** is specified, the **dimensional tolerance CONTROLS** the **geometric form** as well as the size.

1. Rule #1 applies to all features-of-size on a drawing. It is an **INVISIBLE** control that applies to all features-of-size UNLESS it is OVERRIDEN by a geometric tolerance.
2. Rule #1 applies to individual features-of-size only.
3. Geometrical tolerances are to REFINE (smaller than) the size tolerance.

Exceptions:

1. Rule #1 is not applicable to **STOCK material** (thicknesses)
2. A feature of size on a non-rigid part.

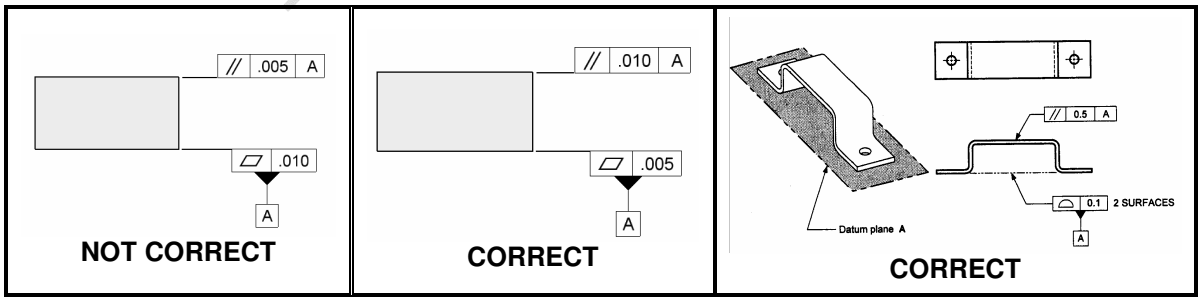
Due to the nature of the sheet metal applications some of the geometrical tolerances are not normally applicable. Following table shows the applicability.

APPLICABLE		NOT APPLICABLE	
FORM TOLERANCES			
	FLATNESS		CIRCULARITY
	STRAIGHTNESS		CYLINDRICITY
ORIENTATION TOLERANCES		RUNOUT TOLERANCES	
	PARALLELISM		CIRCULAR RUNOUT
	PERPENDICULARITY		TOTAL RUNOUT
	ANGULARITY		
LOCATION TOLERANCES			
	POSITION		CONCENTRICITY
			SYMMETRY
PROFILE TOLERANCES			
	SURFACE PROFILE		LINE PROFILE

When more than one geometric tolerance is specified for a single feature or feature of size, the relationship of tolerances shall be as follows:

Form Tolerance < Attitude/Orientation Tolerance < Location Tolerance

The form of the datum feature should be more precise than measurements taken from the datum.
Example:



FINISHES

Following are the finishes and applicable specifications used for sheet metal parts. Refer to Drafting section for callout instructions

Aluminum:

Anodizing: MIL-C-8625, Type II, Class 1 (Clear), or Class 2 (Black or Red for Tooling parts)
Chemical Coating (ALODINE): MIL_C_5541

Aluminum, CRS and HRS:

Painting: Applied Materials specification 0250-01021.

CRS and HRS:

Zinc plating: ASTM B 633, Type ????????????

SST:

Passivation: ASTM A 967.

All Materials:

Cleaning: Applied Materials specification 0250-70351
Applied Materials specification 0250-09720.

Drafting

APPLIED MATERIALS®

DRAWING NOTES

Following requirements ARE NOT TO BE INCLUDED in the general notes, for sheet metal parts, as they are COVERED in the MANUFACTURING STANDARD:

1. INSTALL PER MANUFACTURERS RECOMMENDATIONS.
2. DEBURR
3. ALL RADII AND BEND RELIEFS TO BE MINIMUM.
4. ANY CLEANLINESS REQUIREMENTS (Example: HANDLE WITH LATEX GLOVES)
5. MASK THREADS FOR PAINTING

Following NOTE 1 is to REPLACE current "APPLICABLE STANDARDS/SPECIFICATIONS:" on all sheet metal part drawings:

1. **APPLICABLE STANDARDS/SPECIFICATIONS:**
   APPLIED MATERIALS 0250-XXXXX, SHEET METAL MANUFACTURING SPECIFICATION.

Following Applied Materials specifications are **NOT applicable** to sheet metal parts:
1. Applied Materials 0250-20001 (UHV machined part fabrication)
2. Applied Materials 0250-20000 (UHV part cleaning)

TITLE BLOCK TOLERANCES	
.XX	.02
.XXX	.010
ANGULAR	1º
SURFACE FINISH	——

SURFACE FINISH: NO SURFACE FINISH VALUE IS TO BE SPECIFIED.
ALL DIMENSIONS APPLY BEFORE PAINTING, PLATING AND OTHER FINISHING OPERATIONS.

HARDWARE

- **DO NOT include the delta note: INSTALL PER MANUFACTURERS RECOMMENDATIONS. This note is covered in the manufacturing standards.**

Self-clinching fasteners are to be called out per the following format

[FASTENER TYPE], [MODIFIER], SELF-CLINCHING, [SIZE], MATERIAL

Fastener type: NUT, STANDOFF, STUD, PIN
Modifiers (when applicable): Blind, Flush, Floating, Miniature, Threaded, Thru-hole, Self-locking
Material: For CRS and Aluminum parts: SST (SST 303)
For SST parts: SST PH (Precipitation Hardening SST) in available types else SST (SST 303).

Penn Engineering & Manufacturing Corp. ("PEM") Fastener Numbers are to be used as DEFAULT with OR EQUIVALENT added.

		NUT, SELF-CLINCHING, .190-32, SST PH "PEM" SP-032-2 OR EQUIV.	
		NUT, SELF-CLINCHING, .164-32, SST "PEM" CLS-832-2 OR EQUIV.	
		STANDOFF, SELF-CLINCHING, .190-32 X .500 LG, SST "PEM" SOS-032-16 OR EQUIV.	
ITEM #	QTY	DESCRIPTION	

- **AVOID CONCEALED HEAD FASTENERS. CONTACT THE SUPPLIERS FOR ALTERNATIVES**.

HARDWARE ITEMS ARE TO BE CALLED OUT WITH A BALLOON ITEM NUMBER AND INDICATED NEAR SIDE OR FAR SIDE.
NEAR SIDE IS DEFAULT AND NOT BE CALLED OUT, MEANING **INSTALLED FROM** NEAR SIDE.
FAR SIDE TO BE CALLED OUT, MEANING **INSTALLED FROM** FAR SIDE.
The term **INSTALLED FROM** is to be **OMITTED.**

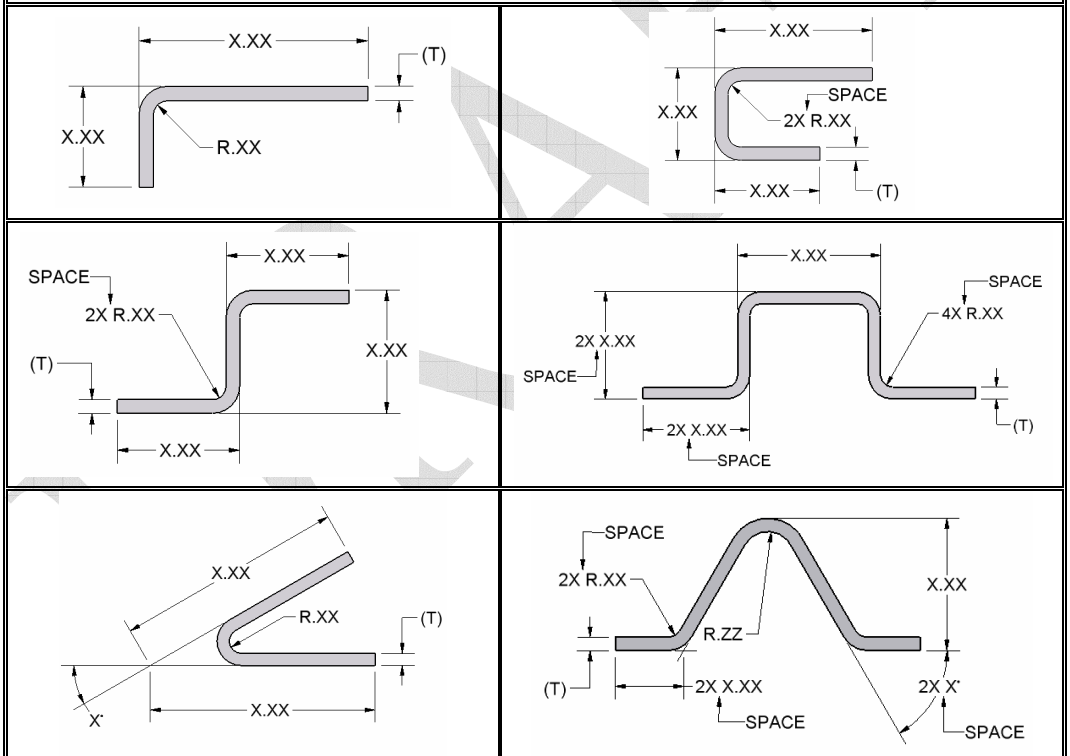
DIMENSIONING: GENERAL

On an engineering drawing every feature has to be fully defined. There are four characteristics that need to be defined on all part features:

Size: Must be given
Form: Within size tolerance if not given (Rule #1)
Orientation (Attitude): Within position if not given
Location (Position): Must be given

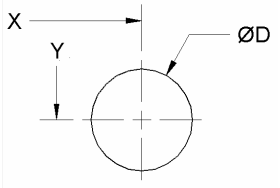
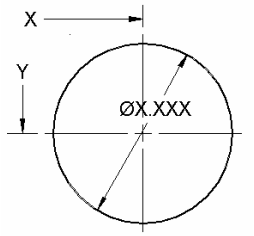
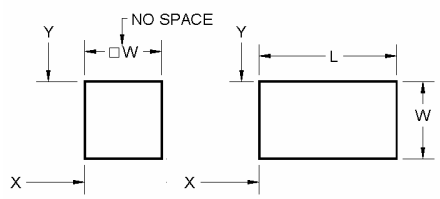
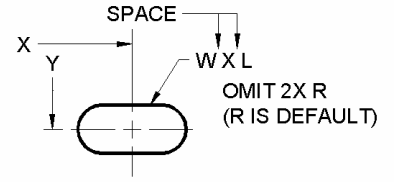
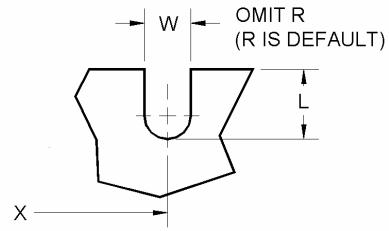
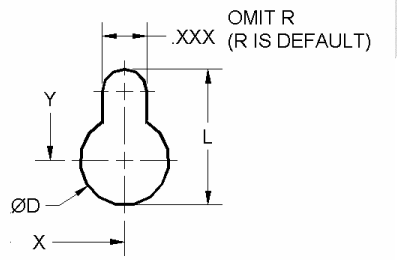
If any one aspect in the above is not defined, the part definition is incomplete.

DIMENSIONING BENDS



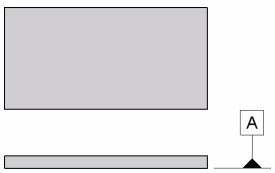
R in the above diagrams is to be specified **ONLY** when a specific radius is needed. Otherwise, the DEFAULT is MINIMUM BEND RADIUSModeling should be done using these values.

MINIMUM BEND RADII ARE NOT TO BE DIMENSIONED.
BEND RELIEFS ARE NOT TO BE DIMENSIONED.

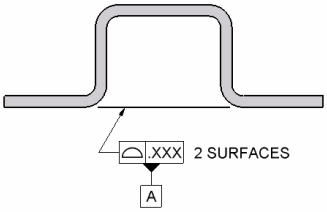
DIMENSIONING FEATURES	
	HOLES < 1.5 DIAMETER: CALL OUT SIZE WITH LEADER. CALL OUT LOCATION TO CENTER LINES. **SIZE: 3 DECIMALS** DO NOT TOLERANCE HOLE DIAMETER.
	HOLES > 1.5 DIAMETER: CALL OUT SIZE AS SHOWN. CALL OUT LOCATION TO CENTER LINES. **SIZE: 3 DECIMALS** DO NOT TOLERANCE HOLE DIAMETERS.
	CALL OUT SIZE AS SHOWN. CALL OUT LOCATION TO EDGES. **SIZE: 3 DECIMALS** DO NOT TOLERANCE SIZE. NOTE: **NO SPACE BETWEEN ϖ SYMBOL AND SIZE**
	CALL OUT SIZE **WITH LEADER** AS SHOWN. CALL OUT LOCATION TO CENTER LINES. **SIZE: 3 DECIMALS** DO NOT TOLERANCE SIZE. NOTE: **OMIT 2X R**. UOS IT IS CONSIDERED R FULL.
	CALL OUT SIZE AS SHOWN. CALL OUT LOCATION AS SHOWN. **SIZE: 3 DECIMALS** DO NOT TOLERANCE SIZE. NOTE: **OMIT R**. UOS IT IS CONSIDERED R FULL.
	CALL OUT SIZE AS SHOWN. CALL OUT LOCATION AS SHOWN **SIZE: 3 DECIMALS** DO NOT TOLERANCE SIZE. NOTE: **OMIT R**. UOS IT IS CONSIDERED R FULL.
IRREGULAR SHAPES (EXAMPLE: SINGLE & DOUBLE D HOLES, D SUB HOLES)	(SIZE: D= 3 DECIMALS DO NOT TOLERANCE SIZE.) **PREFERRED:** USAGE OF **PROFILE TOLERANCE** TO CONTROL SIZE, TOLERANCE AND LOCATION.

COMMONLY USED DATUMS FOR SHEET METAL PARTS

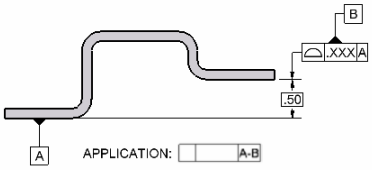
Note the placement of datum symbol in all cases shown below



DATUM: PLANAR SURFACE

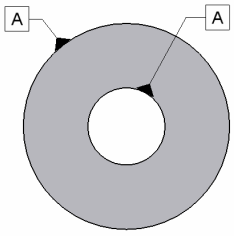


DATUM: COPLANAR SURFACES

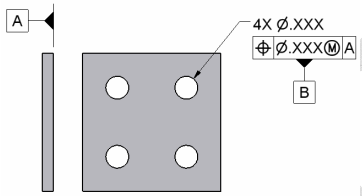


NOTE: BASIC DIMENSION AND COPLANARITY BETWEEN DATUMS A & B

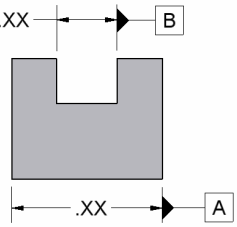
DATUM: OFFSET COPLANAR SURFACES



DATUM: DIAMETER INTERNAL OR EXTERNAL



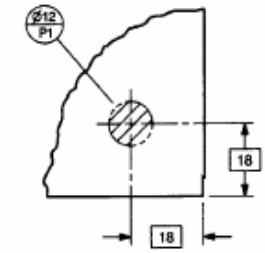
DATUM: PATTERN OF HOLES



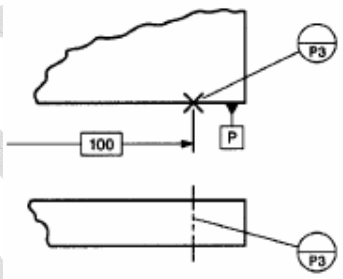
DATUM: WIDTH (INTERNAL OR EXTERNAL)

DATUM TARGETS

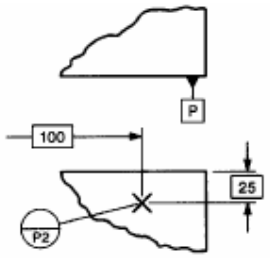
Because of inherent irregularities, the entire surface of some features cannot be effectively used to establish a datum. Datum targets designate specific areas, lines, or points of contact on a part that are used in establishing a datum reference frame.



DATUM TARGET AREA

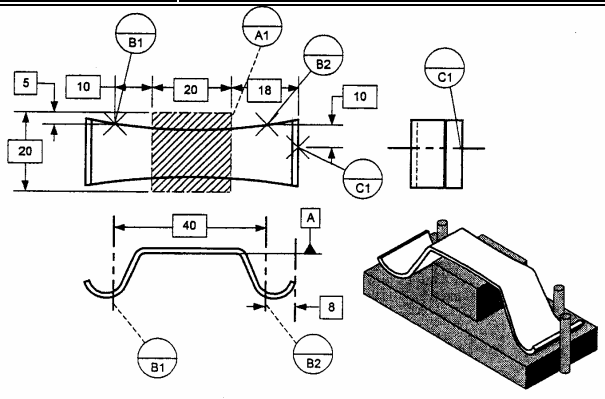


DATUM TARGET LINE



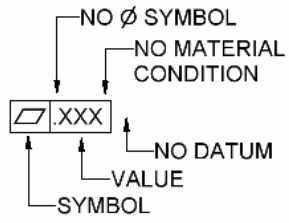
DATUM TARGET POINT

When specifying datum targets on a non-rigid part, the 3-2-1 rule does not always apply. In cases where a non-rigid part is very flexible, it may need more than 3 targets on the primary datum feature, or additional targets on the secondary and tertiary datum features to support the part

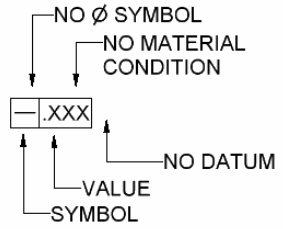


DATUM TARGET APPLICATION

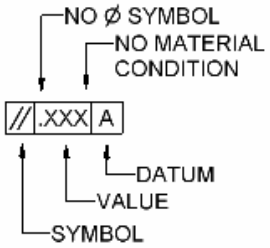
SPECIFYING COMMONLY USED GEOMETRIC TOLERANCES



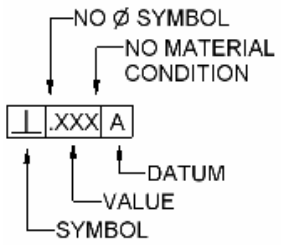
FLATNESS



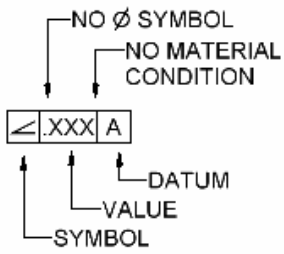
STRAIGHTNESS: SURFACE ONLY



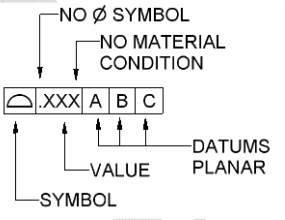
PARALLELISM: PLANAR SURFAE ONLY



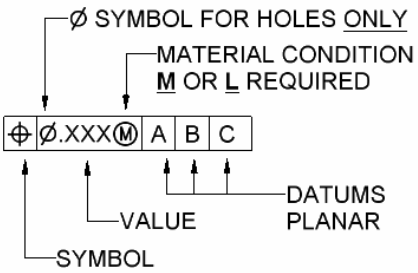
PERPENDICULARITY: PLANAR SURFACE ONLY



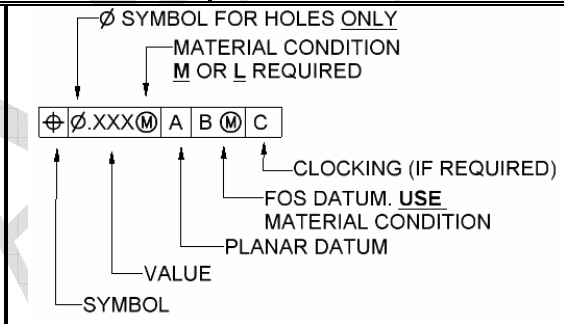
ANGULARITY: PLANAR SURFACE ONLY



PROFILE

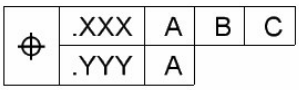


POSITION: PLANAR DATUMS



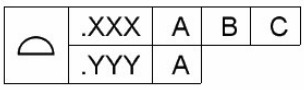
DATUMS: PLANAR & FOS

COMPOSITE TOLERANCING



.YYY IS A REFINEMENT OF .XXX.
(USUALLY HALF OR LESS OF .XXX)

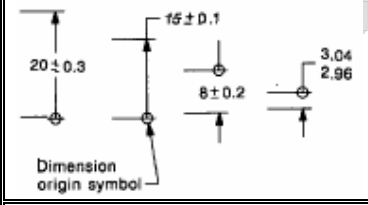
COMPOSITE POSITIONAL TOLERANCING



.YYY IS A REFINEMENT OF .XXX.
(USUALLY HALF OR LESS OF .XXX)

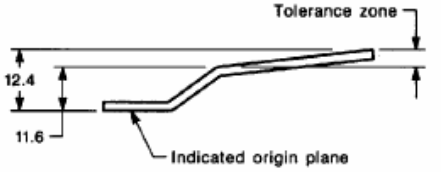
COMPOSITE PROFILE TOLERANCING

DIMENSION ORIGIN SYMBOL IS VERY USEFUL FOR SHEET METAL PARTS

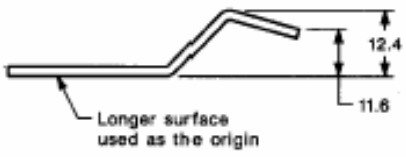


SYMBOL

AS SHOWN ON DRAWING



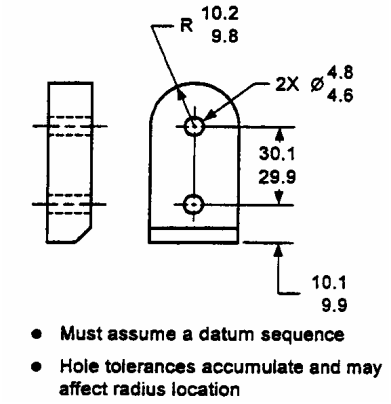
MEANS THIS



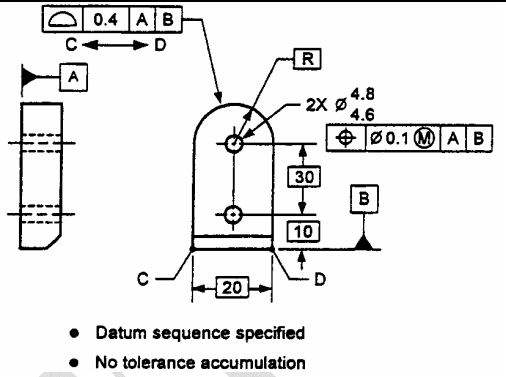
NOT THIS

Profile is a versatile tolerance and is especially of great use in tolerancing irregular shapes. Following are examples showing the usage of profile tolerances.

COORDINATE AND EQUIVALENT PROFILE TOLERANCING COMPARISION

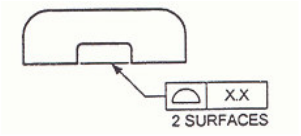


COORDINATE TOLERANCING

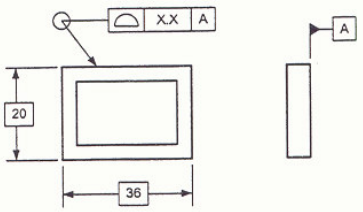


PROFILE TOLERANCING

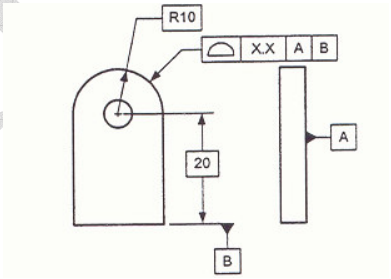
COMMON PROFILE APPLICATIONS



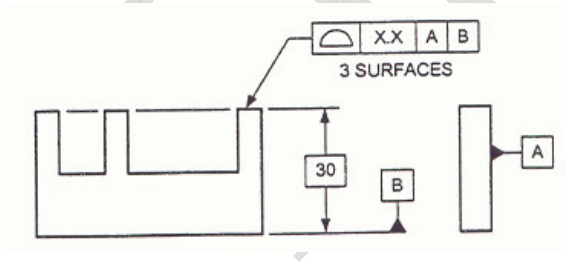
Profile used to control coplanarity and form of multiple surfaces



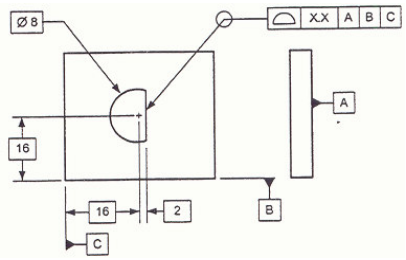
Profile used to control size and orientation of an outline of a part feature.



Profile used to relate a part feature to a datum reference frame

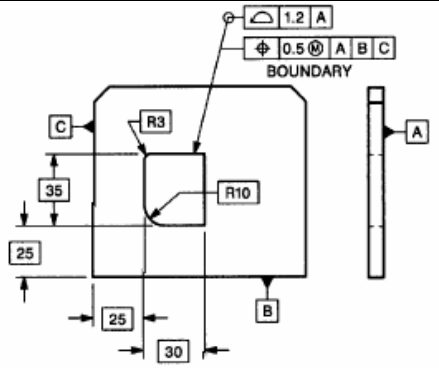


Profile used to control location, orientation, and /or of a part feature or a collection of features

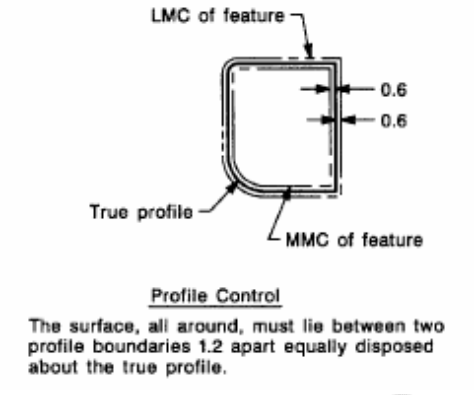


Profile used to control size, and/or location, and/or orientation, and/or form of an irregular-shaped opening or irregular external shape

PROFILE & POSITION TO DEFINE AN IRREGULAR SHAPE

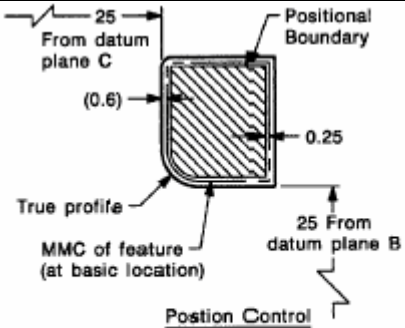


Profile tolerance combined with positional tolerance is used to fully define an irregular feature. The profile tolerance is interpreted as if no positional tolerance were shown.



Profile Control

The surface, all around, must lie between two profile boundaries 1.2 apart equally disposed about the true profile.



Postion Control

No portion of the surface may be permitted to lie within the boundary of MMC contour minus the positional tolerance when positioned with respect to datum planes A, B and C.

Manufacturing

Drawing Interpretation

Interpret drawings per the following specifications as applicable.

1. ASME Y14.5M: Dimensioning and Tolerancing)
2. ASME Y14.6: Screw Thread Representation)
3. ASME Y14.13M: Mechanical Spring Representation)
4. ASME Y14.36: Surface Texture Symbols
5. ASME Y14.38: Abbreviations and Acronyms
6. AWS A2.4: Standard Symbols for Welding, Brazing, and Nondestructive Examination
7. Applied Materials 0251-05096: Inch / Millimeter Conversion Procedure

General Workmanship

Supplier will take the necessary steps and precautions during the manufacturing sequence to provide clean handling of the parts without visible scratches, fingerprints and other such cosmetic defects.

Edge Finishing

Supplier is to provide parts with functional features free of burrs, that is, absence of overhanging material projections that are greater than .003 inch in height
All edge and corner (internal and external) condition that may result in the damage (e.g., cutting) of handling or mating surfaces should be smoothed over by material removal, rounding or chamfering. The extent of material removal should not be greater than .005 X .005 or R.005.
All “micro tie” burrs are to be removed to provide a smooth surface.
Supplier has the latitude for choice of the appropriate processes for deburring and smoothing edges and corners at the appropriate stages of the manufacturing sequence.

Material:

Material to be supplied based on drawing call out is as follows:

Drawing Material Callout	Material to be provided by supplier	Conforming to Specification
Aluminum 5052-H32	Aluminum 5052-H32	ASTM B 209
CRS	Cold Rolled Steel, Commercial Quality	ASTM A 1008
HRS	Hot Rolled Steel, Pickled and Oiled	ASTM A 1011
SST 304	Stainless Steel 304, #2B finish	ASTM A 240

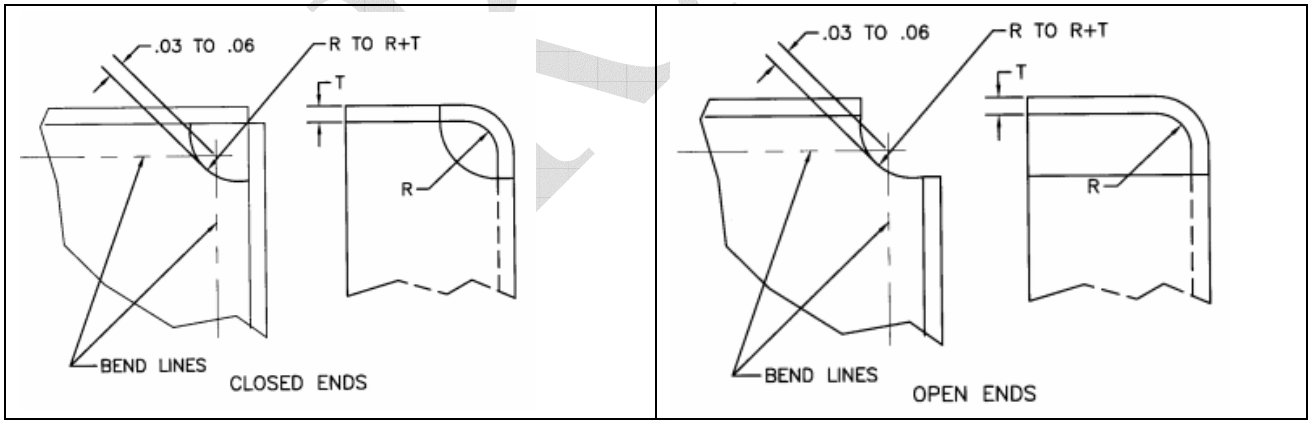
BEND RADII

Unless specified on the drawing, all bends are to be of minimum radius and not to exceed the values shown in the following table.

MINIMUM BEND RADII NOT TO EXCEED							
Aluminum 5052-H32	**T Decimal**	.063	.090	.125	.190		
	R	**.032**	**.063**	**.125**	**.190**		
SST 304	**T Gage #**	18	16	14	12	11	10
	T Decimal	.048	.060	.075	.105	.120	.135
	R	**.032**	**.032**	**.032**	**.063**	**.063**	**.063**
CRS	**T Gage #**	18	16	14	12	11	10
	T Decimal	.048	.060	.075	.105	.120	.135
	R	**.032**	**.032**	**.032**	**.063**	**.063**	**.063**
HRS	**T Gage #**	7					
	T Decimal	.179					
	R	**.125**					
T= Material Thickness **R= Bend Radius**							

BEND RELIEFS

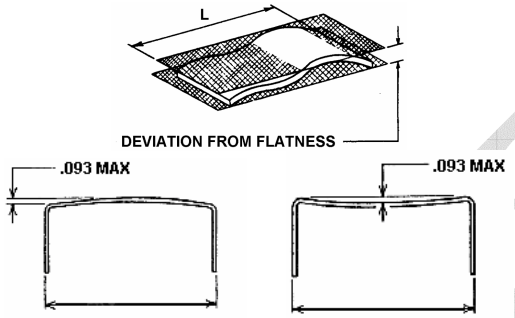
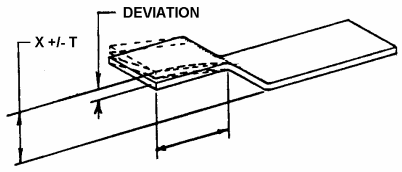
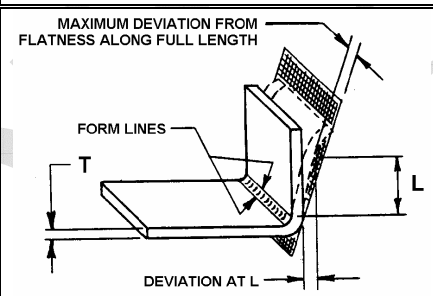
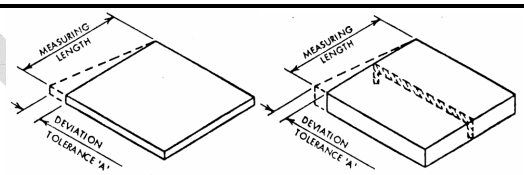
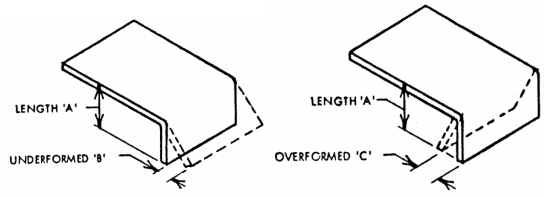
Reliefs are not dimensioned on drawings. Supplier should provide the necessary reliefs to the minimum possible sizes. Type of relief is at suppliers option.



HARDWARE

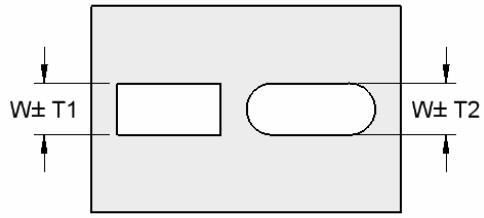
All commercial hardware such a self-clinching fasteners, self-adhering gaskets, tapes etc are to be **INSTALLED PER THE MANUFACTURERS RECOMMENDATIONS.**

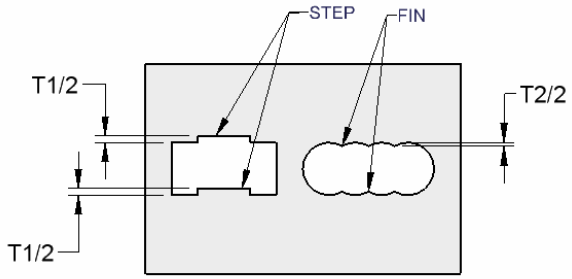
Self-clinching fasteners are called out FAR SIDE or **no side** for NEAR SIDE.
FAR SIDE means **INSTALL FROM FAR SIDE.** No side callout means **INSTALL FROM NEAR SIDE.**

TOLERANCES		
TITLE BLOCK TOLERANCES DO NOT APPLY FOR SINGLE HIT HOLES AND OTHER FEATURES WITH STANDARD PUNCHES. SUPPLIER TO CONTROL THESE HOLES AND OTHER SINGLE HIT FEATURES TO TOLERANCE OF +/-.003. **NIBBLED HOLES AND OTHER MULTIPLE HIT FEATURES ARE TO BE CONTROLLED TO +/- .010**		
UNSPECIFIED GEOMETRIC TOLERANCES		
FLATNESS		**L UPTO 1": .005** **L=1" TO 6: .005/INCH** **L>6: 030+.003 (L– 6)** **MAXIMUM: .093** **Example: FOR 12" LENGTH DEVIATION IS .030+.003 (12-6) = .048**
STRAIGHTNESS: BLANKED EDGES		**.004/INCH MAX < = SIZE TOL (2T)**
PARALLELISM		**.010/INCH OF SURFACE (L). MAX < = SIZE TOL (2T)**
PERPENDICULARITY: RIGHT ANGLE BENDS		**FOR L UPTO 1": .005** **LENGTH L OVER 1" IS CONTROLLED BY FLATNESS TOLERANCE**
PERPENDICULARITY: BLANKED OR FORMED RIGHT ANGLE EDGES		**BLANKED OR FORMED EDGES: .007/INCH** **MAX < = SIZE TOL (2T)**
ANGLE OF BENDS FOR FLANGES		**UNDERFORMED (B):** **UPTO .50 = .01** **.50 UP = .01 + .01 (A - .50)** **OVERFORMED (C):** **UPTO .50 = .02** **.50 UP = .02 + .02 (A - .50)**

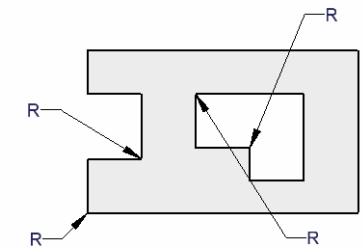
CUTOUTS

Steps and Fins will be acceptable provided the projections do not exceed the tolerance for the feature in that cross section.





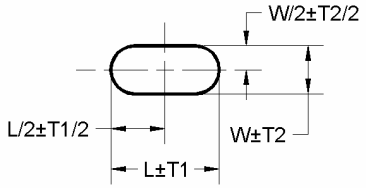
CORNERS

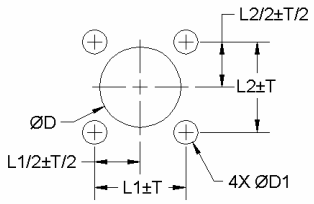


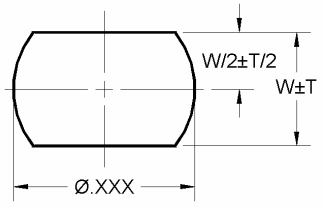
Corners shown sharp (no radius or chamfer and no dimension noted), may have a radius or chamfer .03maximum

SYMMETRICAL FEATURES: TOLERANCE ALLOCATION

Symmetrical features dimensioned around the centerline shall be centered by one half of the overall dimension and tolerance

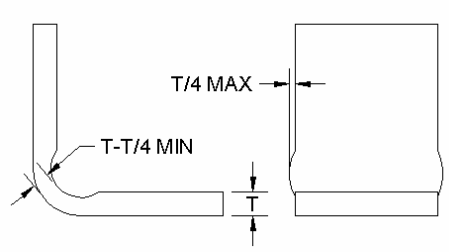






BULGING (FLARE OR DEPRESSION)

A flare or a depression of one-quarter of the material thickness is permitted at bends. This may be noticeable more with thicker material. However, this is NORMALLY ACCEPTABLE from a functional point of view. (If absolutely unacceptable, a note should be given on the drawing).



Reference

APPLIED MATERIALS®

NOTE: Aluminum is specified by thickness and there are no gage numbers.

Stainless Steel

Gage #	Decimal Equiv.	Thickness inches
8	.165	.1601-.1874
9	.156	.1451-.1600
10	.135	.1301-.1450
11	.120	.1141-.1300
12	.105	.0981-.1140
13	.090	.0831-.0980
14	.075	.0721-.0830
15	.070	.0651-.0720
16	.060	.0581-.0650
17	.056	.0521-.0580
18	.048	.0461-.0520
19	.042	.0401-.0460
20	.036	.0351-.0400
21	.034	.0321-.0350
22	.029	.0291-.0320
23	.028	.0261-.0290
24	.024	.0231-.0260
25	.022	.0198-.0230
26	.018	.0177-.0197
27	.017	.0160-.0177
28	.015	.0145-.0160

Aluminum 5052 & 6061 Sheet

Specified Thickness	Sheet Width Thru 39	Sheet Width 39-59	Sheet Width 59-78
	Tolerance +/- (Plus/Minus)		
.025	.0015	.0020	.0030
.032	.0020	.0025	.0035
.040	.0025	.0035	.0045
.050	.0030	.0035	.0050
.063	.0030	.0035	.0050
.080	.0035	.0045	.0060
.090	.0035	.0045	.0060
.100	.0045	.0055	.0070
.125	.0045	.0055	.0070
.160	.0070	.0090	.0110
.190	.0070	.0090	.0110

Steel Sheet

Gage #	Decimal Equiv.	Thickness inches	
		H.R. P.& O.	C.R.
4	.2242	.2152-.2332	
5	.2092	.2002-.2182	
6	.1943	.1853-.2033	
7	.1793	.1713-.1873	.1703-.1883
8	.1644	.1564-.1724	.1554-.1734
9	.1495	.1415-.1575	.1405-.1585
10	.1345	.1265-.1425	.1285-.1405
11	.1196	.1116-.1276	.1136-.1256
12	.1046	.0966-.1126	.0966-.1106
13	.0897	.0827-.0967	.0847-.0947
14	.0747	.0677-.0817	.0697-.0797
15	.0673	.0613-.0733	.0623-.0733
16	.0598	.0538-.0658	.0548-.0648
17	.0538	.0478-.0598	.0498-.0548
18	.0478	.0428-.0528	.0438-.0518
19	.0418		.0378-.0458
20	.0359		.0329-.0389
21	.0329		.0299-.0359
22	.0299		.0269-.0329
23	.0269		.0239-.0299
24	.0239		.0209-.0269
25	.0209		.0179-.0239
26	.0179		.0159-.0199
27	.0164		.0144-.0184
28	.0149		.0129-.0169
29	.0135		.0115-.0155
30	.0120		.0110-.0130

Material	Units	5052-H32	SST 304	HRS-HRPO	CRS-CQ
Applicable Specification		ASTM B 209	ASTM A 240	ASTM A 1011	ASTM A 1008
Properties					
Density	lbs/cubic inch	.097	.292	.284	.284
Ultimate Tensile Strength	KSI	33	85	48	50
Yield Strength	KSI	28	35	25	25
Modulus of Elasticity	KSI X 10E3	10	29	29	29
Hardness Max.	HRB	62	85	65	65
Carbon Content				.10 max	.10 max

Estimating the minimum bend radius

Metal can only be bent to a certain angle before cracks form at the bend and, ultimately, the work piece breaks. To predict the smallest achievable radius that a sheet can be bent to, the following equation can be used:

$$R_{min} = T \times (50/r - 1)$$

- T = thickness
- Rmin = minimum bend radius
- R = reduction in area in atensile test for a given material (%)

PUNCH SIZES (BASED ON PREFERRED DRILL SIZES)					
Decimal	Std. Drill	Decimal	Std. Drill	Decimal	Std. Drill
.063	1/16	.250	1/4 (E)	.756	49/64
.067	51	.257	F	.781	25/32
.070	50	.261	G	.797	51/64
.073	49	.266	17/64	.813	13/16
.076	48	.272	I	.828	53/64
.078	5/64	.277	J	.844	27/32
.081	46	.281	9/32	.859	55/64
.086	44	.297	19/64	.875	7/8
.089	43	.302	N	.891	57/64
.094	3/32	.313	5/16	.906	29/32
.096	41	.316	O	.922	59/64
.100	39	.323	P	.938	15/16
.104	37	.328	21/64	.953	61/64
.107	36	.332	Q	.969	31/32
.109	7/64	.339	R	.984	63/64
.113	33	.344	11/32	1.000	1.00
.116	32	.348	S	1.006	
.120	31	.359	23/64	1.086	
.125	1/8	.368	U	1.094	
.129	30	.375	3/8	1.120	
.136	29	.386	W	1.125	
.141	9/64	.391	25/64	1.250	
.144	27	.397	X	1.418	
.147	26	.406	13/32	1.430	
.152	24	.422	27/64	1.500	
.156	5/32	.438	7/16	1.575	
.161	20	.453	29/64	1.750	
.166	19	.469	15/32	1.875	
.170	18	.484	31/64	1.900	
.172	11/64	.500	1/2	2.000	
.173	17	.516	33/64	2.250	
.177	16	.531	17/32	2.281	
.180	15	.547	35/64	2.300	
.185	13	.563	9/16	2.362	
.188	3/16	.578	37/64	2.500	
.191	11	.594	19/32	2.550	
.194	10	.609	39/64	2.750	
.196	9	.625	5/8	2.875	
.199	8	.641	41/64	3.000	
.203	13/64	.656	21/32	3.500	
.209	4	.672	43/64	3.940	
.213	3	.688	11/16	4.000	
.219	7/32	.703	45/64	4.375	
.228	1	.719	23/32	4.500	
.234	15/64	.734	47/64	4.580	
.246	D	.750	3/4		

Note: Punch sizes greater than 1" in diameter, in the table, are commonly available standard punches and produce holes in single hit.

ANODIZING

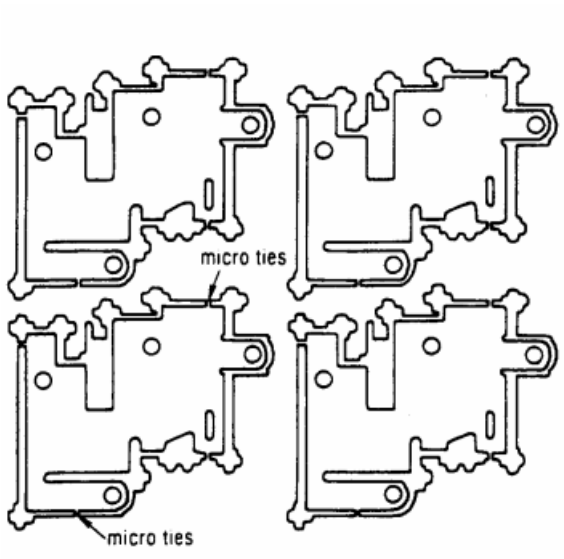
Anodizing is an electrolytic process in which a metal is made the anode in a suitable electrolyte so that when an electric current is passed through the electrolyte, the metal surface is converted to a form of its oxide that has useful decorative, protective, or other desirable properties. Anodizing is an anodic process as opposed to cathodic electroplating processes. Sealing (several methods) of the anodized surface is an important consideration.

MIL-A-8625 is the widely used specification for general purpose anodizing. Following table gives the available types. There are numerous specialty processes utilizing impregnation of Teflon etc. to meet specific demands.

Type No.	Description	Finish Properties
I	Chromic acid	Not to be used for Applied Materials' applications.
II	Sulfuric acid	Widely used form of anodizing. Coating typically .0002 to .001 inch thick. Accepts a wide range of permanent color dyes in their **true colors**. Offers moderate corrosion and abrasion resistance, better than chromate chemical conversion systems (MIL-C-5541, "Alodine" etc)
III	Hard Anodizing	Coating is thicker (from .001 to .004 inch), harder and more dense than Type II. Used for abrasion resistance primarily and corrosion resistance. Natural color of hard anodized aluminum is medium- to dark-gray or olive drab green. Accepts dark permanent color dyes but with gray to gray-green tinge (from the thick anodic coating).

Applicable Specifications:
General Purpose: MIL-A-8625
Types:
Type I & IB: Chromic Acid Anodizing. NOT TO BE SPECIFIED.
Type II: Sulfuric Acid Anodizing
Type III: Hard Anodizing

Class 1: Non-dyed, clear.
Class 2: Dyed. Color to be specified.

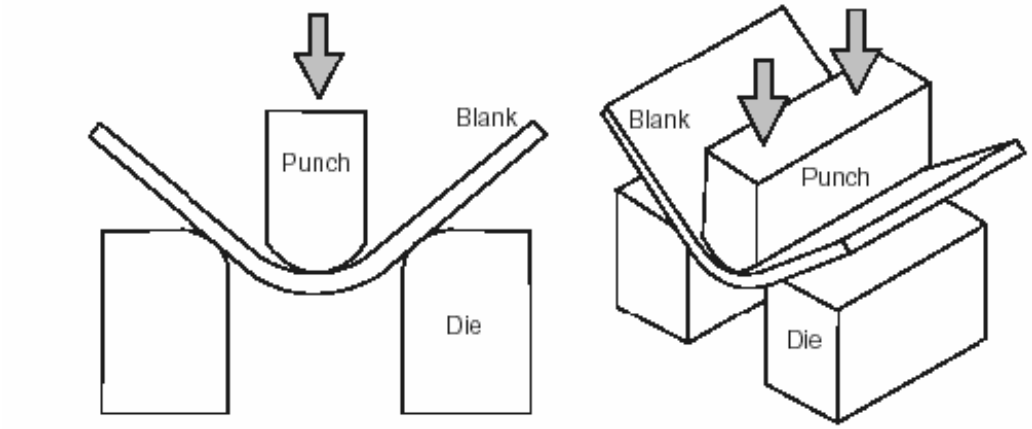


Turret presses, also known as CNC punch presses are particularly suited for small lot production. This is also known as “soft tooled” manufacturing. The CNC press, due to the extreme variety of punch combinations available, eliminates the necessity and lead time for special tooling. It is thus , highly flexible to produce small production quantities.
To optimize processing time and material utilization, it is common for parts to be ganged or nested during turret press production. Several parts in the workpiece sheet are held together during punching by tiny webs or bridges known as “micro ties” which are allowed to remain after punching.
After punching, the sheet is agitated break apart the individual parts. The parts are therefore known as “shaker parts” or “shake aparts”. There will be a tiny burr on the perimeter of the part where the micro tie is broken. Supplier will be removing the burr.

A process called “Air Bending” does most of the bending operations.

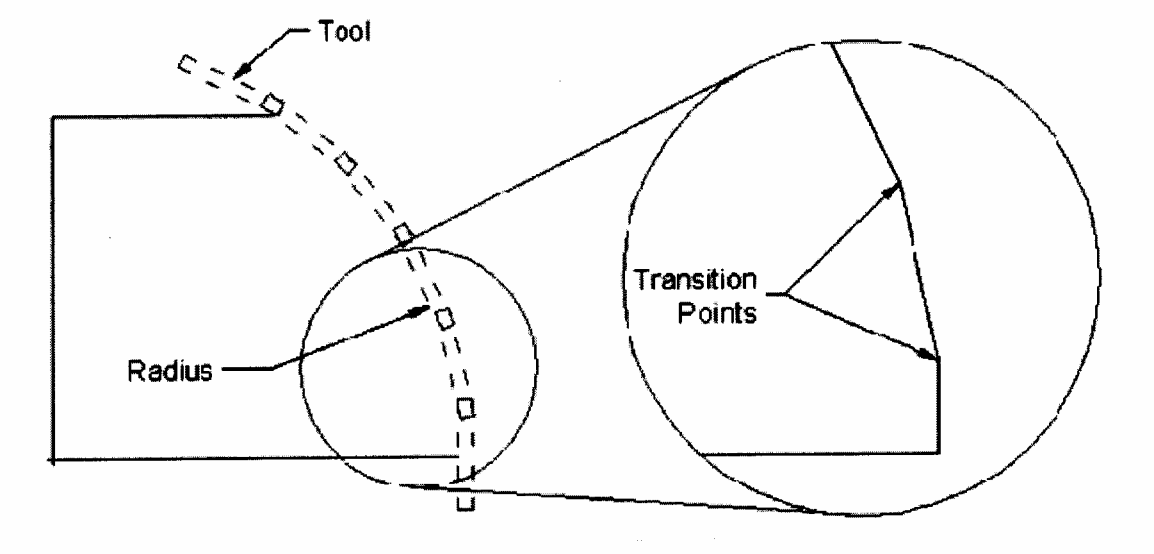
Air Bending is a bending process in which the punch touches the work piece and the work piece does not bottom in the lower cavity. As the punch is released, the work piece springs back a little and ends up with less bend than that on the punch (greater included angle). This is called spring-back.
The amount of spring back depends on the material, thickness, grain and temper. The spring back will usually range from 5 to 10 degrees. The same angle is usually used in both the punch and the die to minimize set-up time. The inner radius of the bend is the same as the radius on the punch.

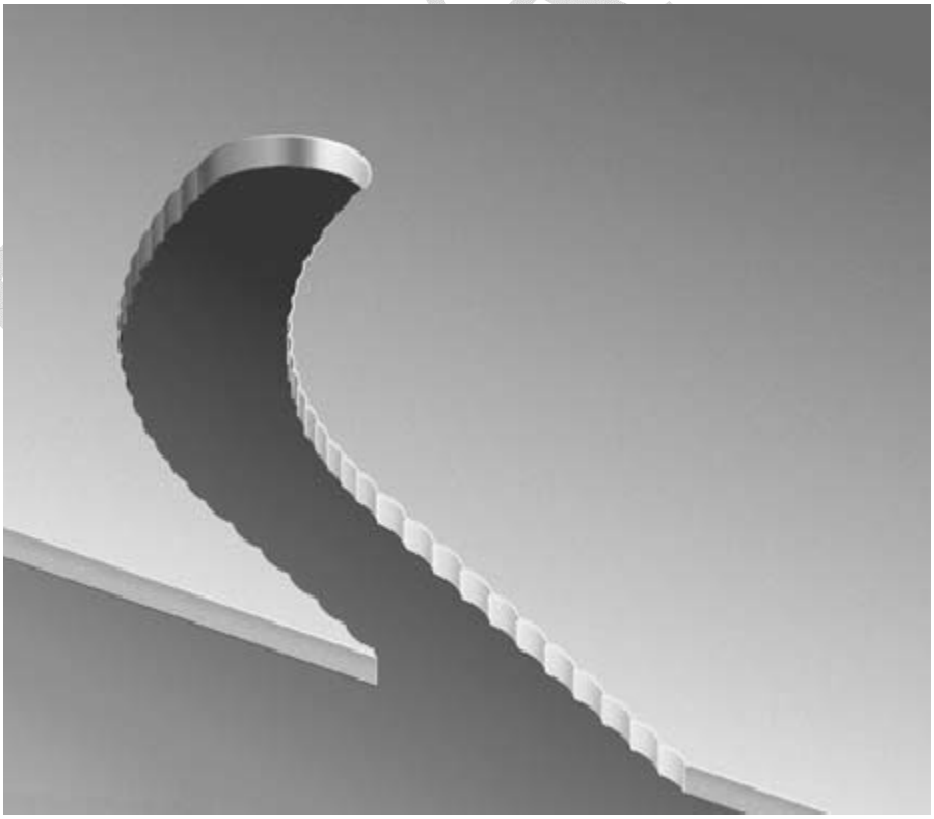
In air bending, there is no need to change any equipment or dies to obtain different bending angles because the bend angles are determined by the punch stroke. The forces required to form the parts are relatively small, but accurate control of the punch stroke is necessary to obtain the desired bend angle.



NIBBLING LARGE RADII AND DIAMETERS

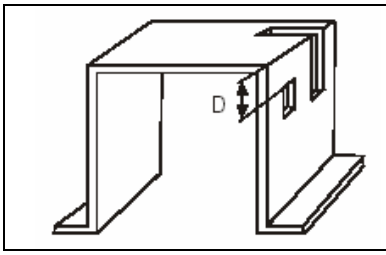
When a large radius or diameter is to be punched, the radius or diameter is made by making several hits with a flat tool. The angle and position of the tool changes as it moves around the edge simulating a large radius or diameter. The edge will have a scalloped look, tiny points formed where one flat is transitioned to another. The scalloped feature is pictured below. After deburring it is hardly perceivable, but still there.



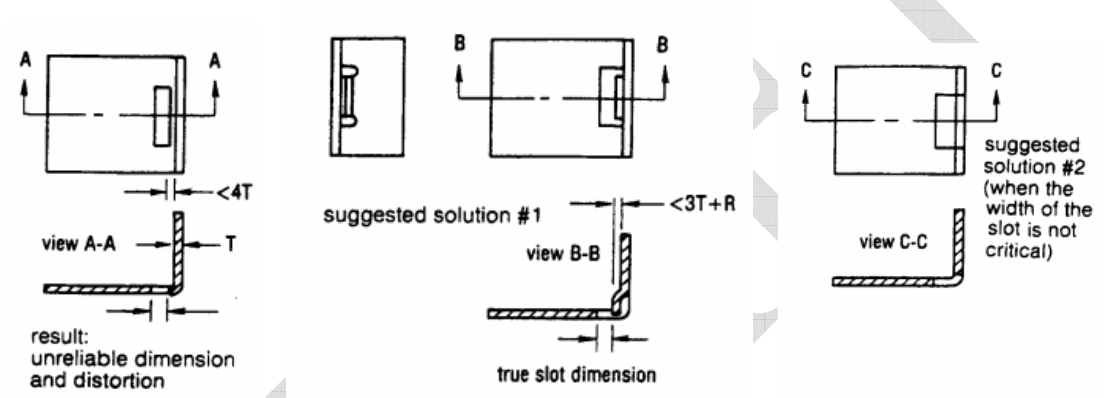


Design Tips

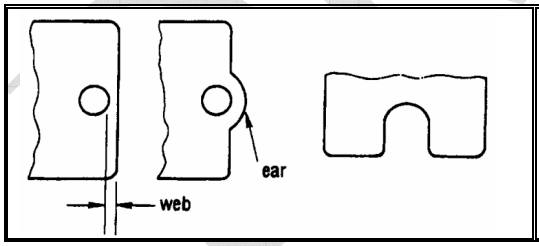
As a rule of thumb, plan on using an aluminum sheet about 40% thicker than steel. Since aluminum weighs only 1/3 as much as steel, this means that the equivalent aluminum sheet will weigh only half as much as the steel sheet it replaces. The designated aluminum thickness will give you about the same stiffness.



Slots and holes too close to bend can cause distortion to these holes. Holes or slots should be located (**D**) a minimum of 3.5 stock thicknesses plus the bend radius. If it is necessary to have the holes closer, then the hole or slot should be extended beyond the bend line.



Design alternative to following recommended slot-to-form spacing is punching out the entire area.



Webs should be at least 2 times the stock thickness (T) to avoid distortion or bulging. Alternate designs include a notch or, a pierced hole with an ear.