DELTA SHOP PRACTICE STANDARDS

CONTENTS

PREFACE

These shop practice standards apply to all Delta products. The contents are intended to provide a basis for acceptance or rejection of parts manufactured by or for Delta from proprietary drawings.

Date of the latest issue of each standard is shown directly below the title block on the lead page of each subject. Additional pages for this booklet will be published as new shop practice standards are issued. Likewise changes to existing standards will result in reissue of compatible pages for this booklet.

For interpretation of these standards, contact Delta's engineering department.

1. Scope

This standard provides a guide to the effectiveness and definition of tolerances, both those specified directly on the drawing and those implied through applicability of Delta Shop Practice Standards or other documentation.

2. Purpose

To provide a basis for acceptance or rejection of parts manufactured by or for Delta from proprietary drawings.

3. Order Of Tolerance Precedence

Tolerances expressed by one or more of the following means take precedence in this order:

a) Tolerances applied directly to a dimension or feature override all other forms of expression.

b) General or local notes may qualify or override portions of implied or general title block tolerances.

c) Title block tolerance entries.

d) Shop Practice tolerances where applicable.

Note: Exceptions to b) c) and d) are:

- raw materials (bar, plate, wire, tubing, etc.)
- hardware items (bolts, screws, washers, etc.)
- dimensions in the material designation
- nominal thread sizes
  - Commercial or National standard tolerances such as ASTM, SAE, AISI, ISO, etc., apply in these cases.

Where a combination of tolerance controls affect the same feature, the more restrictive value shall take precedence.

Acceptance limits established in Shop Practice Standards apply to every pertinent condition not specifically toleranced on a drawing or a supporting document.

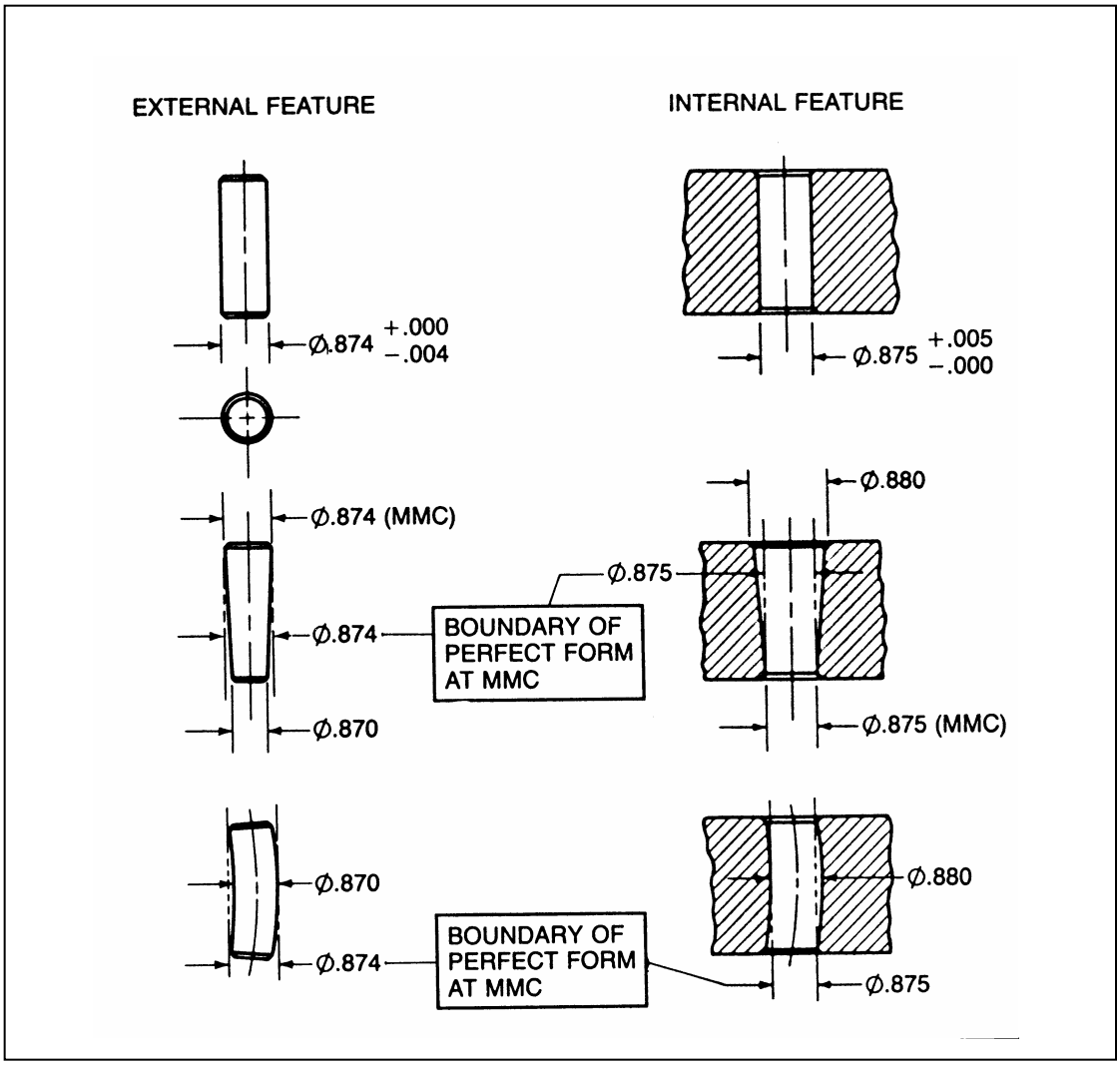
4. Form Control Related To Size Tolerance

Toleranced dimensions for the size of individual features control form as well as size in the following ways:

a) If produced at the maximum material condition (MMC), the actual feature must be at the perfect form implied by the drawing.

b) If produced away from maximum material condition (within size tolerance at any cross section) a deviation in form equal to the amount of such departure is allowable.

This interpretation permits the following extreme variations. Maximum material condition is indicated by phantom lines.



5. Revisions to the Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue			2003-10-24

1. Coverage

This standard establishes tolerances for untoleranced linear features.

2. Tolerances For Linear Dimensions

Dimensions (inches)		
Over	**Thru**	**Tolerance(inches)**
0	6	± 1/64
6	24	± 1/32
24	48	± 3/64
48	72	± 1/16
72	96	± 5/64
96	∞	± 1/8

Note:

1. Dimensions in brackets (x.xxx) are for reference only.

3. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-14

1. Coverage

This standard establishes the extent of positional and alignment control implied by the pictured relationship of features.

2. Machined Surfaces On A Common Plane

When two or more surfaces are shown on a common plane and located by common dimensions they must all lie between the same two parallel planes which are separated by a distance equal to either the location tolerance or by the alignment tolerance given in table 1.

Figure 1 - Multiple surfaces on a single plane

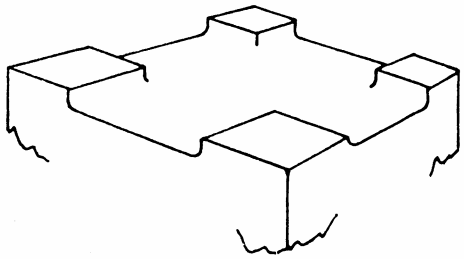
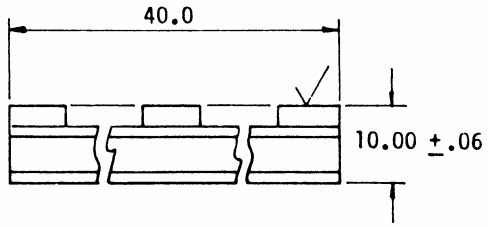


Table 1
Alignment Tolerance- Common Plane

Overall length of machined surface		**- all dimensions are given in inches -**
Over	**Thru**	**Alignment Tolerances**
0	3	.006
3	5	.010
5	10	.015
10	∞	.015 plus .001 for each 2 in. over 10 in.

Figure 2. Alignment of surfaces shown on common plane

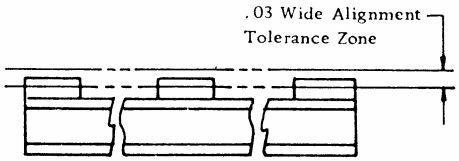
This on the Drawing:



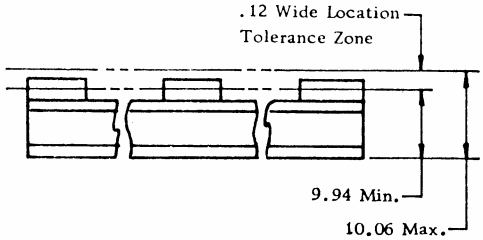
Means This:

Alignment Tolerance Calculation:

For 1st 10" length	.015
For Add'l. 30" length	.015
Total Align. Tol.	.030



And This:



In effect this creates a zone within a zone, since the .03 inch wide implied alignment tolerance zone may fall anywhere within the .12 inch wide location tolerance zone.

If each surface were dimensioned individually, an alignment tolerance between surfaces would not apply.

If the locational tolerance zone were less than the .03 inch wide alignment zone, the lesser value would pertain.

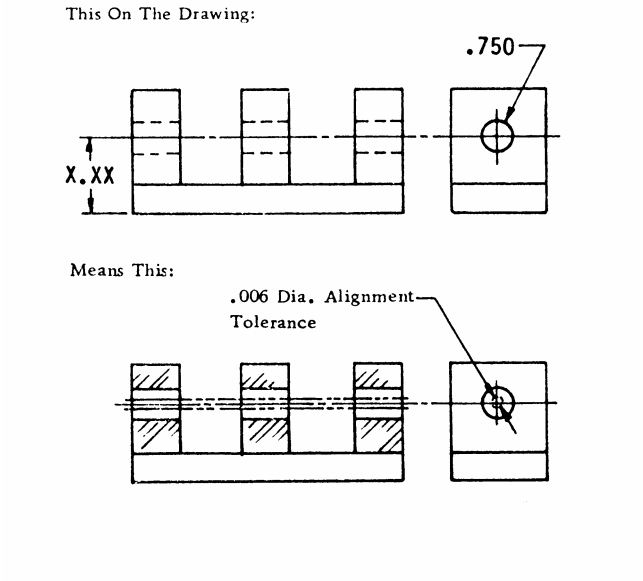
3. Holes With A Common Axis

When two or more round holes are shown on a common axis, the axis of each hole must lie within the same cylindrical tolerance zone. Length of the alignment tolerance zone extends to the extreme limits of the outer holes and diameter of alignment tolerance is given in table 2.

Table 2
Alignment Tolerance - Common Axis

Specified Hole Size		
Over	**Thru**	**Diameter of Alignment Tolerance Zone**
0	.250	Equal to Size Tolerance
.250	1.500	50% of Size Tolerance
1.500	∞	25% of Size Tolerance

Figure 3. Alignment of holes shown with a common axis



Calculation:
Size tolerance for the .750 diameter holes is -.002, +.009 (per Standard 532), or .011 total. 50% of .011 rounds off to the .006 tolerance shown above.

3.1 Gauging Methods

The alignment of multiple holes shall be considered acceptable if a gauge made according to figure 4 or figure 5 will pass through all of the holes simultaneously.

Figure 4 - Gauging of same diameter holes with a common axis

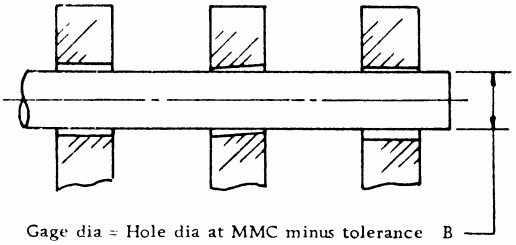


Figure 5 - Gauging of different diameter holes on common axis

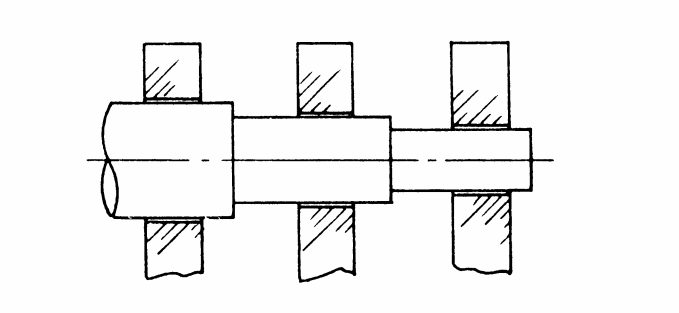


Figure 5a - Gauging of concentric holes

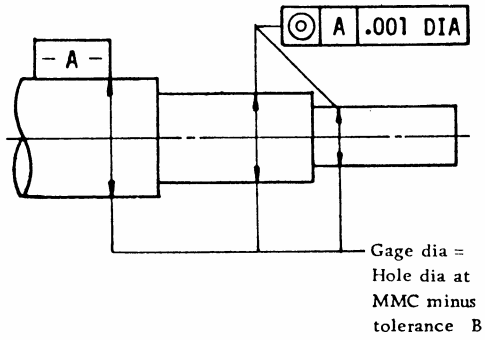


Figure 5b - Specifications for concentric hole gauge

4. Features On A Common Centerline

4.1 Individual Features in Line

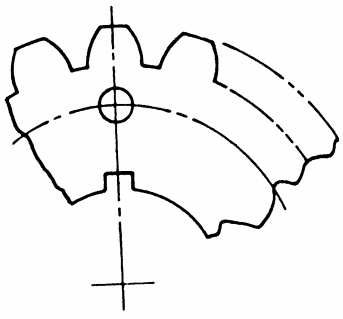
When two or more features are shown on a common centerline, the actual center plane or axis of each feature thus related must lie between two parallel planes that are separated by a distance equal to the alignment. Tolerance specified in table 3 and equidistant from a datum plane coincident with the shown centerline.

Table 3
Alignment Tolerance - Common Centerline

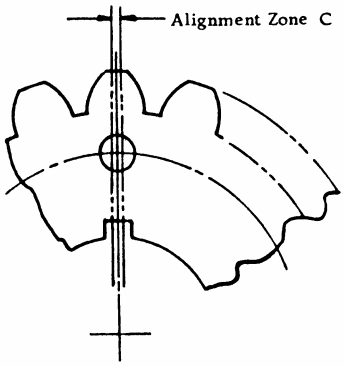
Distance between Features		
Over	**Thru**	**Alignment Tolerance (C)**
0	3	.015
3	6	.02
6	12	.03
12	∞	.06

Figure 6. Alignment of features in same section

This on the drawing:



Means this:

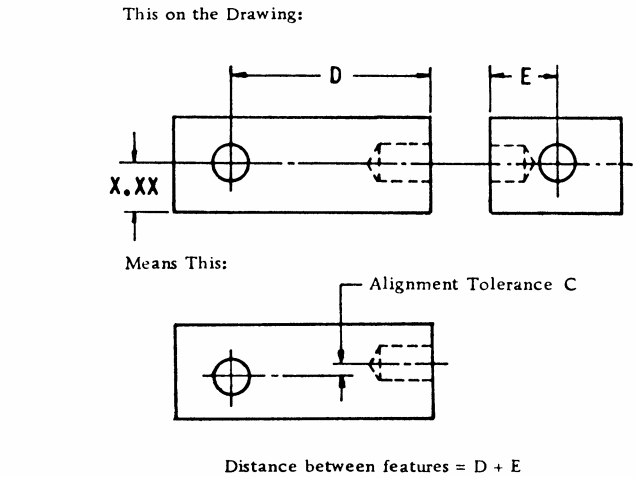


Unless otherwise specified, the pictured relationship between the tooth, hole, and keyway shall be held within the "alignment zone" (C) through the full length of the part.

Alignment controls for feature patterns and individual features apply to the axis or

center plane of the pattern or feature. Features within the pattern are related by positional tolerances independently assigned.

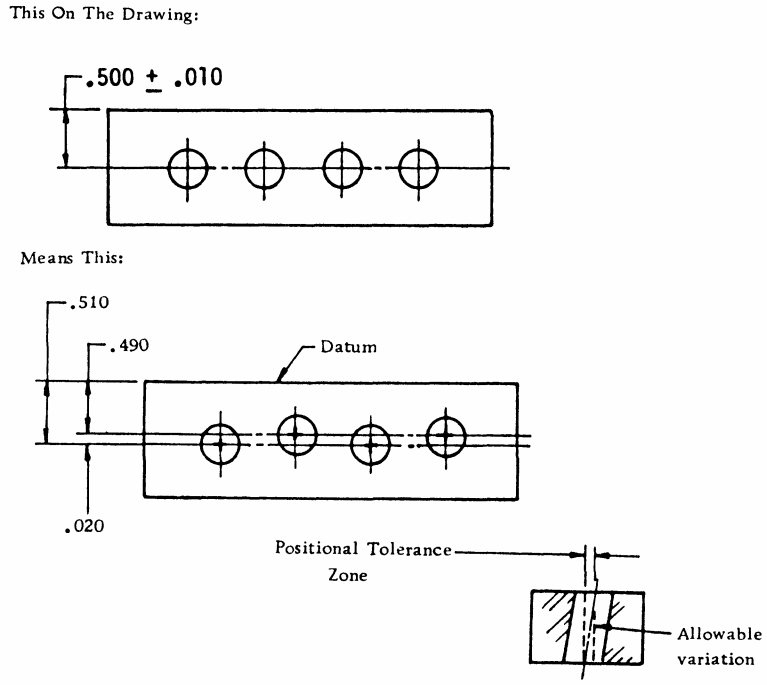
Figure 7. Alignment of features not in the same section



4.2 Hole Pattern

When a pattern of two or more holes is shown on a common centerline, the axis of the holes must lie between two parallel planes, which are separated by a distance equal to the positional tolerance. This zone extends the entire length and depth of the part.

Figure 8. Alignment of two or more holes shown on common center line



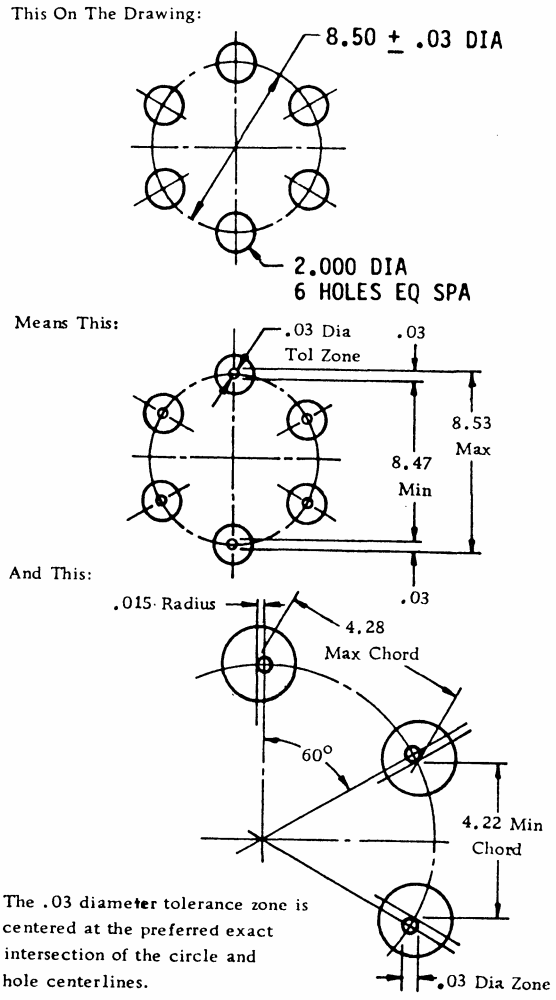
5. Cylindrical Features Located On A Circle

Cylindrical features located on a common circle, either by coordinate, diameter and chord, or diameter and angle dimensions, must be centered within cylindrical tolerance zones having a diameter based on the specified positional tolerance, according to the following formula. Each cylindrical zone shall control both position and squareness of the feature axis throughout the length or depth of the part.

$$\text{Tolerance zone diameter} = \frac{\text{Total positional tolerance}}{2}$$

Center of each feature tolerance zone is located by the preferred exact intersection of the circle and hole centerlines, and radial deviation is permitted in any direction from each center. Tolerances are noncumulative.

Figure 9. Alignment of holes shown on a circle

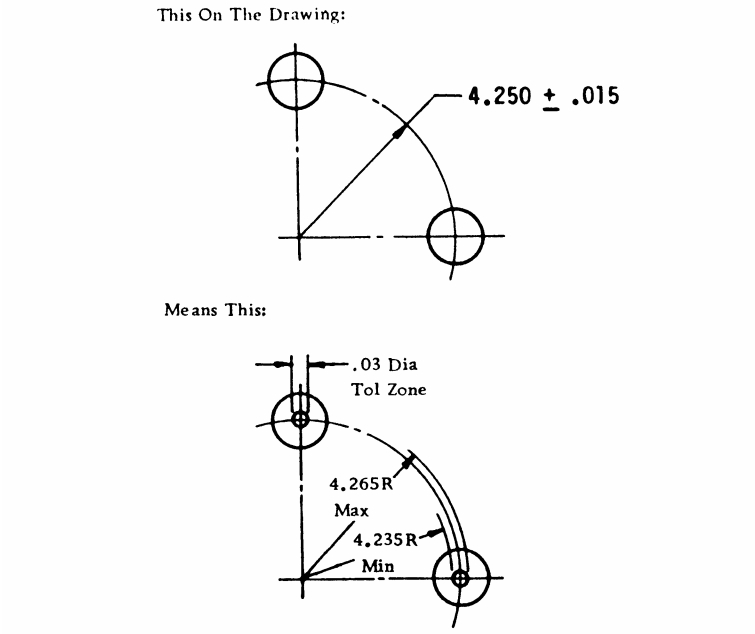


6. Cylindrical Features Located On A Radius

Cylindrical features located on a common radius by radius and chord or radius and angle dimensions must be centered within cylindrical tolerance zones having a diameter based on the specified positional tolerance. Each cylindrical zone shall control both position and squareness of a feature axis throughout the length or depth of the part.

Tolerance zone diameter = Total positional tolerance

Figure 10. Alignment of holes shown on a radius



The .03 diameter tolerance zone is centered at the preferred exact intersection of the radius and hole center lines.

7. Keyway And Slot Alignment In Bores And Turns

Keyways and slots are usually shown on centerlines of the part for convenience. Unless a qualifying note expresses latitude in positional control, the following alignment and position tolerances shall apply.

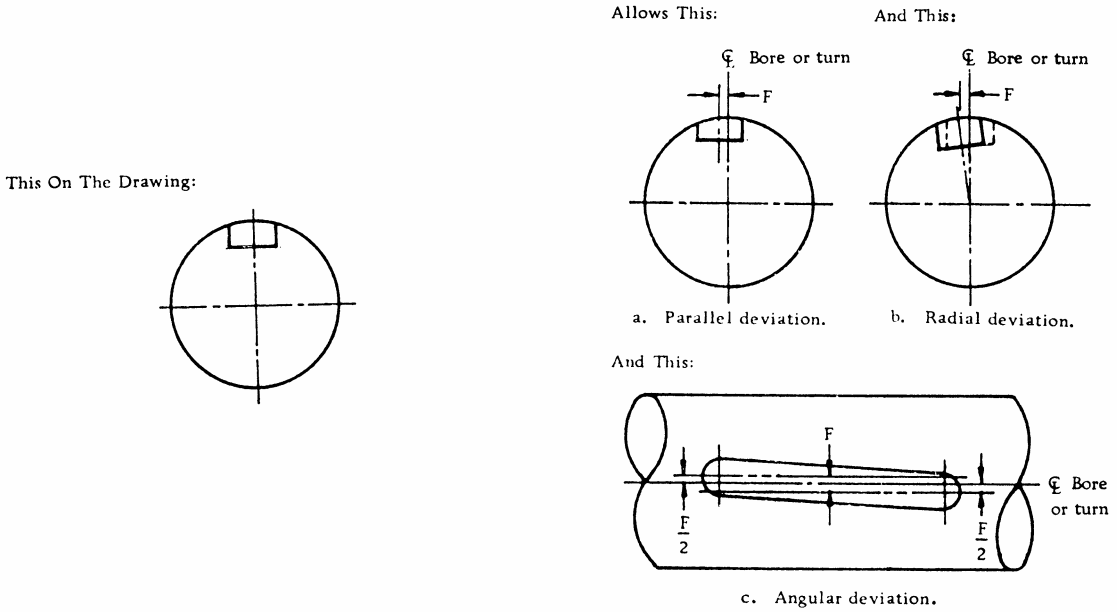
7.1 Single Keyway or Slot

Tolerance values in Table 4 control placement relative to the center plane on which the keyway or slot is shown and apply to parallel, radial and angular deviations.

Table 4
Alignment Tolerance - Keyways and Slots

Bore or Turn Diameter (in)	Keyway or Slot Length (in)	Alignment Tolerance (in) (F)
Over 0 Thru 2	Over 0 - Thru 2	.002
	Over 2 - Thru 10	.004
	Over 10	.004 + .001 for each 2” over 10”
Over 2 Thru 10	Over 0 -Thru 2	.004
	Over 2 -Thru 10	.006
	Over 10	.006 + .001 for each 1” over 10”
Over 10	Over 0 - Thru 2	.006
	Over 2 - Thru 10	.008
	Over 10	.008 + .003 for each 2” over 10”

Figure 1 1. Single keyway alignment control



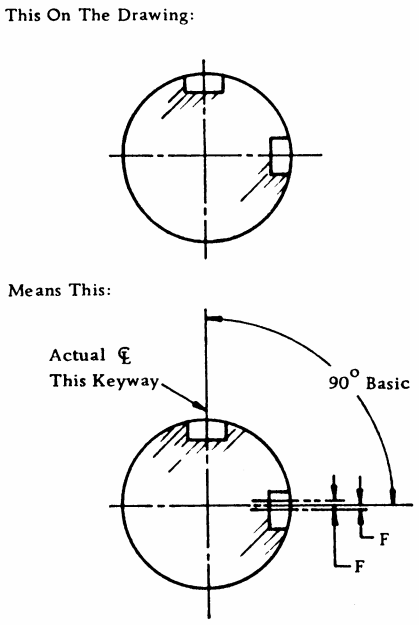
7.2 Multiple Keyways on a Common Centerline

When two or more keyways are shown on a common centerline, alignment deviation limits shall apply on an individual basis with respect to the common centerline. The keyways are assumed to be functionally unrelated unless more precise alignment is specified on the drawing.

7.3 Multiple Keyways in the Same Section

Two-or more keyways contained in the same cross section are assumed to be functionally related. In addition to individual alignment with the bore or turn as provided in Table 4, positional alignment with each other is also required. Thus, cumulative deviation shall not exceed the applicable "F" value from Table 4.

Figure 12. Alignment of Related Keyways



8. Holes On Cylindrical Surfaces

Round holes are usually shown on centerlines of a part for convenience. Unless a qualifying note expresses latitude in position control, the following alignment and position tolerances shall apply.

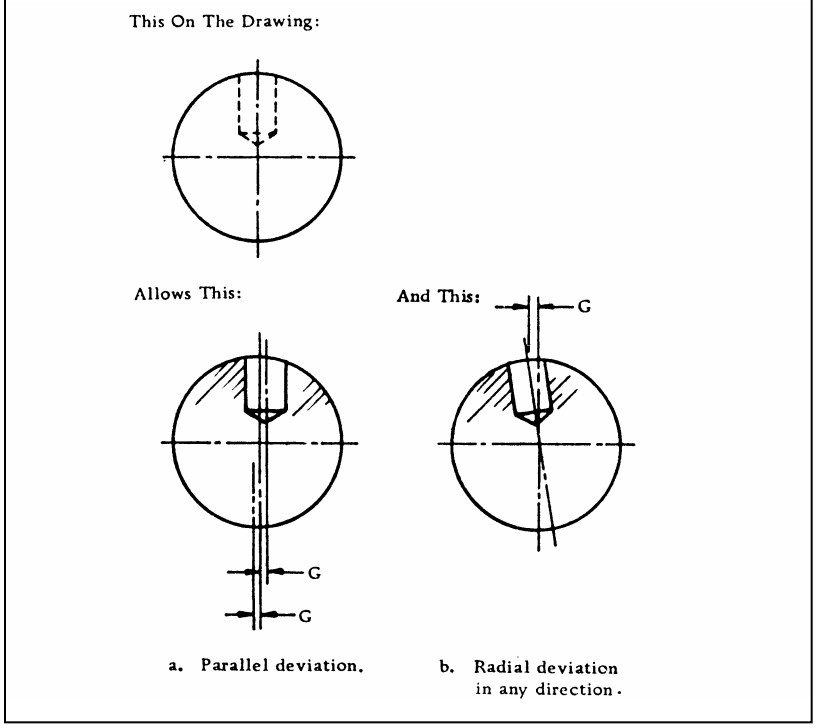
8.1 Single Hole

Tolerance values in Table 5 control placement relative to the center plane on which the hole is shown, and apply to parallel and radial deviations.

Table 5
Alignment Tolerances for Holes on a Cylindrical Surface

Surface Diameter		Alignment Tolerance
Over	**Thru**	**(G)**
0	2	.015
2	5	.020
5	10	.030
10	∞	.030 plus .003 for each 1" over 10"

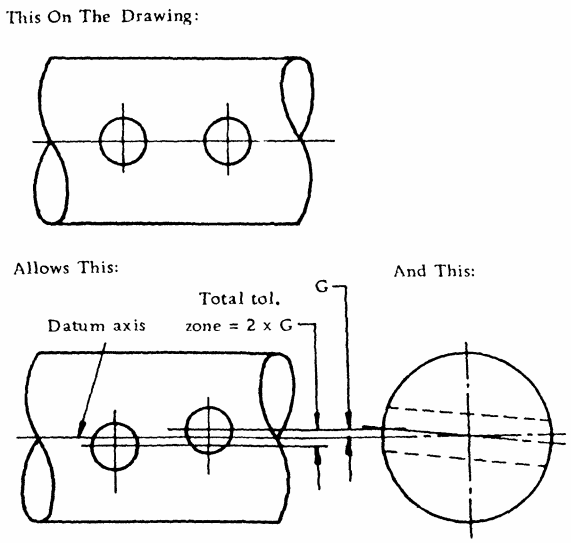
Figure 13. Single hole alignment control



8.2 Multiple Holes on a Common Axial Centerline

When two or more holes are shown on a common axial centerline, the axis of all the holes must lie between two parallel planes that are separated by a distance equal to twice the Alignment Tolerance given in Table 5.

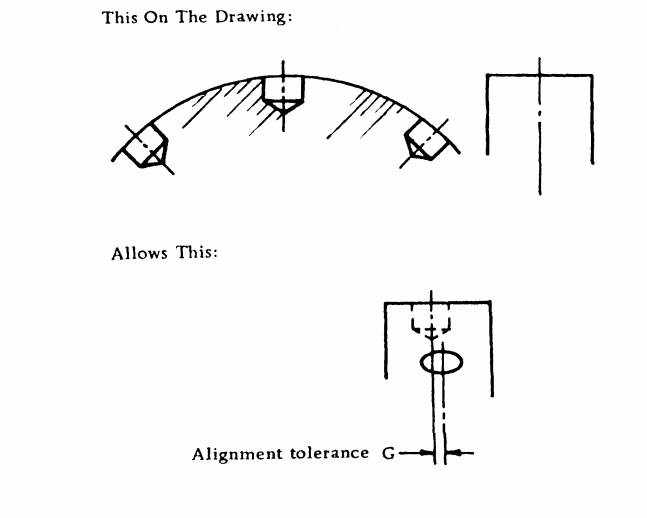
Figure 14. Alignment of Holes shown on a Common Axial Centerline



8.3 Multiple Holes Radially Located on a Common Circumferential Centerline

When two or more holes are shown on a common circumferential centerline the axis of all the holes must lie between two parallel planes that are perpendicular to the axis and separated by a distance equal to the Alignment Tolerance in Table 5.

Figure 15. Alignment of Holes shown in a Radial Arrangement



9. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-24

1. Coverage

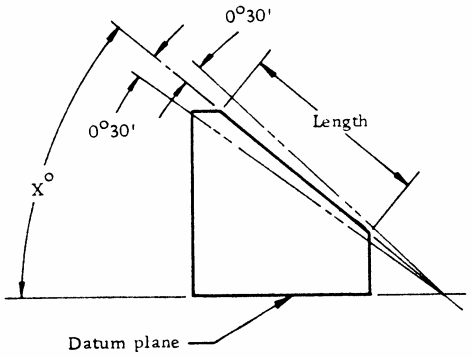
This standard establishes angularity control of surfaces shown at an angle and dimensioned with untoleranced angular values other than 90 degrees.

2. Machined Surfaces

Machined surfaces must be within ± ½° of the specified angle, regardless of length.

NOTE: This amounts to ± 0. 0087" per inch of length.

Figure 1 - Angular machined surface



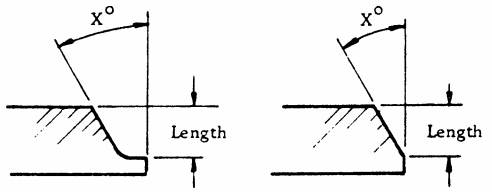
3. Angular Weld Preparations

Groove weld edge preparations shall be within the angular tolerance given in Table 1, regardless of length. Surface texture shall be no rougher than 1000 AA micro-inches.

Table 1
Angularity Tolerance - Weld Preparations

Method	Tolerance
Machined	± 2 °
M.A. Cut	± 5°

Figure 2 - Angular Weld Preparations



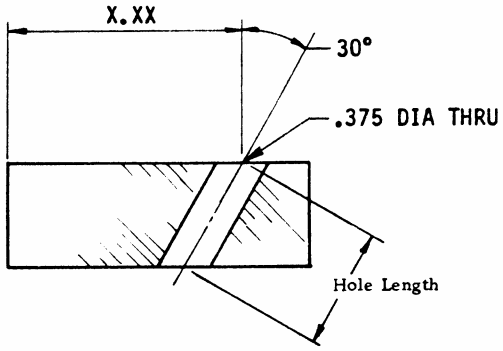
4. Holes Shown At A Specified Angle

Angularity tolerance for holes shown at a specified angle is given in Table 2. This tolerance applies only to the view in which the angle is specified (see figure 3).

Table 2
Angularity Tolerance - Angular Holes

Hole Diameter (inches)	Hole Length (inches)	Angularity Tolerance
Thru .25	Thru 1	± 2°
	Over 1 Thru 2	± 3°
Over .25 - Thru .75	Thru 1	± 1½°
	Over 1 Thru 2	± 1°
	Over 2 Thru 3	± 1½°
	Over 3 Thru 5	± 2°
Over .75	Thru 2	± 1°
	Over 2 Thru 3	± 1½°
	Over 3 Thru 5	± 2°

Figure 3. Angular hole example



5. Chamfers And Bevels

Angularity tolerance for chamfers and bevels is covered in Standard SPS-11.

6. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-24

1. Coverage

This standard establishes concentricity tolerances between two or more machined surfaces of revolution that are shown on a common axis.

2. Concentricity Tolerance

Cylindrical machined surfaces having a common axis shall be concentric within individually determined cylindrical tolerance zones whose axes coincide with the datum axis. The concentricity tolerance for each surface is a function of its size tolerance.

$$\text{Tolerance zone diameter} = \frac{\text{Total size tolerance}}{2}$$

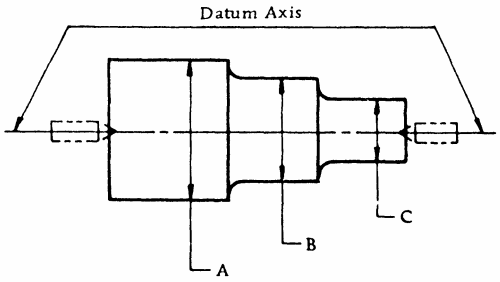
3. Determination Of Datum Axis

Since concentricity is always measured with respect to a datum axis, the various conditions that affect determination of this axis must be considered.

3.1 Parts with Established Centers

Related surfaces of revolution having established centers shall use the axis formed by these centers as the datum axis for checking concentricity of all the surfaces.

Figure 1. Example of concentricity tolerances for a part with centers.



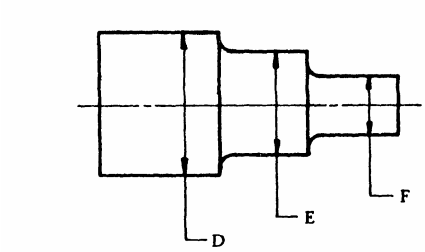
	A diameter	B diameter	C diameter
Upper and Lower Specification Limits	6.001 5.999	3.502 3.498	1.602 1.598
Total Size Tolerance	.002	.004	.004
Concentricity Tol.	.001	.002	.002

3.2 Parts without Established Centers

a) Surfaces of revolution related by a common axis, but without established centers, shall use the cylindrical surface having the smallest size tolerance for determination of the datum axis. All other surfaces of revolution shall be concentric about this axis, within the tolerance.

b) The axis shall be determined by rotating the part through 360°. Concentricity of the surfaces shall be checked in the same setup.

Figure 2. Example of concentricity tolerances for a part without centers.

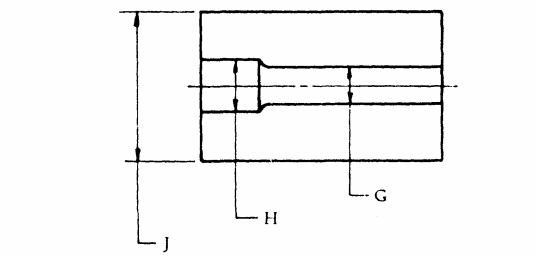


	D diameter	E diameter	F diameter
Upper and Lower Specification Limits	6.001 5.999	3.502 3.498	1.602 1.598
Total Size Tolerance	.002	.004	.004
Concentricity Tolerance	Datum Reference	.002	.002

3.3 Parts with Internal and External Surfaces of Revolution

Internal and external diametrical surfaces shown on a common axis shall be concentric about the axis of the related diametrical surface having the smallest size tolerance.

Figure 3. Example of concentricity tolerances for a part with related internal and external diameters.

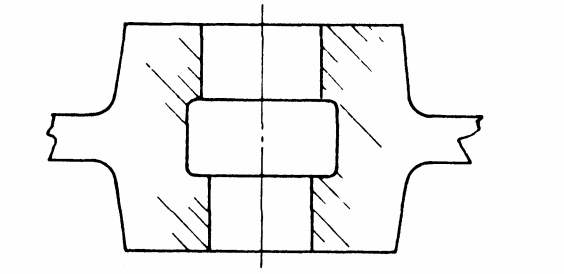


	G diameter	H diameter	J diameter
Upper and Lower Specification Limits	1.260 1.270	1.501 1.503	3.00 2.90
Total Size Tolerance	.010	.002	.10
Concentricity Tolerance	.005	DATUM	.05

3.4 Parts with related Surfaces of Revolution separated

Related diametrical surfaces separated by a distance within a part, but having a common axis, shall follow the same concentricity controls as those for adjacent diameters.

Figure 4. Example of related diameters separated



3.5 Parts with Datum Surface Specified

For parts without established centers and the same degree of tolerance for all related diametrical surfaces, the datum surface should be specified on the drawing. A functional surface with good accessibility is a logical choice.

4. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-24

1. Coverage

This standard establishes roundness or cylindricity controls applicable to round, machined surfaces.

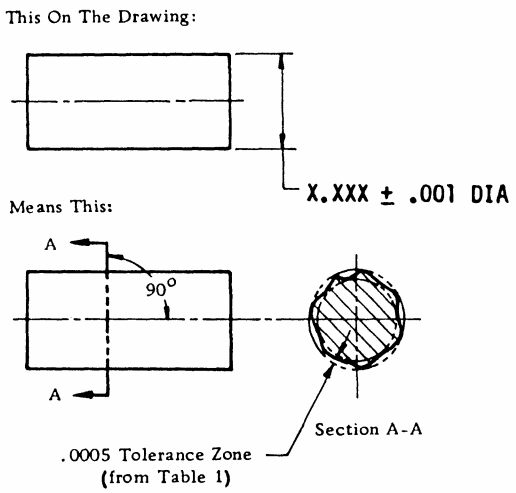
2. Roundness

The roundness tolerance value is derived from the diametrical size tolerance of the feature. Deviation must not exceed corresponding zone width, listed in Table 1. This zone is defined by two concentric circles perpendicular to the feature axis. Roundness control does not relate to any other feature as datum.

Table 1
Roundness Tolerance - Machined Parts

Total Diametrical Tolerance (in)	Roundness Tolerance Zone Width (in)
.001	.0003
.002	.0005
.004	.0007
.010	.001
.020	.002
.040	.005
.120	.010

Figure 1. Example of roundness tolerance application



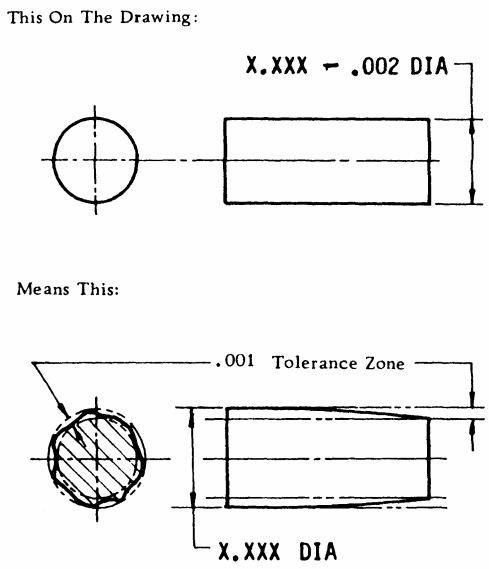
The entire periphery of the feature must lie within the zone. Roundness tolerance also applies to conical and other shapes if the cross section is circular.

3. Cylindricity

a) Where a true circular surface maintains a constant diameter throughout the length of a common axis, roundness becomes cylindricity. The tolerance zone then must lie between two concentric cylinders instead of two circles, and the tolerance controls parallelism and straightness as well as roundness of the considered surface.

b) Tolerance dimensions for the size of individual features control cylindricity as well as size.

Figure 3. Example of cylindricity tolerance application.



The feature must be within the specified tolerance of size and must lie between two concentric cylinders, one having a radius .001 larger than the other.

4. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-14

1. Coverage

This standard establishes flatness tolerances of machined surfaces and sheet metal Stampings.

2. Machined Surfaces

Flatness tolerances for machined surfaces are based on surface texture and length (or diameter). Flatness does not relate to a datum feature. However, the entire surface must be within the applicable size tolerance of the part.

Figure 1. Flatness of a machined surface

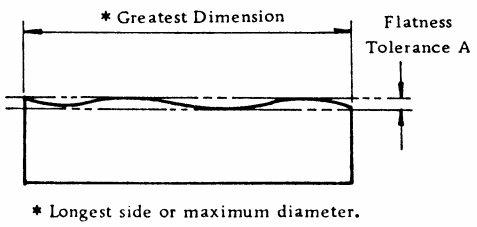
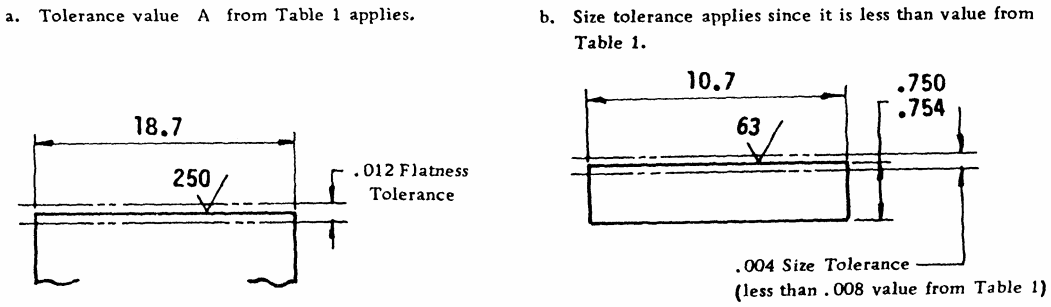


Table 1
Flatness Tolerance - Machined Surfaces

Surface Roughness AA – Micro-inches	Greatest Dimension (I.e. longest side or greatest dia.)		Flatness Tolerance (A)
	Over	Thru	
Over 0✓ - Thru 32✓	0	6	.001
	6	12	.002
	12	24	.004
	24	∞	.010
Over 32✓ - Thru 250✓	0	6	.004
	6	12	.008
	12	24	.012
	24	∞	.020
Over 250✓	0	12	.010
	12	∞	.020

Figure 2. Examples of flatness tolerance application



3. Stamped Or Punched Sheet Metal Parts

Plain sheet metal parts without louvers, dimples, or other designated depressions, shall be flat within the limits shown in Table 2. The acceptable method for checking flatness is shown in Figure 3. For parts with louvers or other designated depressions, these tolerance values apply individually to each continuous surface span.

Figure 3. Checking flatness of sheet metal

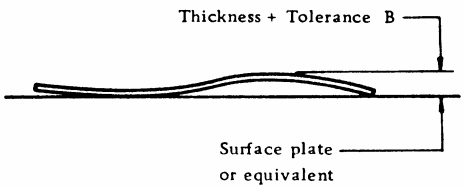


Table 2
Flatness Tolerance (B) - Sheet Metal

Sheet Thickness		**Greatest Dimension (I.e. longest side or maximum diameter)**				
Over	**Thru**	**0 to .999**	**1 to 2.999**	**3 to 9.999**	**10 to 19.999**	**Over 20**
0	.016	.010	.020	.035	.050	.080
.016	.047	.009	.017	.030	.040	.050
.047	.063	.008	.015	.025	.030	.040
.063	.094	.007	.010	.020	.025	.030
.094	.125	.006	.008	.020	.020	.025
.125	.250	.005	.008	.020	.020	.025

4. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-24

1. Coverage

a) This standard establishes parallelism tolerances for machined surfaces shown as equidistant flat planes, or shown equidistant from a datum plane. Tolerances apply to the entire surface and control flatness, if not specified, as well as parallelism.

b) This standard does not cover parallelism of holes or of elements or axes of cylindrical features.

2. Single Machined Surface

Figure 1. Parallelism of machined surface to a datum plane

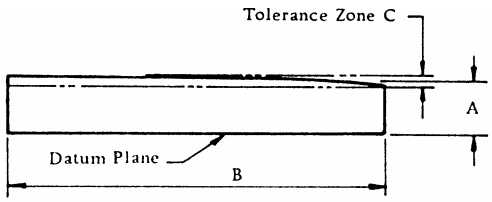
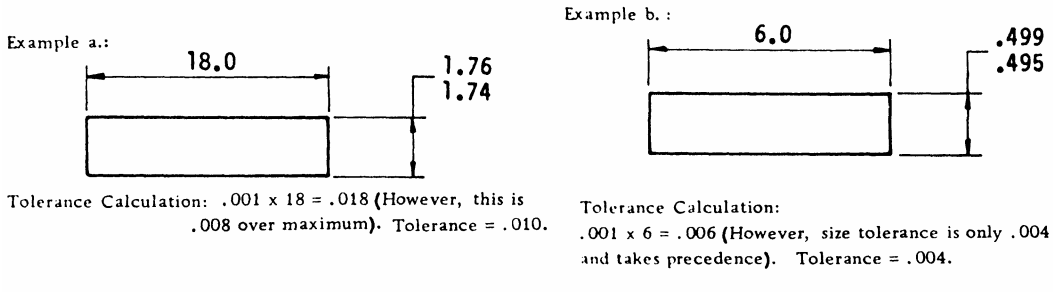


Figure 1. Parallelism of machined surface to a datum plane.

Table 1
Parallelism Tolerance - Machined Flat Surface

Total Tolerance for Dimension A	Parallelism Tolerance (C)
over .000 thru .020	.001 x B (.010 max)
over .020 thru .060	tolerance for A divided by 2
over .060	.030

Figure 2. Examples of parallelism tolerance calculation



3. Multiple Parallel Surfaces

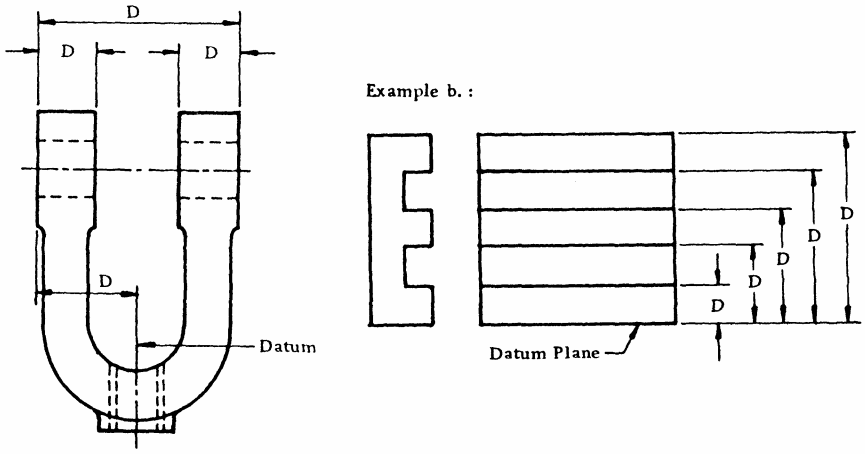
a) Parallelism tolerances applicable to two or more related machined surfaces are determined according to formulas given in Table 2 by applying values relevant to "D" dimensions indicated.

b) Parallelism tolerance between any two surfaces is independently determined and applied.

Table 2
Parallelism Tolerance - Machined Flat Surfaces

Total Tolerance for Dimension D	Parallelism Tolerance (E)
over .000 thru .020	.001x D (.010 max.)
over .020 thru .060	Tolerance for D divided by 2
over .060	.030

Figure 3. Examples of parallelism tolerances for parts with multiple parallel surfaces



4. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-24

1. Coverage

c) This standard establishes parallelism tolerances for machined surfaces shown as equidistant flat planes, or shown equidistant from a datum plane. Tolerances apply to the entire surface and control flatness, if not specified, as well as parallelism.

d) This standard does not cover parallelism of holes or of elements or axes of cylindrical features.

2. Single Machined Surface

Figure 1. Parallelism of machined surface to a datum plane

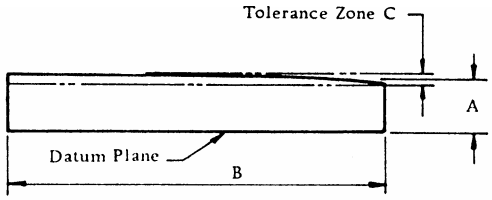
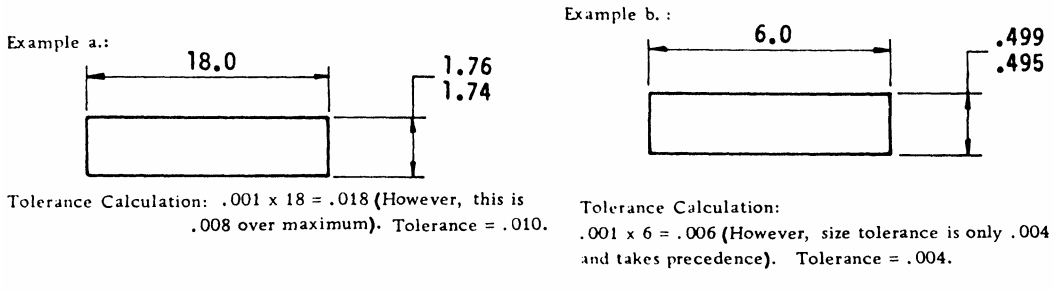


Figure 1. Parallelism of machined surface to a datum plane.

Table 1
Parallelism Tolerance - Machined Flat Surface

Total Tolerance for Dimension A	Parallelism Tolerance (C)
over .000 thru .020	.001 x B (.010 max)
over .020 thru .060	tolerance for A divided by 2
over .060	.030

Figure 2. Examples of parallelism tolerance calculation



3. Multiple Parallel Surfaces

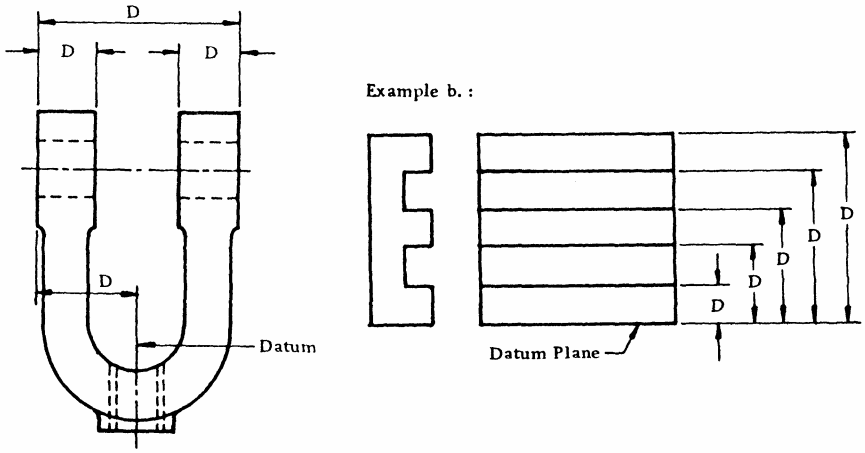
c) Parallelism tolerances applicable to two or more related machined surfaces are determined according to formulas given in Table 2 by applying values relevant to "D" dimensions indicated.

d) Parallelism tolerance between any two surfaces is independently determined and applied.

Table 2
Parallelism Tolerance - Machined Flat Surfaces

Total Tolerance for Dimension D	Parallelism Tolerance (E)
over .000 thru .020	.001x D (.010 max.)
over .020 thru .060	Tolerance for D divided by 2
over .060	.030

Figure 3. Examples of parallelism tolerances for parts with multiple parallel surfaces



4. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-24

1. Coverage

This standard establishes perpendicularity tolerances for features or surfaces shown at a right angle (90°) to a datum plane or axis.

2. Machined Surfaces

Machined surfaces shall be perpendicular within .002 per inch of length, up to a maximum deviation of .03 of the shorter surface. The longer surface shall be considered the datum reference.

Figure 1. Perpendicularity of machined surfaces

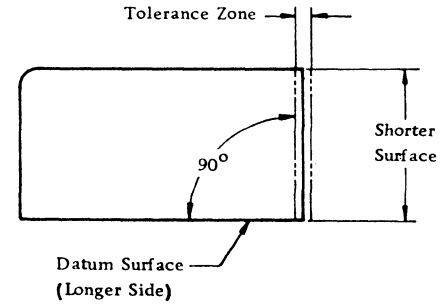


Figure 1. Perpendicularity of machined surfaces.

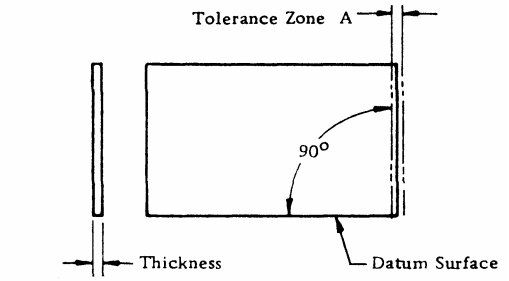
Tolerance Zone = .002 x Length Shorter Surface
(.03 maximum)

3. Sheared Sheet And Plate

3.1 Related Sides

Related sides of sheared parts shall be perpendicular within the applicable tolerance A values in Table 1. Related sides of punched surfaces are a function of the die manufactured for the operation. Tolerances will correspond to those for machined surfaces, as shown in paragraph 2, above.

Figure 2. Perpendicularity of cut sides



Tolerance zone “A” applies to each side with respect to adjoining sides shown at right angles.

Table 1
Perpendicularity Tolerance

Thickness (inches)		
Over	**Thru**	**Tolerance A (inches)**
0	.188	.06
.188	.500	.08
.500	.750	.10

3.2 Condition of Cut Surfaces

Surface condition through the thickness of cut plate or sheet materials shall be within the following limits of acceptability. The cut edge surface (dimension C in Figure 3) shall be at least 1/3 of the material thickness (dimension B). The remaining surface (B minus C) may break away as shown within tolerance D values in Table 2.

Figure 3. Cut surface conditions

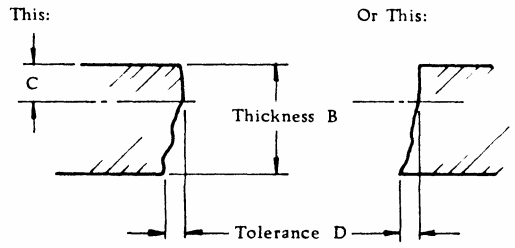


Table 2
Perpendicularity Tolerances for Punched or Sheared Surfaces

Thickness B (inches)		**Tolerance D (inches)**	
Over	**Thru**	**Punched**	**Sheared**
0	.015	.003	.008
.015	.040	.005	.010
.040	.125	.010	.015
.125	.250	.015	.030
.250	.500	.020	.050
.500	.750	.025	.080

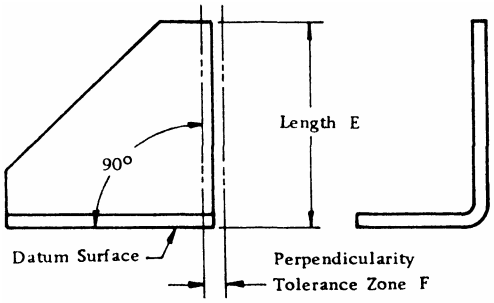
4. Formed Sides

4.1 Cut Surface to Bend

Surfaces shown at a right angle (90°)to a bend shall be perpendicular to the bend within these limits:

Length E (inches)	Perpendicularity Tolerance Zone F
Thru the first 6 inches of length E	.03
Over 6 inches thru 12 inches	.06
For every inch over 12 inches	.06 + (.002 per each extra inch)

Figure 4. Perpendicularity of cut surface to bend.

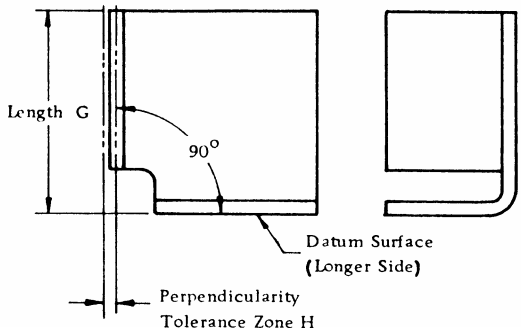


4.2 Bend to Bend

Two bends are shown at a right angle (90°) shall be perpendicular within these limits, as shown in figure 5.

Length G (inches)	Perpendicularity Tolerance Zone H
Thru the first 6 inches of length G	.03
Over 6 inches thru 12 inches	.06
For every inch over 12 inches	.06 +(.002 per each extra inch)

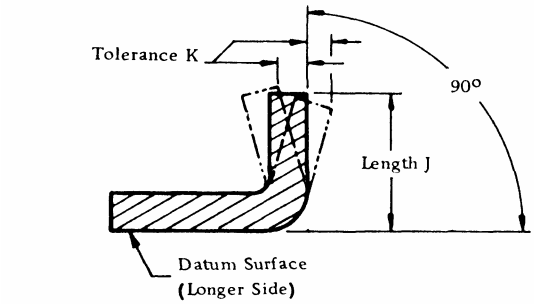
Figure 5. Perpendicularity of bend to bend



4.3 Right Angle (90°) Bends

A bend shown at a right angle (90°) shall be perpendicular within the applicable tolerance value given in Table 4.

Figure 6. Perpendicularity of a bend



Length of Short Leg J		Material Thickness (inches)					
Over	**Thru**	**>.000 Thru .015**	**>.015 Thru .035**	**>.035 Thru .083**	**>.083 Thru .125**	**>.125 Thru .250**	**>.250 Thru .500**
0	2	.02	.02	.02	.03	.03	.04
2	4	.03	.03	.03	.04	.04	.05
4	6	.04	.04	.05	.05	.06	.07
6	10	.05	.06	.07	.08	.08	.09
10	20	.07	.07	.08	.09	.09	.10

5. Cylindrical Machined Projections

The axis of a cylindrical machined projection must be within a cylindrical tolerance zone perpendicular to the datum surface and equal to the projection length; the tolerance zone diameter shall not exceed the applicable listed percentage of the diametrical size tolerance for the projection.

Figure 7. Perpendicularity of cylindrical projections

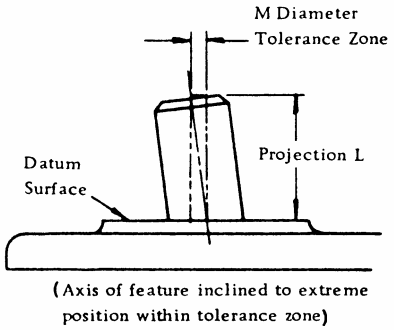


Table 6
Perpendicularity Tolerance - Cylindrical Projections

Projection Length L (in)		
Over	**Thru**	**Tolerance M = % of size tolerance**
0	2	50%
2	5	75%
5	-	100%

Example:
A cylindrical feature, .749/.745 diameter, projects 1.00 from the datum. A .004 diametrical size tolerance for a 1.00 projection permits out-of-squareness of .002 (50% of .004), thru .747 diameter. Over .747 thru .749 diameter the perpendicularity tolerance decreases to the point where perfect form is required at maximum material condition (.749).

6. Hole To Surface

The axis of a hole must be within a cylindrical tolerance zone perpendicular to the datum surface; the tolerance zone diameter shall not exceed the applicable listed value.

Figure 8. Perpendicularity of hole to surface

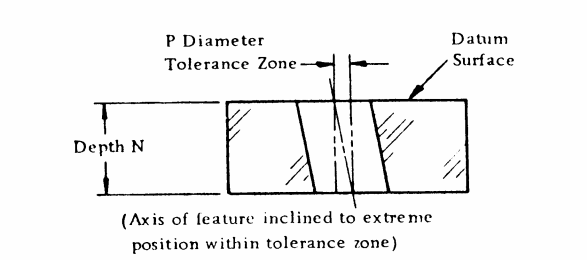


Table 7

Perpendicularity Tolerance - Hole to Surface
(all dimensions in inches)

N - Depth of Hole		Diameter of Hole					
Over	**to**	**>.000 to .125**	**>.125 to .250**	**>.250 to .500**	**>.500 to 1.00**	**>1.00 to 2.0**	**>2.0**
0	.5	.010	.010	.008	.005	.005	.005
.5	1	.015	.015	.010	.010	.010	.010
1	2	.030	.020	.015	.015	.015	.015
2	3	-	.030	.020	.020	.020	.020
3	4	-	-	.025	.025	.025	.025
4	5	-	-	-	.030	.030	.030

Perpendicularity Tolerance for
Holes 5 or more inches deep = 0.032 + (.005" for each additional inch)

7. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-24

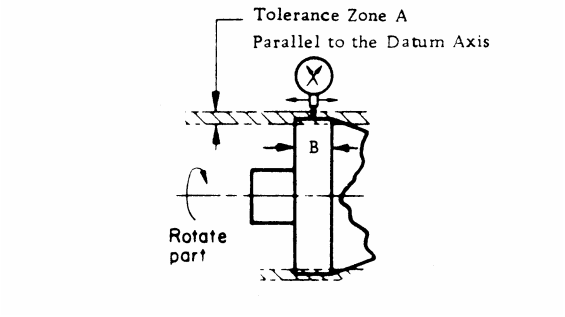
1. Coverage

This standard establishes run out control as applied to machined surfaces having a common axis.

2. Run Out On Diameter

Run out is not to be confused with cylindricity and roundness, which are measured without respect to a datum. Run out on a machined diameter with respect to a common axis shall not exceed the limits in Table 1, or the diametrical size tolerance, whichever is most restrictive.

Figure 1 - Run out on a Machined Diameter



3. Run Out Of Face

Run out on a machined face with respect to axis or outside diameter shall not exceed the limits in Table 1.

Figure 2 - Run out on a Machined Face

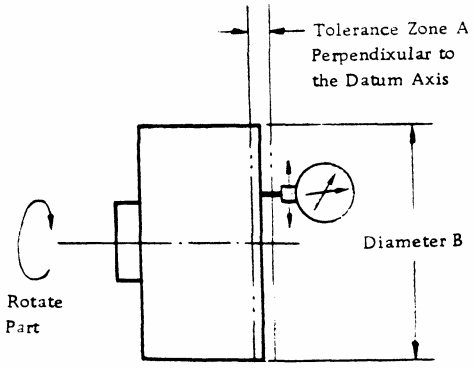


Table 1
Run out Tolerance - Machined Surfaces

Surface Texture	*Dimension B (inches)		Run out Tolerance
AA Micro-inches	**Over**	**Thru**	**Inches **A**
Over 0 – Thru 32	0	6	.001
	6	12	.002
	12	24	.004
	24	36	.006
	36	48	.008
	48	-	.010
Over 32 – Thru 125	0	6	.002
	6	12	.004
	12	24	.008
	24	36	.012
	36	48	.016
	48	-	.020
Over 125 – Thru 250	0	6	.003
	6	12	.006
	12	24	.012
	24	36	.018
	36	48	.024
	48	-	.030

* Diameter of face or length of cylindrical surface.

** Tolerance zone A as shown in Figures 1 and 2.

4. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-24

1. Scope

This standard establishes tolerances and conditions of edges, corners, and fillets on finished or semi-finished parts.

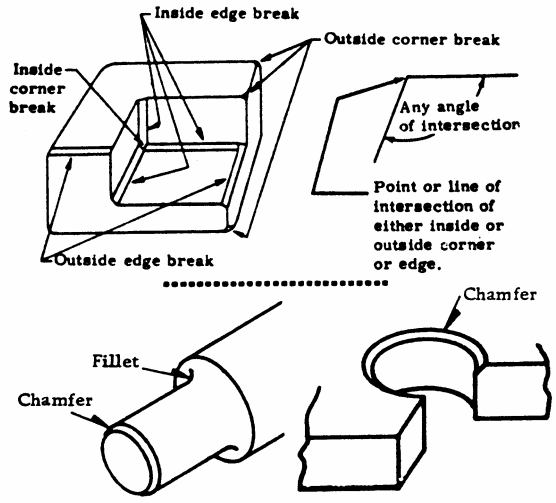
2. Definitions

Edge is the intersection of two surfaces.

Corner is the intersection of three or more surfaces.

Fillet is a continuous transition, connecting inside surfaces.

Figure 1. Edge and corner conditions



3. General Requirements

3.1 Burr Removal

With the exception of sheet metal parts (reference section 3.7), all burrs shall be removed.

3.2 Manufacturing Methods

Unless otherwise specified on the drawing, edges and corners shall be conditioned by the most practical method available. Manufacturing shall select the method and proper tools.

3.3 Sharp Corners And Edges

When a sharp corner or edge is required for functional reasons, the drawing will specify "SHARP" at the appropriate location. Corners or edges designated as "SHARP" shall not exceed a break of 0.005 inches.

3.4 Safety Requirements

In the absence of specific requirements and at the discretion of manufacturing, all may be conditioned for safe handling within the limits of this standard. If further conditioning beyond the scope of this standard is required, a drawing revision should be requested.

3.5 Outside Edges And Corners

Outside edges and corners may be broken (chamfer or radius) within Table 1 limits.

Table 1
Outside Edge and Corner Chamfers

Length of Edge or Part Diameter		**all dimensions in inches**
Over	**Thru**	**Maximum * Chamfer or Radius**
0	5	.03
5	10	.06
10	20	.12
20	-	.20

* Chamfer or radius may not exceed 20% of part thickness in any case.

3.6 Inside Edges And Corners

A minimum .015 inch fillet or radius is acceptable for inside edges and corners.

3.7 Sheet Metal Burrs

Burred edges on sheet metal parts are allowable within Table 2 limits.

Table 2
Allowable Burr Heights - Sheet Metals

Thickness of Sheet Metal		**all dimensions in inches**
Over	**Thru**	**Maximum Burr Height**
0	.020	.002
.020	.030	.003
.030	.038	.004
.038	.080	.005
.080	.188	.006

3.8 Burr Side

If no indication of BURR SIDE is given on the drawing, the burr side may be considered optional and the most practical method of manufacture should be the deciding factor.

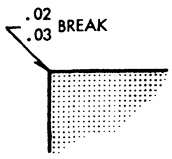
4. Specified Edge And Corner Condition

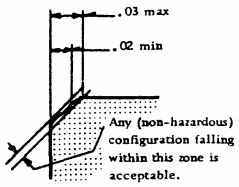
This section outlines the requirements and interpretation of specified edge and corner conditions.

4.1 Edge And Corner Breaks

a) When a specific limit of break is desired, it will be shown as follows:

This on the drawing: Means this:

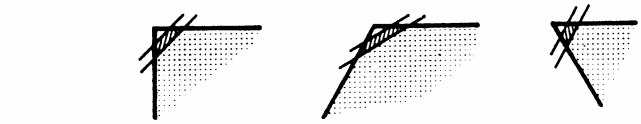




b) When an untoleranced or nominal dimension is used, ±20% tolerance is allowable.

This on the drawing: Means this:

c) The zone defining acceptable break limits shall be equally spaced on the intersecting surfaces, regardless of the included angle.



4.2 Chamfers And Bevels

A chamfer (or bevel) is a flat edge or corner break having a specific angle and size. Tolerance on chamfers is ±2°.

a) Limiting or bilateral dimensions shall be interpreted as illustrated.

This on the drawing: Means this:

b) Untoleranced or nominal dimensions shall be interpreted as illustrated.

This on the drawing: Means this:

4.3 Fillets

When a fillet is shown on a part but no size is indicated, it may be any size convenient to manufacturing.

4.4 Machined Radii

a) Application (Draftsman's Reference)
The specification of machined radii should be limited to these areas:
1) Where a specific radius is required to ensure proper mating or clearance between parts.
2) Where a minimum radius is required to reduce local stresses.
3) Where a minimum radius is required to perform a particular function, e.g., latch face, cam, etc.

b) Specification of Radii
The selection of machined radii should be limited to sizes corresponding to standard cutting tools and gauges, and specified in this manner:

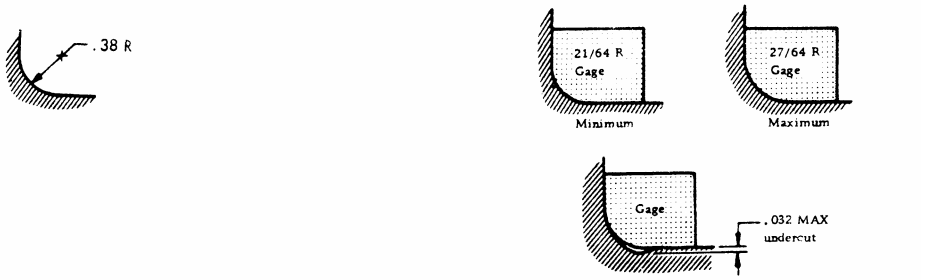
Specification	Meaning
.XXXR Max	Specified size or smaller
.XXXR Min	Specified size or larger
.XXX .XXX R	Within specified limits
.XXX R	Nominally, within limits notes in table 3
BY CONTOUR CONTROL	See 4.4e

c) Nominally Expressed Radii

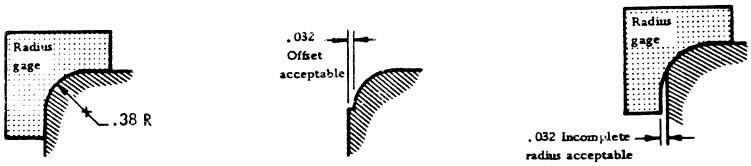
1) Radii, expressed in nominal two-place decimals, shall be selected from Table 3. This will permit high and low limits to be inspected with standard fractional radius gauges through .500R (Note the permissible undercut for nominally expressed radii).

2) The contours of nominal radii shall fall within a zone equal to the difference between the high and low limit gauges per Table 3.

This on the drawing: Means this is acceptable:



3) External radii are governed by the same conditions noted above. Offset values are equal to the acceptable undercut values for the same nominal radius.



d) Blending of Radii

1) When the undercut and offset conditions (in the previous section) are not acceptable, the radius callout shall include the term BLEND. This will require a uniform curvature, blended smoothly with the adjacent surfaces, within the maximum and minimum limits.

2) This is a more restrictive form of radius control and should only be used when justified by specific stress or functional requirements.

Specification of blend requirement

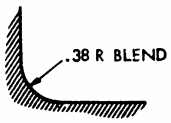


Table 3
Nominal Radius Limits

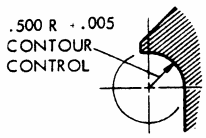
		Standard Radius Gauge*			
Drawing Radius	**Tolerance**	**Nominal**	**Lo-Limit**	**Hi-Limit**	**Max Undercut or Offset**
.015	±.015	1/64	1/64	1/32	.005
.030	±.015	1/32	1/64	3/64	.010
.050	±.032	3/64	1/64	5/64	.015
.060		1/16	1/32	3/32	
.090		3/32	1/16	1/8	
.120		1/8	3/32	5/32	
.160		5/32	1/8	3/16	
.190		3/16	5/32	7/32	
.220		7/32	3/16	¼	
.250		¼	7/32	9/32	
.280		9/32	¼	5/16	
.310	±.046	5/16	17/64	23/64	.032
.380		3/8	21/64	27/64	
.440		7/16	25/64	31/64	
.500		½	29/64	35/64	
to 1.000	±.060	-	-	-	-
to 2.000	±.100	-	-	-	-

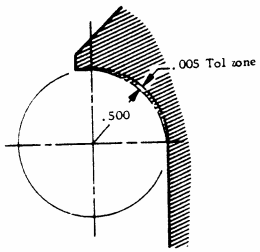
e) Critical Radii

1) When the function of a radius requires restrictive control with smooth blending at the tangent point, the radius shall be specified as Illustrated.

2) This form of radius tolerance requires that the radius be within the illustrated zone, gradually diminishing toward the tangent points. This is a very restrictive method of tolerancing and should only be used when fully justified. Inspection is usually performed on an optical comparator.

This on the drawing: Means this:

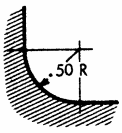


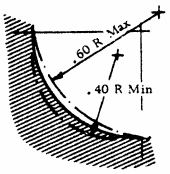


f) Cast or Formed Radii

Unless otherwise specified directly on the drawing, all radii specified in nominal terms on castings, forgings, and formed metal parts shall be within ±20% of the nominal radius.

This on the drawing: Means this:





5. Hole Edges

Manufacturing will be permitted to break edges of plain holes, counterbores, and tapped holes to provide safety and to facilitate assembly within the noted limits.

5.1 Plain Holes And Counterbores

Unless otherwise specified on the drawing, the maximum countersink or break permitted on plain holes and counterbores shall be as shown in Table 4.

Table 4
Edge Breaks on Plain Holes and Counterbores

Nominal Hole Size		**all dimensions in inches**
Over	**Thru**	**Maximum Edge Break - Each Side**
0	.25	.020
.25	.500	.025
.500	-	.030

5.2 Punched Holes

Punched holes may have burrs removed, depending upon application and safety requirements.

5.3 Tapped Holes

Edge conditions of tapped holes are covered in Shop Practice Standard 533.

6. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-24

1. Scope

This standard establishes acceptable size limits for untoleranced holes.

2. Application

List implied tolerances and preferred hole sizes of Table 2 are for clearance holes. The tolerances are not applicable to holes drilled for tapping, and the list of sizes is not intended to be used for selection of tap drills. For punched holes an additional tolerance applies as explained in Section 3.

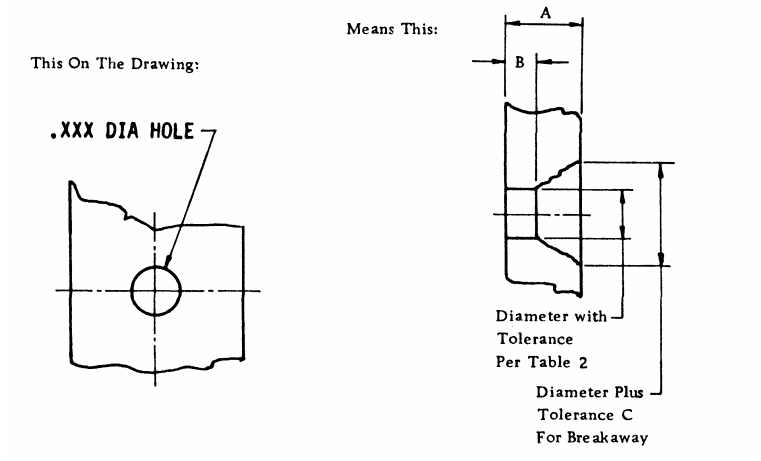
3. Hole Surface

Hole surface texture of 500 AA micro-inches or less is acceptable unless otherwise specified. Punched hole surfaces through the thickness of plate or sheet materials shall be within the following limits of acceptability. The cut surface (dimension B in Figure 1) shall be at least 1/3 of the material thickness (dimension A). Diameter through this thickness shall be within diametrical hole tolerance established in Table 2. The remaining surface (A minus B) may break away an additional amount as shown, within tolerance C values in Table 1.

Table 1
Punched Hole Surface Tolerance

Thickness (A)		
Over	**Thru**	**Hole diameter with diametrical tolerance + tolerance C**
0	.015	.006
.015	.040	.008
.040	.125	.020
.125	.250	.030
.250	.500	.040
.500	.750	.050

Figure 1. Punched hole surface conditions



4. Untoleranced Holes And Holes Toleranced With A Note Referencing This Standard

If a hole does not have a tolerance and the drawing does not have a general tolerance block or the drawing has a note referencing this standard, then the following tolerances apply:

Table 2
Diametrical Hole Tolerance and Preferred Sizes

Hole Diameter			
Over	**Thru**	**Tolerance**	
0	.125	-.002	+.005
.125	.250	-.002	+.006
.250	.500	-.002	+.008
.500	.750	-.002	+.009
.750	1.000	-.002	+.010
1.000	2.000	-.002	+.016
2.000	-	-.005	+.025

5. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-24

1. Scope

This standard establishes implied thread forms, thread classes and tolerances for the axial length of externally and internally threaded features of parts.

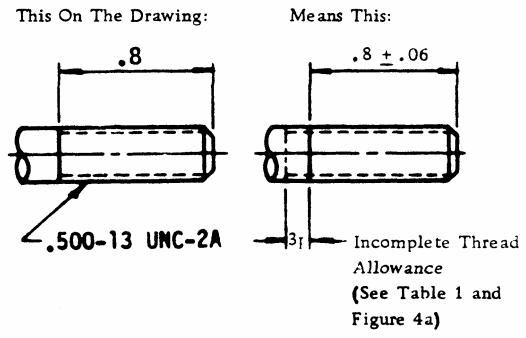
2. Thread Form And Thread Class

Unless otherwise specified, all threads must conform to the American National Standard for Unified Screw Threads (ANSI - B1.1). Unless otherwise specified, all external threads shall be class 2A and all internal threads shall be class 2B.

3. Thread Length

Dimensioned axial length of external threads shall carry an implied tolerance of ±. 06 for diameters through one inch and ±. 12 for diameter over one inch.

Figure 1 - External Thread Length



4. External Threads

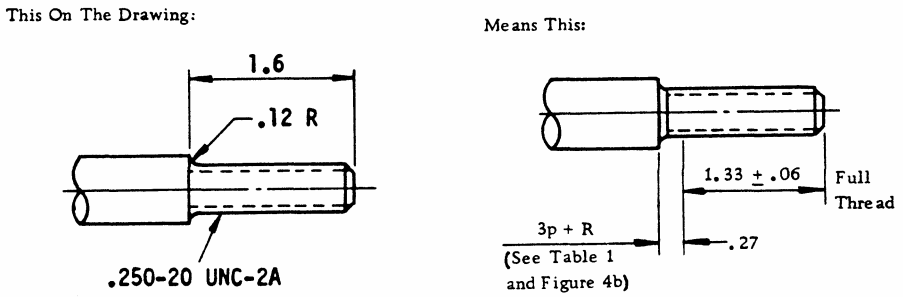
4.1 Straight Shank

Dimensioned or called out axial length shall be interpreted as full effective thread. This includes permissible chamfered end but does not include allowance for incomplete threads. See figure 1.

4.2 Threads To Shoulder

a) Any thread shown to a shoulder shall be interpreted as including the normal incomplete thread allowance plus the fillet radius. Incomplete threads shall not encroach upon the fillet area. See figure 2.

Figure 2 - Threads to shoulder



b) If a specific full thread length is required, it should be stated in the drawing call out or dimensioned with provisions for incomplete threads or thread relief, amounting to at least three thread pitches, as shown in Figure 4. See Table 1 for specific values.

Figure 3 - Thread Relief Allowance

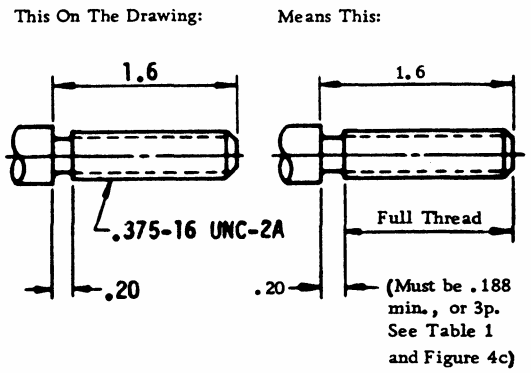


Table 1
Thread Run out and Relief Allowances and Profile Component Values

	Values for Pitch Multiples		Chamfer, Countersink and Relief Components	
Threads per Inch	**1p**	**3p**	**Int. Thread A**	**Ext. Thread B**
40	.025	.075	.008	.037
36	.028	.083	.009	.041
32	.031	.094	.010	.046
28	.036	.107	.011	.053
24	.042	.125	.013	.061
20	.050	.150	.014	.072
18	.056	.167	.016	.081
16	.062	.188	.018	.090
14	.071	.214	.020	.103
13	.077	.231	.022	.110
12	.083	.250	.023	.120
11	.091	.273	.025	.130
10	.100	.300	.027	.143
9	.111	.333	.030	.158
8	.125	.375	.033	.178
7	.143	.429	.038	.203
6	.167	.500	.044	.237
5	.200	.600	.052	.283
4½	.222	.667	.058	.314
4	.250	.750	.064	.353

Thread relief diameter can be determined from the following formula:

Relief Diameter = Major Diameter - B

Example: Thread relief diameter for a 3/8-16 UNC-2A is:

.375 Major Diameter
- .090 B value from table 1
.285 - .020 Dia. (Tolerance from table 2)

Table 2
Tolerance for Relief, Chamfer, and Countersink Diameters

Major Diameter				
Over	**Including**	**Relief Diameter**	**External Chamfer**	**Internal Countersink**
0	.250	-.010	-.010	+.010
.250	.500	-.020	-.020	+.020
.500	-	-.030	-.030	+.030

4.3 External Thread Chamfer

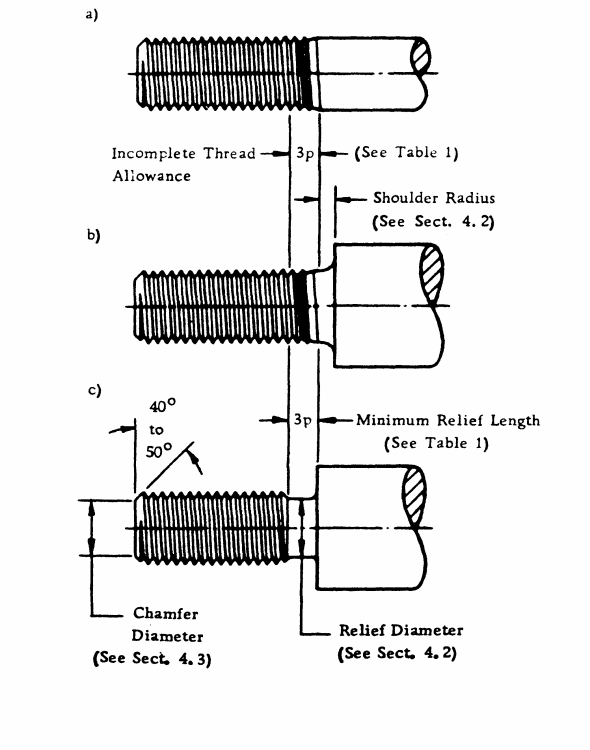
Normal allowable chamfer shall be 40 to 50 degrees from the chamfer diameter. Formula for determining this diameter is:

Chamfer Diameter = Major Diameter - B

Example: The chamfer diameter for ½-13 UNC-2A is:

.500 Major diameter
\- .110 B value from table 1
.390 - .020 diameter (tolerance from table 2)

Figure 4 - External Thread Profiles



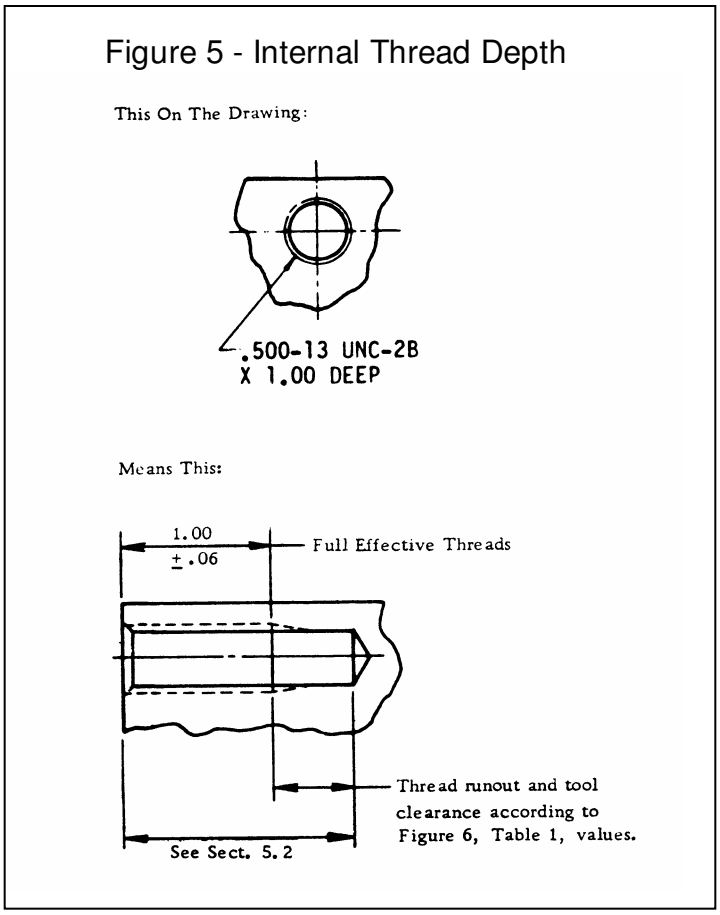
5. Internal Threads

5.1 Effective Thread Depth

Dimensioned or called out axial depth of internal threads shall be interpreted as full effective thread to the specified depth. There shall be no incomplete threads within the specified thread depth. See Figure 5. Number of incomplete threads is not significant, provided the full effective thread depth is as specified.

NOTE: For all blind hole applications the drawing must state the depth of complete thread required.

Figure 5 - Internal Thread Depth



5.2 Hole Depth

When tap hole depth is specified on the drawing, a standard blind hole tolerance of ±. 06 shall apply (the hole apex not included). See figure 5.

When no hole depth is specified, allowance for thread run-out and tool clearance shall be according to figure 6 and table 1.

NOTE: Depth of the tap hole is an optional callout, needed only when design conditions require close control.

5.3 Countersink Before Tapping

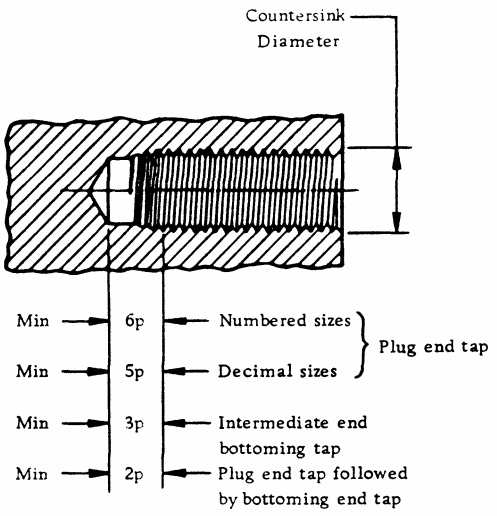
Countersinking preparation of a hole before tapping is a preferred practice. When a countersink is used, its diameter should conform to the following formula:

Countersink Diameter = Major Diameter + A

Example: The countersink diameter for a ¾-l6 UNF-2B thread is:

.750 Major Diameter
+ .018 "A" value from table 1
.768 + .030 Dia. (tolerance from table 2)

Figure 6 - Internal Thread Profile



5.4 Gauging With Thread Ring And Plug Gauges

A calibrated nogo thread gauge may be threaded a maximum of three revolutions onto an acceptable thread.

6. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-24

1. Scope

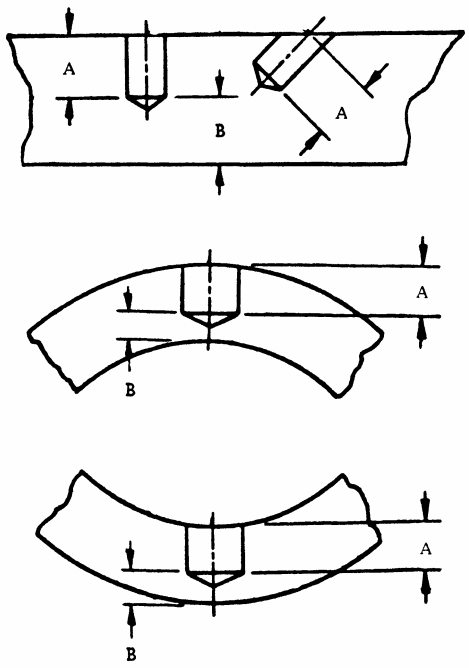
This standard establishes tolerances for depth of blind holes and counterbores, and countersink and spotface limits.

2. Blind Hole Depth Tolerance

a) Blind hole depth "A", as shown in Figure 1, not including the tool lead (or point), shall be within ±.06 inch tolerance when measured between machined surfaces. That is when the surface where the hole is applied and a second surface, which acts as a datum for locating the part, are both machined. When only one machined surface is involved, either a datum surface or the surface where the hole is applied, the stock or casting tolerance shall be added to the hole depth tolerance to determine the total allowance.

b) Holes dimensioned per "B", from a surface opposite the starting point of the hole, shall be within the same tolerances as those dimensioned per "A".

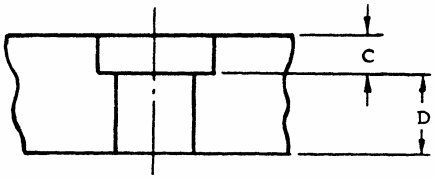
Figure 1 - Blind Hole Depth Tolerance



3. Counterbore Depth Tolerance

Counterbore depth "C", as shown in Figure 2, shall be within ± .03 inch tolerance for depths to and including one inch, and ± .06 inch for depths over one inch when measured between machined surfaces. When only one machined surface is involved, the stock or casting tolerance shall be added to the counterbore depth tolerance to determine the total allowance.

Figure 2 - Counterbore Depth Tolerance

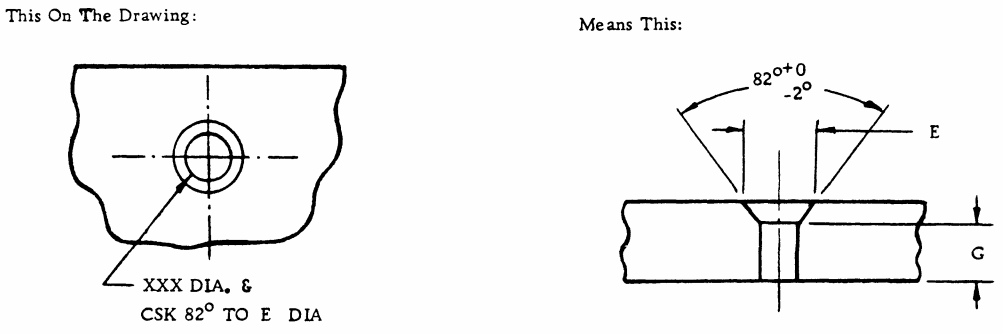


4. Countersink Tolerances

a) Countersink angle shall be within +0 and -2 degrees tolerance and the countersink face diameter "E" shall be within ± .02 inch tolerance, except for unmachined sand cast surfaces, which shall be within ± .06 inch tolerance. See Figure 3.

b) Countersinks dimensioned per "G", from a surface opposite the starting point, shall be within the same tolerance as those dimensioned per "E".

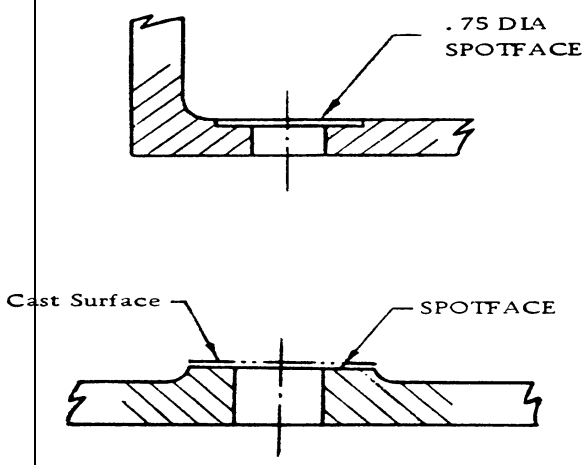
Figure 3 - Countersink Tolerances



5. Depth Of Spotface

Depth of spotface, Figure 4, shall be sufficient to clean up at least 90% of the seating surface. In all cases, the spotfacing tool shall cut below the hard surface of a casting.

Figure 4 - Spotface



6. Perpendicularity Of Spotface To Hole

The spotface surface shall be perpendicular to the axis of the hole within 1/4°.

7. Surface Texture Of Spotface

Texture of the cleaned up surface shall be 250 micro-inch AA or less.

8. Diameter Of Spotface

The diametrical tolerance of spotface up to and including one inch diameter shall be ± .03 inch, and ± .06 inch for diameters over one inch.

9. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-24

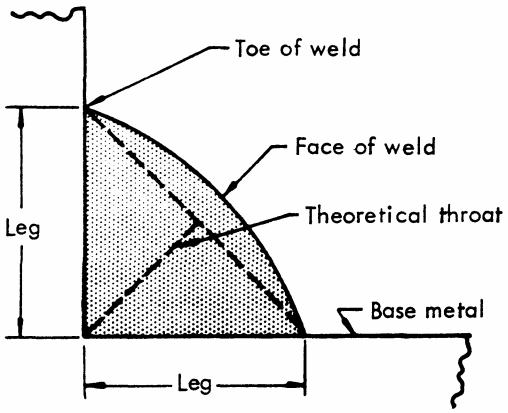
1. Scope

This standard establishes acceptable limits for welds and welded assemblies inspected by visual methods.

2. Definitions

a) Weld bead is a weld deposit resulting from a pass.
b) Leg of a fillet weld is the distance from the root of the joint to the toe of the weld.
c) Toe of a weld is the junction between the face of a weld and the base metal.
d) Face of a weld is the exposed surface of a weld, on the side from which the welding was done.
e) Throat of a fillet weld (theoretical) is the distance from the beginning of the root of the joint perpendicular to the hypotenuse of the largest right triangle that can be inscribed within the fillet weld cross-section.
f) Parent or base metal is the metal to be welded.

Figure 1 - Terms Applicable to Fillet Welds



3. Dimensional Defects And Conditions

3.1 Contour Of Butt Welds

Contour of butt welds of any type, with or without edge preparation, shall be flush or convex unless otherwise indicated on the drawing. If a ground-flush weld is allied for on the drawing, convexity before grinding need be only enough to ensure that the groove is full. Repairs shall be made by addition of weld metal as required.

3.2 Contour Of Fillet Welds

a) **Leg Undersize**
Permissible leg undersize, in one or both legs, shall be within Table 2 limits.

Figure 2 - Undersize in Weld Profile

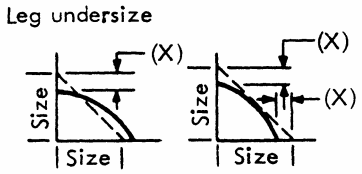


Table 2
Undersize Fillet Weld Leg Limits

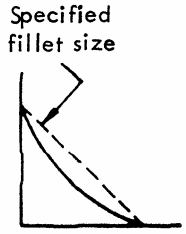
Weld Size	*Undersize - (X)
0 thru .24	0
Over .24, thru .44	.03 max.
Over .44	.06 max.

* Individual sections of undersize are permissible provided that the cumulative length of undersize does not exceed 20 percent of the full length of the weld and that individual sections of underside do not exceed a length equal to 12 times the thickness of the thinner member.

b) **Concavity**

The throat size of a concave fillet weld must not be less than the minimum theoretical throat for the weld size specified.

Figure 3 - Unacceptable Concave Fillet Weld



c) **Intermittent Fillet Welds**

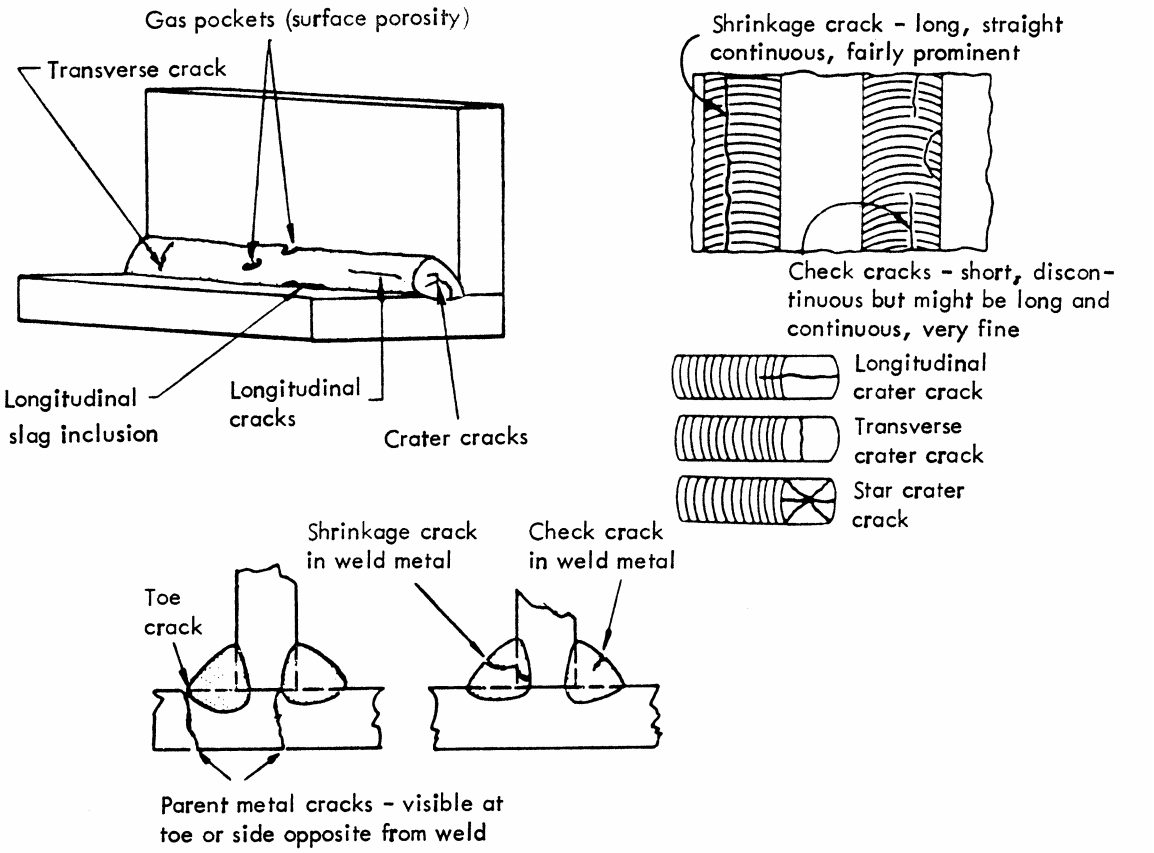
Intermittent fillet welds shall permit a plus tolerance on length of the welds and a minus tolerance on length of space between welds, unless otherwise stated on the drawing.

3.3 Visual Inspection Requirements

The welder shall visually inspect all welds, without exception. A weld is acceptable if visual inspection shows:

a) No surface cracks. Note: Cracks or blowholes that appear on the surface of any pass shall be removed before depositing the next covering pass. The procedure and technique shall be such that undercutting of base metal and adjacent passes is minimized.

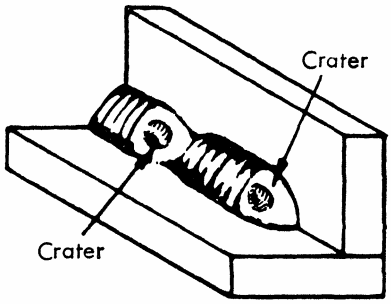
Figure 4 - Visually Detectable Cracks



b) No visible lack of fusion between welds and base metal.

c) No craters.

Figure 5 - Crater in Weld



d) The frequency of visible porosity in fillet welds does not exceed one in each 4 inches of length and the maximum diameter does not exceed 3/32".

e) No visible porosity in groove welds.

f) Undercut is not more than 0.01" deep when its direction is transverse to the primary stress in the part that is undercut and that undercut is not more than 1/32" deep when its direction is parallel to the primary stress in the part that is undercut.

g) Weld profiles are in accordance with W59 Figure 5-2 items a, b or c. Weld profiles with insufficient throat, excessive convexity, overlap, insufficient leg or inadequate penetration as in W59 Figure 5-2 items C or E are unacceptable.

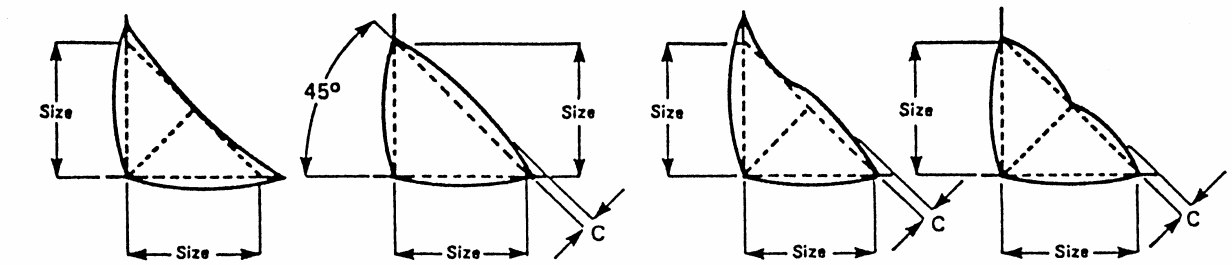
Fillet welds (1/4” size and over) in any continuous weld shall be permitted to under-run the nominal fillet size by 1/16" without correction, provided that the undersize weld does not exceed 10% of the length of the weld and does not occur within ½” of the end of the weld.

The finishing passes of all butt and corner joints shall provide reinforcement at the center of the weld not exceeding 1/8”. The reinforcement shall be built up uniformly from the surface of the base metal to a maximum at the center of the weld. There shall be no valley or groove along the edge or in the center of the weld. The deposited metal shall be smooth and uniform in cross-section. The ends of butt joints shall be of sound metal finished smoothly and conforming to

the cross-section of the welded joint as in W59 Figure 5-2 item d. They shall be free from defects as shown for butt joints in W59 Figure 5-2 item e.

h) Welds shall be free from overlap.

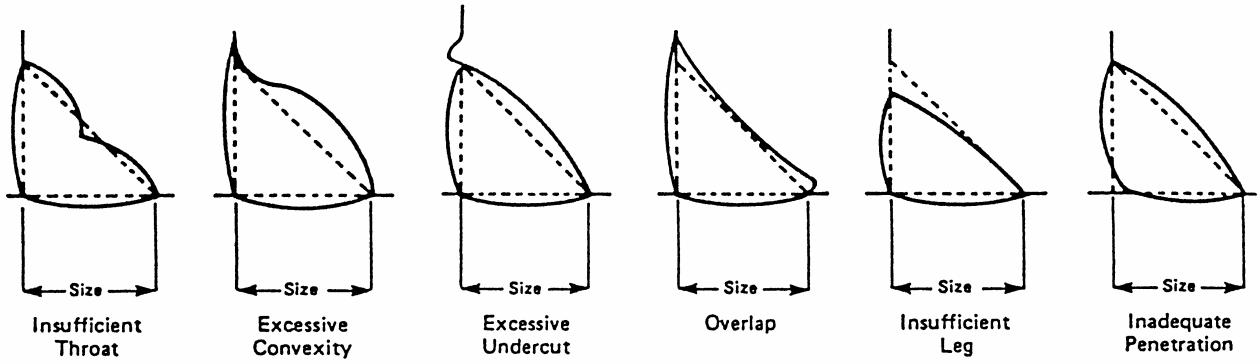
i) Weld spatter shall not exceed the criteria specified in section 3.4



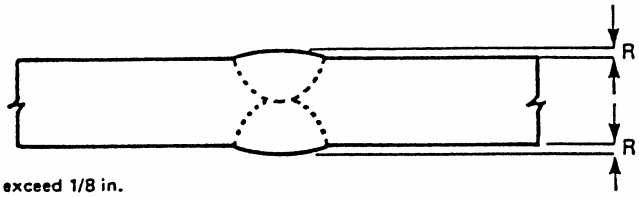
NOTE: Convexity, C, of a weld or individual surface bead shall not exceed 0.07 times the actual face width of the weld or individual bead, respectively, plus 0.06 in.

(a) — Desirable Fillet Weld Profiles

(b) — Acceptable Fillet Weld Profiles

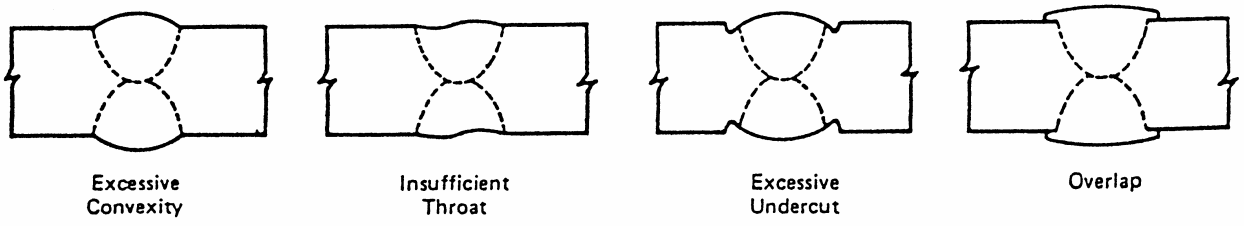


(c) — Unacceptable Fillet Weld Profiles



NOTE: Reinforcement R shall not exceed 1/8 in.

(d) — Acceptable Groove Weld Profiles in Butt Joints



(e) - Unacceptable Groove Weld Profiles in Butt Joints

Figure 5-2 (See also Clause 5.9)
ACCEPTABLE AND UNACCEPTABLE WELD PROFILES

3.4 Acceptance Criteria for Weld Spatter

For the purpose of this standard, weld spatter is defined as foreign material, which has been welded to a surface and is greater than 1/32" in diameter. Reference shop practice standard number SPS-19 for definitions of critical, major and minor surfaces.

a) **Critical Surfaces**
Critical surfaces are allowed 1 occurrence of weld spatter in every square foot of surface area, provided that the size of the spatter is less than 1/16" in diameter and that it does not pose a safety/cutting hazard.

b) **Major Surfaces**
Major surfaces are allowed 5 occurrences of weld spatter in every square foot of surface area, provided that the size of the spatter is less than 3/32" in diameter and that it does not pose a safety/cutting hazard.

c) **Minor Surfaces**
Spatter in minor areas may not exceed 10 occurrences in every square foot of surface area, provided that the spatter does not interfere with the assembly or the function of the machine and that it does not pose a safety/cutting hazard to assembly or service personnel.

4. Revisions to this Standard

Rev.	Description of Change	Changed by	Approved by	Date
Nil	• original issue	JG	MT	2003-10-24

1. Scope

This standard provides a guide to the minimum acceptance criteria for paint quality on parts manufactured by or for Delta from proprietary drawings. The standard applies only to paint applied to steel substrate.

2. Purpose

To provide a basis to accept certain paint defects.

3. Definitions

3.1 Critical Surface

A surface is considered critical when it is clearly visible on a surface commonly viewed by an elevator passenger

3.2 Major Surface

A surface is considered major when it is clearly visible during elevator maintenance (e.g., visible control panels)

3.3 Minor Surface

A surface is considered minor when it is not normally visible (e.g. outside surfaces of cab, behind or underneath manifolds, motors, wires, hoses, etc.)

4. Paint Defect Acceptance Criteria for CRITICAL SURFACES

Type of Defect	Minimum Acceptance Criteria
Paint Chip	• Paint chips, which expose bare metal, must be repaired.
Paint Run	• Paint runs may be up to ¼" long, provided that there is not more than one run in every 9 square feet of surface area and the width of the run does not exceed ¼".
Thickness	• 0.0030" minimum
Overspray	• No overspray which is visible from a distance of 5 feet (in daylight). • No overspray is allowed on cylinder rods.
Paint Appearance	• Surfaces of the same color should match in tone (Delta prefers a color difference Delta E of less than 1.0 from our master) • Paint must be smooth & glossy as per workmanship standards.
Scratches	• Any scratch, which penetrates to the base metal, must be repaired. • Shallow scratches may be up to ¼ " long provided that there is not more than one scratch in every 4 square feet of surface area.
Weathering	• Using the ASTM G-53 procedure, the paint may not show any signs of chalking, cracking, blistering, peeling, rusting or other loss of protecting properties after a minimum of 600 hrs of exposure. Loss of gloss shall not be more than 10% • after 1 year of outdoor Canadian exposure, test panels must pass an ASTM D3359-90 method B adhesion test - meeting class 2B
Adhesion	• per ASTM D3359-90 method B, paint must meet class 4B
Chip resistance	• per ASTM D3170 Gravelometer, equal to or better than 6
Flexibility	• per ASTM D-522 Conical Bend, equal to or better than ½"
Hardness	• per ASTM D3363-74 (min. H pencil hardness)
Impact Resistance	• per ASTM D-3281 direct (forward) impact, equal to or better than 30+
Solvent Resistance	• per ASTM D-4752 MEK Rub. After 50 double rubs with MEK solvent, the paint shall show no loss of gloss, no loss of color and have no effect on topcoat.
Salt Spray	• per ASTM B-117: 100 hours. Paint shall show blister rating equal to or better than 8 and loss of gloss shall not be more than 10%. After a recovery period of one hour, adhesion shall be minimum 4B.
Cure time	• after curing for 30 minutes at 200°F the paint shall pass the above MEK rub test.

5. Paint Defect Acceptance Criteria for MAJOR SURFACES

Type of Defect	Minimum Acceptance Criteria
Paint chip	• Paint chips may be up to 1/16 diameter in size, provided that there are not more than 2 chips (1/32" diameter or larger) in every 4 square feet of surface area.
Paint Run	• Paint runs may be up to ½" x ½", provided that there are not more than 2 runs in every 9 square feet of surface area.
Paint Thickness	• 0.0025" minimum
Overspray	• Overspray which is visible from a distance of 10 feet must be removed. • No overspray is allowed on cylinder rods.
Paint Appearance	• Surfaces of the same color should match in tone. • Paint must be smooth & glossy as per workmanship standards.
Scratches	• Scratches may be up to ½" long, provided there are not more than 2 scratches in every 4 square feet of surface area and the width of any one scratch is not more than 1/32".
Adhesion	• per ASTM D3359-90 method B class 2B
Hardness	• per ASTM D3363-74 (min. F pencil hardness)

6. Paint Defect Acceptance Criteria for MINOR SURFACES

Type of Defect	Minimum Acceptance Criteria
Paint Chip	• Paint chips may be up to 1/8 inch in diameter, provided that there are not more than 3 chips (1/16 inch diameter or larger) in every 4 square feet of surface area.
Paint Run	• Paint runs may be up to 3" x 3", provided that there are not more than 3 runs in every 9 square feet of surface area.
Paint Thickness	• 0.0020" minimum.
Overspray	• Overspray which is visible from a distance of 15 feet must be removed. • No overspray is allowed on cylinder rods.
Paint Appearance	• From a distance of 15 feet, surfaces of the same color should match in tone and paint must appear smooth & glossy
Scratches	• Scratches may be up to 2" long, provided there are not more than 3 scratches in every 4 square feet of surface area and the width of any one scratch are not more than 1/32".
Adhesion	• per ASTM D3359-90 method B class 1B
Hardness	• per ASTM D3363-74 (min. HB pencil hardness)

Rev	Description of Change	Changed by	Approved by	Date
Nil	original issue	JG	MT	2003-10-24

1. Coverage

This standard establishes surface finish requirements, based on component visibility and type of manufacturing process.

2. Definitions

a) **Flaws** are scratches, digs, and holes, peaks or ridges that occur at one place or at relatively infrequent intervals on the surface, usually without consistent pattern.

b) Definitions of Critical, Major and Minor Surfaces are given in Shop Practice Standard number SPS-19.

3. Critical Surface Areas

Material Type	Max. Surface Finish (μin RMS)	Max Flaw Size* (inches)	Maximum Number of flaws per 2 ft^2
Polished Stainless Steel Sheets (see photos on page 2)	32	1/16" x 1/16"	1
Rolled sheet metal or steel plate	125	¼"X¼"X1/32 DP	1
Rolled tubing (excluding seam welded area)	125	¼"X¼"X1/32 DP	1
Flame cut surfaces	500	½"X½"X1/8" DP	5
Forging & Castings	250	¼"X¼"1/32 DP	1
Ground Surfaces	125	¼"X¼"1/32 DP	1
Machined surfaces	125	¼"X¼"X1/32 DP	1

* Note: Material flaws are only allowed if they do not interfere with the proper assembly and function of the part.

The following flaws are unacceptable when found on polished stainless steel surfaces.

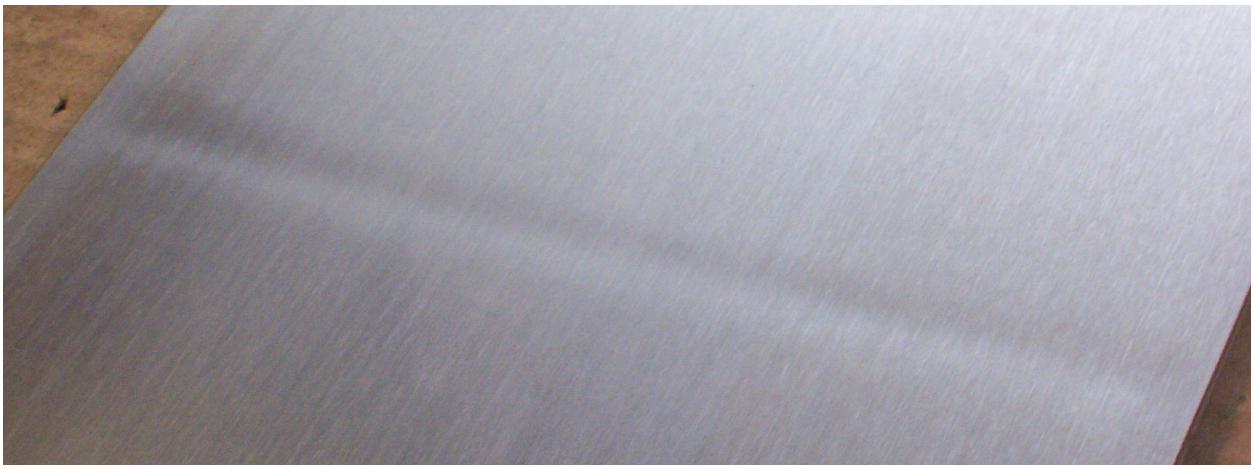


Fig. 1 - Sanding mark on polished stainless steel.

Fig. 2 - Pit Mark

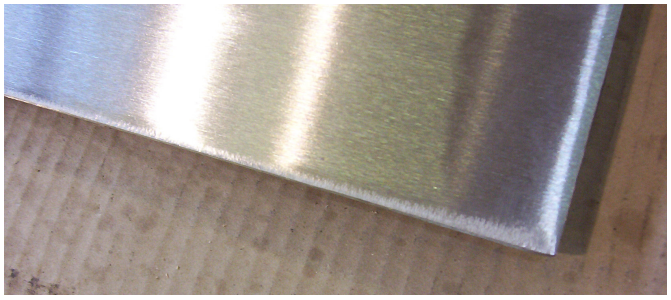


Fig. 3 – Unfinished edge

Major Surface Areas

Material Type	Max. Surface Finish (µin RMS)	Max Flaw Size* (inches)	Maximum Number of flaws per 2 ft^2
Rolled sheet metal or steel plate	250	½"X½"X1/16" DP	2
Rolled tubing (excluding seam welded area)	250	½"X½"X1/16" DP	2
Flame cut surfaces	1000	3/8"X3/8" X3/16" DP	7
Forging & Castings	500	½"X½"X1/16" DP	2
Ground Surfaces	250	½"X½" X1/16" DP	2
Machined surfaces	250	½"X½"X1/16" DP	2

4. Minor Surface Areas

Material Type	Max. Surface Finish (µin RMS)	Max Flaw Size* (inches)	Maximum Number of flaws per 2 ft^2
Rolled sheet metal or steel plate	500	1" X 1" X 3/32" DP	3
Rolled tubing (excluding seam welded area)	500	1" X 1" 3/32" DP	3
Flame cut surfaces	2000	1½X1½X¼" DP	10
Forging & Castings	1000	1"X1"X3/32" DP	3
Ground Surfaces	500	1"X1"X3/32"dp	3
Machined surfaces	500	1"X1"X3/32"dp	3

5. Revisions to this Standard

Rev	Description of Change	Changed by	Approved by	Date
Nil	• Original issue	JG	MT	2003-10-24

1. Purpose

To assist the Purchasing and Quality Departments in assessing quality assurance requirements for various parts.

2. Criticalness Ratings

All engineering drawings produced after May 1, 2006 contain a “Criticalness" field, in the title block. This rating is assigned by Engineering and helps the Purchasing Department to select appropriate Suppliers; it also helps the Quality Department in determining required inspection levels, for product manufactured in-house.

Criticalness Rating	Meaning	Subcontract Requirements
1	• most critical; • high safety risk and/or • high economic risks and/or • extremely tight tolerances and/or • large number of complex manufacturing processes • one or more “CRITICAL” dimensions highlighted on drawing	• Supplier must be ISO 9001 Registered. • or Delta must conduct an audit to ISO 9001 and supplier must correct all nonconformances (minor and major) before contract is awarded.
2	• highly critical; • significant safety risk and/or • significant economic risks and/or • tight tolerances and/or • significant number of complex manufacturing processes • one or more “MAJOR” dimensions highlighted on drawing	• Supplier must be ISO 9001 registered • or Delta must conduct an audit to ISO 9001 or ISO 9002 and supplier must correct all major nonconformances before contract is awarded.
3	• important; • limited safety risk and/or • limited economic risks and/or • few tight tolerances and/or • few complex manufacturing processes	• on-site assessment of Supplier’s capability or • evaluation of product samples or • approval based on past history supplying similar product or • published experience of other users or • ISO quality system registration

Criticalness Rating	Meaning	Subcontract Requirements
4	• non-critical; • minimal safety risk • minimal economic risks • no tight tolerances • no complex manufacturing processes	• no special requirements

3. Revisions to this Standard

Rev	Description of Change	Changed by	Approved by	Date
Nil	• Original issue	JG	MT	2003-10-24

1. Purpose

To help the Quality Department determine the required level of inspection for various features.

2. Definitions

2.1 Defect

Any deviation of a product feature or property that exceeds established acceptance criteria.

2.2 Characteristic

Features or properties (mechanical, physical and chemical) that comprise the functional design of a product or component and can be measured or otherwise evaluated to determine design conformance.

Characteristics will be divided into three classes according to their influence on the performance and reliability of the end product.

2.3 Critical Characteristic

A characteristic should be classified as critical when judgment and experience indicate that, if defective, it could result in a hazardous or unsafe condition for persons operating, maintaining or in the vicinity of the product.

Critical characteristics should be inspected 100% or should have a documented $\pm 5\sigma$, process capability.

2.4 Major Characteristic

A characteristic should be classified as major (if not already classified critical) when, if defective, it could cause a sudden failure resulting in serious damage to the equipment or surroundings or result in the complete lack of useful performance of the product, (sometimes referred to as a "catastrophic" failure).

Major characteristics should be subject a minimum of 25% inspection or should have a documented $\pm 3\sigma$, process capability.

2.5 Minor Characteristic

A characteristic is considered minor if, when defective, it will not materially reduce the usability of a product for its intended purpose, or a departure from established standards would have little or no significant bearing on product performance.

The inspection frequency of minor characteristics is left to the discretion of Production Supervisors.

2.6 Critical Surface

A surface is considered critical when it is clearly visible on the outside of a fully retracted machine.

2.7 Major Surface

A surface is considered major when it is clearly visible during routine machine maintenance (e.g. visible areas inside of cabinets, tape head boxes, car operating stations, etc.)

2.8 Minor Surface

A surface is considered minor when it is not normally visible (e.g. bottom of platform, behind or underneath manifolds, motors, wires, hoses, etc.)

3. Method Of Expression

Critical characteristics are identified with the word **"CRITICAL"** written below or beside the dimension.

Major characteristics are identified with the word **"MAJOR"** written below or beside the dimension.

Minor characteristics are not identified in any special manner (i.e. if the characteristic is not identified as critical or major, then it is a minor characteristic).

Critical and major surfaces are not identified unless they deviate from the definitions given in section 2. Shop practice standards SPS-15 and SPS-16 describe welding and painting criteria for these surfaces.

4. Revisions to this Standard

Rev	Description of Change	Changed by	Approved by	Date
Nil	• Original Issue	JG	MT	2003-10-24