Geometric Dimensioning and Tolerancing

The views and opinions expressed in this article are those of the author and DO NOT represent the official position of Louisiana Tech University or the State of Louisiana.

Waiver of Author's Liability

The author, Dale O. Anderson, Ph.D., provides the information in this article "as is" and makes no warranty, either express or implied, concerning its accuracy or fitness for use in any particular situation. The author's sole intent in providing this information is to enhance the educational experience of his engineering students and to stimulate their thought processes. Anyone choosing to use this information does so at their own risk. The author recommends that any design procedure or data be carefully examined by a potential user to determine if it is both accurate and appropriate for use in each potential application.

Table of Contents:

Introduction

Geometric dimensioning and tolerancing (GDT) is a uniform method of stating and interpreting engineering design requirements listed on a shop drawing. It replaces the traditional dimension with attached tolerance (1.000 ± 0.050 inch) previously used on engineering drawings. The basis of GDT is the American standard ANSI Y14.5M-1994. Adherence to standards facilitates the exchange of data within an organization and between organizations. There is less likelihood of misinterpretation.

GDT is closely based on the actual measurement techniques (metrology) used to check the tolerance. Therefore, all tolerances are related to specific properties of a particular feature.

A dimensional tolerance is, or should be, related to the variance of the measurement, which includes variations in locating the part on the machine tool, positioning variations in the machining process, variations in the tool, and metrology variations in the inspection process. The most common practice in engineering design is to assign tolerances by "rule of thumb" based on experience with a particular machining process. However, the recommended practice is to assign "six sigma" tolerances (plus or minus three standard deviations) to basic dimensions based on the expected variance in the machining process [Bowker, 1972, pp. 86-87]. If statistical process control techniques are being used on the machining process, sufficient historical data should be available to produce a good estimate of the variance of the process. Six sigma tolerances lead to three failures (out of tolerance parts) in every thousand parts on average.

Table 1. Confidence (Reliability) Levels for Dimensional Tolerances			
Tolerance/ Standard Deviation	**Symmetric**	**Asymmetric**	**Comments**
0	0%	0%	
0.675	50%	25%	
1.65	90%	45%	
1.96	95%	47.5%	Usual experimental confidence level -- 5 failures per hundred
2.58	99%	49.5%	
3.00	99.73%	49.87%	"six sigma" tolerances -- 3 failures per thousand
3.29	99.9%	49.95%	one failure per thousand
3.89	99.99%	49.995%	
4.42	99.999%	49.9995%	
4.89	99.9999%	49.99995%	one failure per million
NOTE: the total confidence of an asymmetric tolerance is the sum of the upper and lower confidences.			

...go back to the table of contents...

Dimensioning

The basic mechanics of adding dimensions to a drawing have not changed with the advent of GDT. What has changed is the way the dimension value is specified. In GDT, a dimension value is enclosed in a box WITHOUT the tolerances attached. Tolerances are added in feature control frames.

Table 2. GDT Dimensioning Symbology	
Symbol	**Description**
$\boxed{1.250}$	This is a basic (exact) linear dimension. Notice the box around the value. Tolerances are added to a basic dimension by a feature control frame.
$\boxed{Ø\ 0.375}$	This is a basic (exact) diameter. The symbol Ø denotes diameter. Tolerances are added to a basic diameter by a feature control frame.
$\boxed{R\ 0.375}$	This is a basic (exact) radius. The symbol R denotes radius. Tolerances are added to a basic radius by a feature control frame.

1.250 ±0.005	This is an OLD STYLE (nonGDT) linear dimension with symmetric bilateral tolerances applied. The first number is the basic dimension and the second number is the tolerance applied to the basic dimension. The tolerance specification allows the dimension to vary from 1.245 to 1.255.
1.250 +0.008 -0.002	This is an OLD STYLE (nonGDT) linear dimension with asymmetric bilateral tolerances applied. The first number is the basic dimension, the second number is the positive tolerance applied to the basic dimension, and the third number is the negative tolerance. The tolerance specification allows the dimension to vary from 1.248 to 1.258.
1.250 +0.010 -0.000	This is an OLD STYLE (nonGDT) linear dimension with a unilateral tolerance applied. The first number is the basic dimension, the second number is the positive tolerance applied to the basic dimension, and the third number is the negative tolerance. The tolerance specification allows the dimension to vary from 1.250 to 1.260.

...go back to the table of contents...

Datums

A datum is a reference surface, plane, line or point used to facilitate the definition of features on a part. A datum is normally a locating surface to be used for measurement. For flat features, the locating surface should be a flat, machined surface on one of the outside boundaries of the part. For round features, the locating surface should be an exterior cylindrical surface.

Normally for a part with flat surfaces, three mutually perpendicular reference planes are defined for the part (see figure 1). The intersections of those reference planes form the traditional three-axis coordinate system -- a rectangular (Cartesian) coordinate system. The primary datum plane or locating surface is defined by three reference points (because three points define a plane). The secondary datum plane or locating surface is defined by two additional points (because only two points are necessary to define a plane perpendicular to the primary locating plane). The tertiary datum plane or locating surface is defined by a single additional point (because only one point is required to define a plane perpendicular to both the primary and secondary planes).

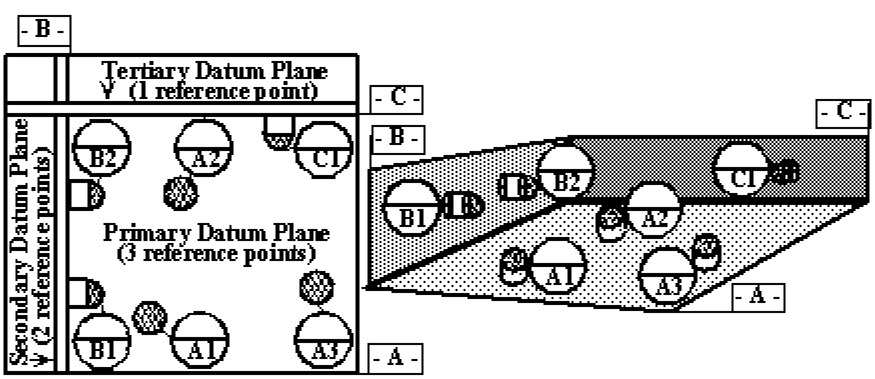

Figure 1. A Cartesian Coordinate System Defined by Reference Planes

Table 3. GDT Datum Symbology	
Symbol	**Description**
- A -	A datum identification symbol -- a reference surface, line, point or feature labeled A. The referenced feature must actually exist on the part and be accessible for measurement.
A-B	A datum line or axis defined between points A and B.
A1	A datum target symbol for point #1 on datum A.

[...go back to the table of contents...](#)

Feature Control Frame - Tolerances

A feature control frame (see figure 2 below) is a standard way to specify tolerances for specific features on a part.

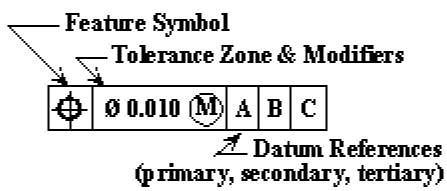

Figure 2. A Generic Feature Control Frame

The symbols listed in the following table MUST be attached to a feature symbol to become tolerances.

Table 4. GDT Feature Control Frame Symbology	
Symbol	**Description**
0.010	This is a symmetric feature tolerance that denotes the size of the tolerance zone about the basic dimension (±0.010). The tolerance zone for a single datum is a line segment. The tolerance zone for two perpendicular datums is a circle. The tolerance zone for three perpendicular datums is a sphere. The units must be specified on the drawing.
+0.008 -0.002	This is an asymmetric feature tolerance that denotes the size of the tolerance zone about the basic dimension (+0.008, -0.002). The tolerance zone for a single datum is a line segment. The tolerance zone for two perpendicular datums is a rectangular box. The tolerance zone for three perpendicular datums is a right rectangular prism. The units must be specified on the drawing.
Ø 0.010	This is a symmetric diameter tolerance (±0.010). The units must be specified on the drawing.
R 0.010	This is a symmetric radius tolerance (±0.010). The units must be specified on the drawing.
M	A tolerance modifier that denotes "most material condition." This modifier indicates a feature that contains the maximum amount of material (i.e. minimum hole diameter & maximum shaft diameter).

Ⓛ	A tolerance modifier that denotes "least material condition." This modifier indicates a feature that contains the minimum amount of material (i.e. maximum hole diameter & minimum shaft diameter).
Ⓢ	A tolerance modifier that denotes "regardless of feature size." The tolerance is applied to the feature dimension only and does not include any size information. This is the preferred modifier for statistical process control.
Ⓟ	A tolerance modifier that denotes "projected tolerance."

...go back to the table of contents...

Position Tolerancing

Table 5. Position Tolerance Symbology

Symbol	Description	Metrology
⌖	This is a position target to which a linear measurement tolerance zone is applied. For one datum, the tolerance zone is a line segment. For two perpendicular datum lines or surfaces, the tolerance zone is a circle (symmetric tolerance) or an ellipse (asymmetric tolerance). For three perpendicular datum surfaces, the tolerance zone is a sphere (symmetric tolerance) or an oblate spheroid (asymmetric tolerance)	This measurement may be made by hand or on a linear coordinate measurement system.

Table 6. Typical "Best" (Minimum) Tolerances for Various Positioning Operations (Rao, 1992, p. 191)

	Tolerance		Relative
Machining Operation	**inches**	**mm**	**Cost**
Flame Cutting (by hand)	±1/8 (±0.125)	±3.20	0.25
Drilling (by hand on a drill press)	±1/16 (±0.063)	±1.60	0.50
Standard Milling & Turning	±0.005	±0.13	1.00
Fine Milling & Turning	±0.001	±0.03	2.00
Precision Milling & Turning	±0.0005	±0.013	3.5
Spot/Fusion Welding (by hand)	±0.010	±0.25	0.25

...go back to the table of contents...

Linear Feature Tolerancing

Linear features are the easiest to machine and to measure.

Table 7. Linear Feature Tolerance Symbology

Symbol	Description	Metrology
⏤	A linearity or straightness tolerance. The tolerance zone is a rectangular area centered along the specified line and bounded by two parallel lines.	This is a 1-D measurement in which a dial indicator may be moved along the line in question. The tolerance refers to the total indicator movement.
⏥	A surface flatness tolerance. The tolerance zone is a rectangular prismatic volume aligned along the specified surface and bounded by two planes.	This is a 2-D measurement that may be checked by locating the part on the surface in question and then moving a dial indicator over the entire surface. The tolerance refers to the total indicator movement.
//	A parallelism tolerance with respect to a datum line or surface. The tolerance zone is a rectangular area or volume aligned along the line or surface feature and bounded by two parallel lines or planes.	This tolerance can be measured by locating the part on datum surface and then moving a dial indicator along the line or across the surface. The tolerance refers to the total indicator movement.
⊥	A perpendicularity tolerance with respect to a datum line or surface. The tolerance zone is a rectangular area or volume aligned along the perpendicular line or surface and bounded by two parallel lines or planes.	This tolerance can be measured by locating the part on the datum line or surface and then moving a dial indicator along the line or across the surface. The tolerance refers to the total indicator movement.
∠ expensive!	An angularity tolerance with respect to a datum line or surface. The tolerance zone is a rectangular area or volume aligned along the angled line or surface and bounded by two parallel lines or planes. Note that this approach is substantially different from the old method of tolerancing angularity since it uses an indicator reading perpendicular to the angled line or surface and NOT an angular tolerance.	This tolerance can be measured by locating the part on the datum surface and then moving a dial indicator along the angled line or across the angled surface. NOTE a sine table is useful for making this measurement by hand. THIS IS A DIFFICULT TOLERANCE TO CHECK.

Table 8. Typical "Best" (Minimum) Tolerances for Various Linear/Surface Manufacturing Operations (Rao, 1992, p. 191)

	Tolerance		Relative
Machining Operation	**inches**	**mm**	**Cost**
Flame Cutting	±1/8 (±0.125)	±3.20	0.25
Sawing	±0.050	±1.30	0.35
Rough Machining	±1/32 (±0.030)	±0.80	0.50
Standard Milling & Turning	±0.005	±0.13	1.00

Fine Milling & Turning, Rough Grinding	±0.001	±0.03	2.00
Precision Milling & Turning, Ordinary Grinding	±0.0005	±0.013	3.50
Fine Grinding, Shaving, Honing	±0.0002	±0.005	5.50
Very Fine Grinding, Fine Shaving, Honing & Lapping	±0.0001	±0.0025	10.00
Honing, Lapping & Polishing	±0.00005	±0.0013	15.00

...go back to the table of contents...

Circular/Cylindrical Feature Tolerancing

Table 9. Circular/Cylindrical Feature Tolerance Symbology		
Symbol	**Description**	**Metrology**
○ expensive!	A roundness (circularity) tolerance. The tolerance zone is an annular area concentric with the specified circle and bounded by two concentric circles.	The roundness of a circle must be checked without reference to the central axis of the circle. This may be done manually by supporting the called out surface in a vee-block (NOT a 45° vee), positioning a dial indicator perpendicular to the central axis of the circle, and then rotating the part 360 degrees. This measurement is probably best done using an electronic profilometer. The tolerance refers to the total indicator movement during the measurement. THIS IS A DIFFICULT TOLERANCE TO CHECK.
⌭ expensive!	A cylindricity tolerance. The tolerance zone is an annular volume concentric with the called out cylindrical surface and bounded by two concentric cylindrical surfaces.	The cylindricity must be checked without reference to the central axis of the cylinder. This may be done manually by supporting the called out surface in a vee-block (NOT a 45° vee), positioning a dial indicator perpendicular to the central axis of the circle, and then running the indicator over the entire surface. This measurement is probably best done using an electronic profilometer. The tolerance refers to the total indicator movement during the measurement. THIS IS A DIFFICULT TOLERANCE TO CHECK.
◎ very expensive!	A concentricity tolerance. This tolerance refers to comparing the location of the central axes (center lines) of two circles without regard to the roundness of the circles. The perpendicular distance between the two axes must be less than the specified tolerance. By definition, the	The central axis of each circle must be calculated. This measurement is probably best done using an electronic profilometer. The tolerance refers to the total indicator movement during the measurement. THIS IS A VERY DIFFICULT TOLERANCE TO CHECK.

	central axis lines must be parallel.	
↗ expensive!	A circular runout tolerance. The tolerance zone is an annular surface of revolution about the center axis of the circle and perpendicular to the feature surface.	This tolerance is checked by locating the part on a rotating fixture, positioning a dial indicator perpendicular to the called out feature surface on the specified circle, and then rotating the part 360 degrees. The tolerance refers to the total indicator movement during the measurement. THIS IS A DIFFICULT TOLERANCE TO CHECK.
⌰ very expensive!	A total runout tolerance. The tolerance zone is an annular cylindrical volume of revolution about the center axis of the circle and concentric with the feature surface.	This tolerance is checked by locating the part on a rotating fixture, positioning a dial indicator perpendicular to the called out feature surface, and then running the indicator over the entire called out surface. The tolerance refers to the total indicator movement during the measurement. THIS IS A VERY DIFFICULT TOLERANCE TO CHECK.

Table 10. Typical "Best" (Minimum) Tolerances for Various Circular Manufacturing Operations (Rao, 1992, p. 191)

	Tolerance		**Relative**
Machining Operation	**inches**	**mm**	**Cost**
Drilling a Hole (drill press)	+0.002 -0.000	±0.05	0.50
Milling a Hole	±0.001	±0.03	1.00
Blanking	±0.001	±0.03	0.20
Standard Turning (Lathe)	±0.005	±0.13	1.00
Fine Turning (Lathe)	±0.001	±0.03	2.00
Hot Extrusion	±0.005	±0.13	0.20
Cold Drawing	±0.002	±0.05	0.40
Forging	±0.030/inch	±0.03/mm	0.50
Sand Casting	±0.003/inch	±0.003/mm	0.50
Die Casting	±0.002/inch	±0.002/mm	0.70

...go back to the table of contents...

Profile Feature Tolerancing

Table 11. Profile Feature Tolerance Symbology		
Symbol	**Description**	**Metrology**
⌒ expensive!	A curve profile tolerance. The tolerance zone is an area centered about the specified curve and bounded by parallel curves of the same shape as the basic profile.	This tolerance can be measured by locating the part on a datum surface and then moving a dial indicator along the profile line -- a 1-D movement. The dial indicator must move along a reference curve. The tolerance refers to the total indicator movement. This measurement is probably best done using an electronic profilometer. THIS IS A DIFFICULT TOLERANCE TO CHECK.
⌓ very expensive!	A surface profile tolerance. The tolerance zone is a volume centered along the specified surface and bounded by parallel surfaces of the same shape as the basic profile.	This tolerance can be measured by locating the part on a datum surface and then moving a dial indicator across the profile surface. The dial indicator must move along a reference surface. The tolerance refers to the total indicator movement. This measurement is probably best done using an electronic profilometer. THIS IS A VERY DIFFICULT TOLERANCE TO CHECK.

Table 12. Typical "Best" (Minimum) Tolerances for Various Profile Manufacturing Operations (Rao, 1992, p. 191)			
	Tolerance		**Relative**
Machining Operation	**inches**	**mm**	**Cost**
Standard Milling	±0.005	±0.13	1.00
Fine Milling	±0.001	±0.03	2.00
Hot Extrusion	±0.005	±0.13	0.20
Cold Drawing	±0.002	±0.05	0.40
Forging	±0.030/inch	±0.03/mm	0.50
Sand Casting	±0.003/inch	±0.003/mm	0.50
Die Casting	±0.002/inch	±0.002/mm	0.70

...go back to the table of contents...

Misc. Feature Tolerancing

Table 13. Misc. Feature Tolerance Symbology

Symbol	Description	Metrology
↧	A depth tolerance. The tolerance zone is a disk at the bottom of the hole.	This tolerance can be measured by locating on the surface where the hole starts, dropping the probe of a dial indicator down into the hole, and then moving the probe over the bottom surface of the hole. The tolerance refers to the total indicator movement.

Table 14. Typical "Best" (Minimum) Tolerances for Misc. Manufacturing Operations (Rao, 1992, p. 191)

	Tolerance		Relative
Machining Operation	**inches**	**mm**	**Cost**
Drilling	+0.002 -0.000	±0.05	0.50
Milling or Boring (Lathe) a Hole	±0.005	±0.13	1.00
Forging	±0.030/inch	±0.03/mm	0.50
Sand Casting	±0.003/inch	±0.003/mm	0.50
Die Casting	±0.002/inch	±0.002/mm	0.70

...go back to the table of contents...

Tolerance Stacking

It is often necessary to assemble two or more parts together. Each part will have its own tolerances, but what is the tolerance of the assembly? This is the question addressed by tolerance stacking. The basic dimension of the assembly is simply the sum of the appropriate basic dimensions of each part. The tolerance that should be applied to the assembly is the square root of the sum of the squares of the individual part tolerances. Therefore, it is readily seen that the tolerance range for the assembly will be greater than the largest tolerance range of any of the component parts.

If an assembly is to be bolted or riveted together, then the basic dimension and tolerance of the fastener MUST be compatible with the assembly. The shortest expected length of the fastener must be greater than the greatest expected height of the assembly being fastened.

If a shaft is to run in a machined journal, then the shaft diameter MUST always be less than the diameter of the hole it fits into. The largest expected diameter of the shaft must be less than the smallest expected diameter of the hole.

If a pin is to be placed in a hole using an interference fit, then the diameter of the pin MUST always be greater than the diameter of the hole. The smallest expected diameter of the pin must be greater than the largest expected diameter of the hole.

...go back to the table of contents...

General Guidelines

ALWAYS put a note on each shop drawing stating the unit of linear measure (i.e. inches or cm).

Try to reference features with respect to a minimum number of common machined reference surfaces on the part.

Total distance from the datum to the feature is preferred to incremental distances from feature to feature.

"Tight" tolerances tend to increase the manufacturing cost and time, while tolerances that are greater than customary for a particular machining operation tend to confuse machine operators and increase part variability.

...go back to the table of contents...

Glossary

Allowance -- the minimum clearance (or maximum interference) between two mating parts.
Base-line Dimensioning -- the use of a common set of datums for dimensions. (Preferred!)
Basic Dimension --The exact theoretical (nominal) dimension without tolerances applied. This is the dimension portion of a traditional dimension entry.
Bilateral Tolerance -- A tolerance specification that includes + and - values (two sided). The values DO NOT need to be identical. Examples: ±0.005, +0.005 -0.001.
Bolt Circle -- a circular center line on which two or more bolt hole centers are located.
Bore -- enlarge a hole diameter using a boring bar. Can be held to close tolerances.
Boss -- a raised cylindrical projection (pad) which provides extra material around a hole in a casting or forging.
Broach -- produce a noncylindrical hole from a cylindrical starter hole by pulling a linear cutting tool through the hole in a reciprocating motion.
Burnish -- to polish a metal surface by rolling or sliding a tool over the surface under pressure.
Callout -- A specific note on a blueprint stating dimensions, tolerances, geometric controls, or feature specifications.
Center Plane -- A reference plane that passes through a center line.
Chain Dimensioning -- Successive dimensions that run from one feature to another rather than originating at a common datum. Tolerances accumulate! (Usually NOT desirable!)

Chamfer -- cutting off the exterior or interior edge between two perpendicular surfaces at an angle. A chamfer dimension is an offset distance, not the length of the chamfer surface.
Circular Runout -- a measure of out-of-roundness of a circular feature.
Clearance -- the maximum intentional difference between mating parts.
Clearance Hole -- A hole just slightly larger in diameter than the bolt that goes through it.
Collar -- A projecting ring around a shaft.
Concentric Circles -- circles of different radii with a common center point.
Counterbore -- a cylindrical hole bored concentrically in a hole of smaller diameter. The counterbore has a flat bottom and is usually used to submerge a bolt head into a surface.
Countersink -- A conical hole bored concentrically in a hole of smaller diameter. The countersink is usually used to submerge a conical screw head into a surface.
Datum -- A point, line, plane, or reference surface used as a reference for the position and orientation of other features.
Datum Feature -- Any feature on a part used to locate other features.
Datum Identification Symbol -- A rectangular box associated with a datum with a dash letter dash identifier in it (i.e. $\boxed{\text{- A -}}$).
Datum Line -- A reference line on a part used to locate other features.
Datum Plane -- A plane defined by features on a part used to locate other features.
Datum Surface -- A surface, usually flat, used to locate other features on the part.
Datum Target Symbol -- A circle with a horizontal bisecting line dr
Fillet -- an interior rounded surface that connects two intersecting surfaces. A fillet eliminates a sharp intersection. Usually used in castings or forgings.
Fin -- A thin projecting edge on cast or molded parts.
Finish Marks -- symbols on the edge view of surfaces to be machines. Usually used in drawings of castings or forgings.
Flange -- A rim or collar on an object to hold it in place.
Form Tolerance -- a tolerance that controls the form of a geometric shape.
Full Indicator Movement (FIM) -- The fill movement range of a measurement instrument, such as a dial indicator, over the line, curve or surface to be measured. It means the same as TIM, TIR or FIR.
Full Indicator Reading (FIR) -- The fill movement range of a measurement instrument, such as a dial indicator, over the line, curve or surface to be measured. It means the same as TIM, TIR or FIM.
Gauge -- the thickness of sheet metal or the diameter of wire by a number rather than by dimension.
Geometric Tolerance -- tolerances with emphasis on the actual function or relationship of part features for interchangeable parts. It includes form and locational tolerancing.
Gusset -- an additional piece of material used to reinforce a frame, usually at the corners.
Honing --
Hub -- the central part of a wheel, gear or pulley.
Implied Datum -- An unspecified datum (not recommended). A feature control frame with no datum specified.
Kerf -- the ragged edge left by a saw or cutting torch.
Key -- a metal bar or wedge used to connect a hub to a shaft. A key requires a keyway in the hub and a keyseat in the shaft.
Keyseat -- a groove in a shaft to hold a key.
Keyway -- a grove in a hub to hold a key.
Knurl -- a surface roughened by a rolling tool to improve hand grip.
Lapping -- a surface finishing method consisting of rubbing the surface with a fine abrasive.
Least Material Condition (LMC) -- a feature that contains the minimum amount of material (i.e.

maximum hole diameter & minimum shaft diameter).
Limits -- the maximum and minimum permissible dimensions calculated from the basic dimension and the tolerances.
Maximum Material Condition (MMC) -- a feature that contains the maximum amount of material (i.e. minimum hole diameter & maximum shaft diameter).
Neck -- A groove cut in a shaft at a change in diameter to allow the end surface of the larger diameter part to seat flush against a mating surface.
Nominal Size -- a size specification used for identification purposes.
Orthographic Projection -- a 3-D, corner view of a part without perspective.
Pad -- A raised projection on a casting used to provide extra metal around a hole. It is like a Boss, but it is not necessarily cylindrical.
Parallel Lines -- lines that never intersect.
Pattern -- a particular arrangement of features on a part.
Perpendicular Lines -- lines that intersect at right angles.
Polishing -- a surface finishing method consisting of rubbing the surface with a very fine abrasive.
Positional Tolerancing -- the allowable variation on the position of a feature.
Profile View -- the view that most clearly shows the shape of the part.
Projected Tolerance Zone -- A tolerance that is projected away from the part in a feature. This is used to insure that mating parts fit together. The symbol is a circle with the letter P inside.
Radius -- the distance from the center of a circular arc to the circumference. (= 2 x diameter)
Ream -- to enlarge a hole slightly using a rotating tool.
Reference Dimension -- an informational dimension that does not have tolerances applied.
Regardless of Feature Size (RFS) -- a position or form tolerance that must be met regardless of the size tolerance of a feature. This is the preferred modifier for statistical process control, since it is the easiest to check and verify.
Relief -- a surface set slightly lower than another surface usually for clearance.
Rib -- A thin, flat feature extending beyond a surface used as a support or brace for the surface.
Round -- a rounded external edge between two surfaces. Like a fillet, but on an external edge.
Section -- a cross-sectional view.
Shim -- a thin piece of material placed between two mating parts to adjust the fit or positioning.
Shoulder -- a step change in the diameter of a shaft. The shoulder is perpendicular to the shaft axis.
Spline -- a raised area on a shaft designed to mate with a grove in a hub.
Spot Face -- to machine a smooth round spot on a rough surface, usually around a hole, to give a good seat for a bolt head or nut.
Stud -- a cylindrical piece of metal threaded on both ends.
Surface Roughness -- a measure of roughness of a surface (microinches or micrometers).
Tabular Dimensioning -- letters are substituted for dimensions and a table of dimensions and tolerances is keyed to the letters. This is usually used when several similar parts are specified on the same drawing.
Tap -- a toll for cutting standard threads in a hole.
Taper -- a conical shape given to a shaft or hole usually used for removable mating surfaces.
Tolerance -- the allowable variation in a dimension.
Total Indicator Movement (TIM) -- The fill movement range of a measurement instrument, such as a dial indicator, over the line, curve or surface to be measured. It means the same as TIR, FIM, or FIR.
Total Indicator Reading (TIR) -- The fill movement range of a measurement instrument, such as a dial indicator, over the line, curve or surface to be measured. It means the same as TIM, FIM, or FIR.
Total Runout -- a measure of out-of-roundness and axis misalignment of a cylindrical feature.
Undercut -- a grove cut on the inside of another cut.
Unilateral Tolerance -- A tolerance specification that includes only nonzero + or - values (one sided). For

example +0.005 -0.000 or +0.000 -0.001.
Web -- a section of thin surface connecting other features of a part

. ...go back to the table of contents...

References

Anonymous, 1994, **Dimensioning and Tolerancing**, ANSI Y14.5M-1994, ASME International, New York.

Bowker, A. H. and G. J. Lieberman, 1972, **Engineering Statistics**, 2nd. ed., Prentice-Hall, Elglewood Cliffs, NJ. (ISBN 0-13-279455-1, LOC TA153.B67)

Hammer, Warren, 1989, **Blueprint Reading Basics**, Industrial Press, New York. (ISBN 0-8311-1186-0, LOC T379.H34)

Meadows, James D., 1995, **Geometric Dimensioning and Tolerancing**, Marcel Dekker, New York. (ISBN 0-8247-9309-9, LOC TS172.M43)

Puncochar, Daniel E., 1990, **Interpretation of Geometric Dimensioning and Tolerancing**, Industrial Press, Inc., New York. (ISBN 0-8311-3010-5, LOC T357.P96)

Rao, S. S., 1992, **Reliability Based Design**, McGraw-Hill, New York (ISBN 0-07-051192-6, LOC TA169.R37)

Wilson, Bruce A., 1995, **Dimensioning and Tolerancing Handbook**, Genium Publishing, Schenectady, NY. (ISBN 0-931690-80-3)

...go back to the table of contents...

Links to Related Articles: Engineering Design, PDS, PRP,

...go back to the table of contents...

(This article was last updated 16 February 2000 by Dale O. Anderson. Ph.D.)