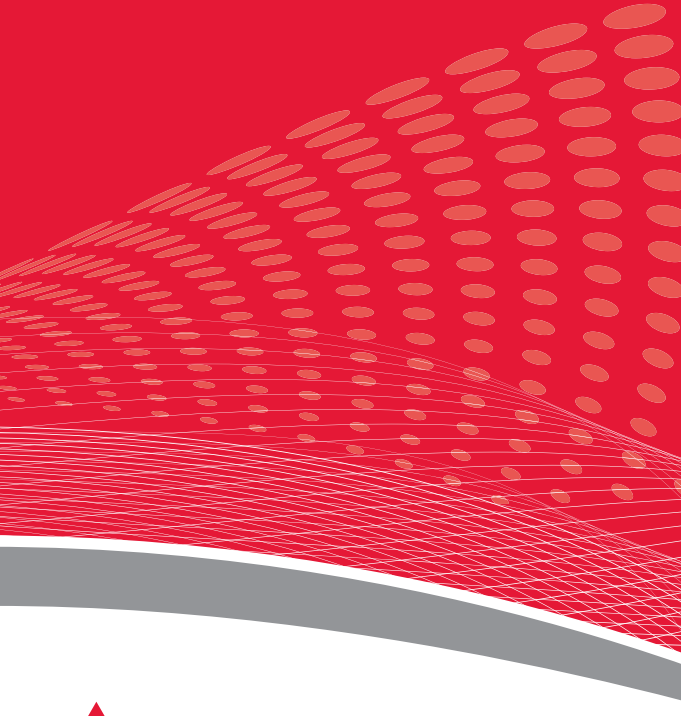
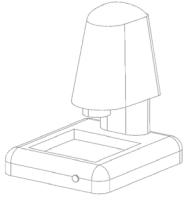
Geometric Dimensioning and Tolerancing (GD&T)

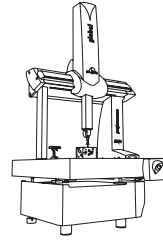
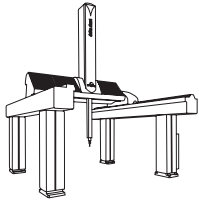


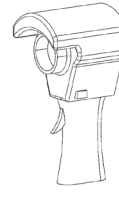
HI-TECH METROLOGY

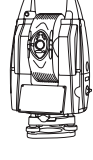
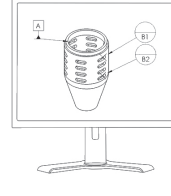
HI-TECH METROLOGY

After more than 20 years Hi-Tech Metrology continues to deliver world-class metrology and reverse engineering solutions for Australian industry, solutions that are designed to deliver enhanced customer outcomes in terms of productivity, quality and innovation. An extensive range of metrology and scanning equipment, combined with a complete array of after sales support services, enables us to tailor these total solutions incorporating hardware and software for virtually any measurement or reverse engineering task.





INDEX

Angularity 11
Circular Runout 18
Coaxiality 16
Concentricity 17
Coordinate Measurement Conversion 20-21
Cylindricity 9
Datum Planes 6
Datums 5
Datum Targets 7
Decimal Equivalents 24
Establishing Datum Planes 7
Feature Control 5
Flatness 8
Form Tolerance 8
General Rules 2-3
Geometric Characteristic Symbols 1
Line Profile 9
Location Tolerance 15
Modifiers 4
Modifying Symbols 1
Orientation Tolerance 11
Parallelism 13
Perpendicularity 11
Position 15
Profile Tolerances 9
Roundness 9
Runout 18
SI Conversions 23
SI Units 22
Straightness 8
Surface Profile 10
Symmetry 16
Three Plane Concept 6
Total Runout 19

GEOMETRIC CHARACTERISTIC SYMBOLS

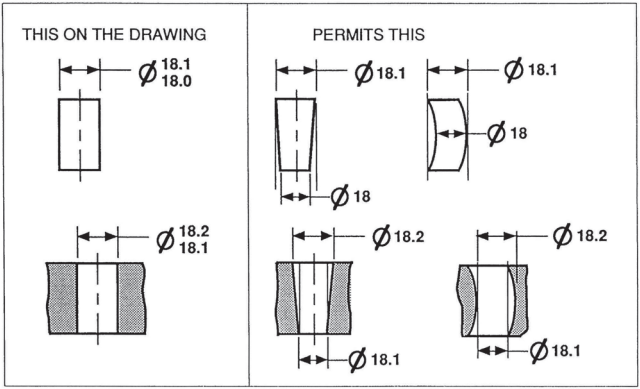
	TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL (ANSI)	MMIV
FOR INDIVIDUAL FEATURES	FORM	STRAIGHTNESS		
		FLATNESS		
		CIRCULARITY (ROUNDNESS)		
		CYLINDRICITY		
FOR INDIV. OR RELATED FEATURES	PROFILE	PROFILE OF A LINE		
		PROFILE OF A SURFACE		
FOR RELATED FEATURES	ORIENTATION	ANGULARITY		
		PERPENDICULARITY		
		PARALLELISM		
	LOCATION	POSITION		
		CONCENTRICITY		
	RUNOUT	CIRCULAR RUNOUT		
		TOTAL RUNOUT		

MODIFYING SYMBOLS

TERM	SYMBOL
AT MAXIMUM MATERIAL CONDITION	Ⓜ
REGARDLESS OF FEATURE SIZE	Ⓢ
AT LEAST MATERIAL CONDITION	Ⓛ
PROJECTED TOLERANCE ZONE	Ⓟ
DIAMETER	∅
SPHERICAL DIAMETER	S∅
RADIUS	R
SPHERICAL RADIUS	SR
REFERENCE	( )
ARC LENGTH	⌒

GENERAL RULES

1. The form of an individual feature is controlled by its limits of size.



The Perfect Form Rule does not apply to the following:

a) Stock such as bars, sheets, tubing, structural shapes, and other items produced to established industry or Government standards that prescribe limits for straightness, flatness, and other geometric characteristics. Unless geometric tolerances are specified on the drawing of a part made from these items, standards for these items govern the surfaces that remain in the "as-furnished" condition on the finished part.

b) Parts subject to free state variation in the unrestrained condition.

Perfect Form[1] at MMC not required:

Where it is desired to permit a surface or surfaces of a feature to exceed the boundary of perfect form at MMC, a note such as PERFECT FORM AT MMC NOT REQD is specified, exempting the pertinent size dimension.

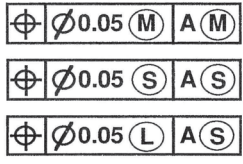
Relationship between individual features:

The limits of size do not control the orientation or location relationship between individual features. Features shown perpendicular, coaxial, or symmetrical to each other must be controlled for location or orientation to avoid incomplete drawing requirements. These controls may be specified by one of the appropriate geometric tolerances.

GENERAL RULES

2. **The application of RFS (S), MMC (M), and LMC (L) is limited to features subject to variation in size.**

   For a tolerance of position (M), (S), or (L) must be specified on the drawing with respect to the individual tolerance, datum reference, or both as applicable.

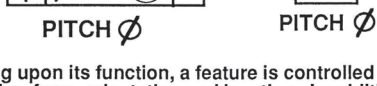


3. **For all other geometric tolerances, other than position tolerance, RFS applies with respect to the individual tolerance, datum reference, or both, where no modifying symbol is specified. MMC must be specified on the drawing where it is required.**

4. **Each tolerance of orientation or position and datum reference specified for a screw thread applies to the axis of the thread derived from the pitch cylinder. If an exception is required, the specific feature (MAJOR Ø, MINOR Ø, etc.)must be stated beneath the feature control frame or beneath the datum feature symbol as applicable.**



Each tolerance of orientation or location and datum reference specified for gears , splines, etc. must designate the specific feature of the gear, spline, etc. to which it applies (MAJOR Ø, PITCH Ø, etc.). Thisinformation is listed below the feature control frame or beneath the datum feature symbol.

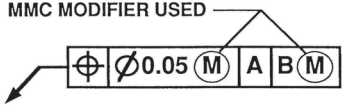


5. **Depending upon its function, a feature is controlled by tolerances such as size, form, orientation and location. In addition MMC or RFS may also be applicable. Consideration must be given to the collective effect of these factors in determining the clearance between mating parts and in establishing gaging sizes. From such consideration, a net resultant boundary is derived, called *virtual condition.***

MODIFIERS

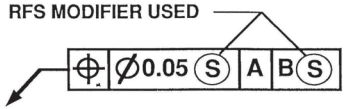
(M) Maximum Material Condition (MMC)

The condition in which a feature of size contains the maximum amount of material within the stated size limits (for example, minimum hole diameter or maximum shaft diameter). A feature identified as MMC is permitted greater positional or form tolerance as its size departs from MMC.



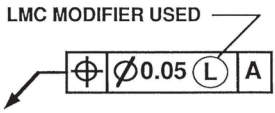
(S) Regardless of Feature Size (RFS)

The term used to indicate that a geometric tolerance or datum reference applies at any increment of size of the feature within its size tolerance. An increase in the tolerance of form or position is not permitted with size variations.

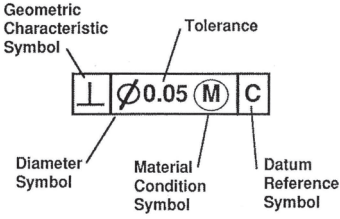


(L) Least Material Condition (LMC)

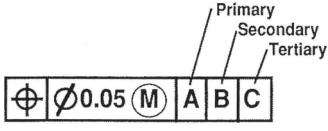
The condition in which a feature of size contains the least amount of material within the stated limits of size (for example, maximum hole diameter or minimum shaft diameter).



FEATURE CONTROL FRAME



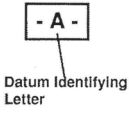
Control frame with 3 datum references



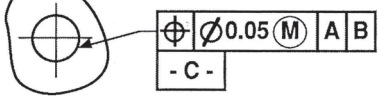
DATUMS

A Datum is a theoretically exact point, axis or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location or geometric characteristics of features of a part are established.

A Datum Feature is an actual feature of a part (for example a surface, hole, or slot) that is used to establish a datum. Each datum requiring identification is assigned a different letter (except I, O, or Q). Double letters (AA, AB, etc.) may be used when single letters are exhausted.

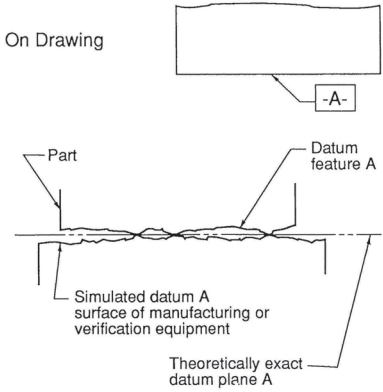


When a feature controlled by a geometric tolerance also serves as a datum, the control frame and datum are combined.



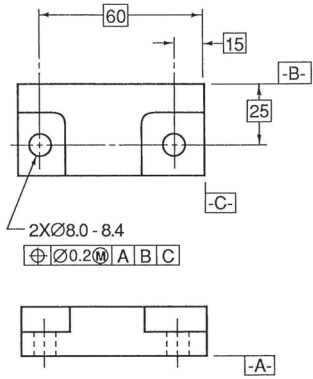
DATUM PLANES

When a nominally flat surface is specified as a datum feature, the corresponding datum is simulated by a plane contacting the high points of that surface.



THREE PLANE CONCEPT

To properly position a part on the datum reference plane, datums must be specified in order of precedence. This order is indicated by entering the appropriate datum reference letters, left to right, in the feature control frame. In this example, A is the primary datum plane, B is the secondary datum plane and C is the tertiary datum plane.

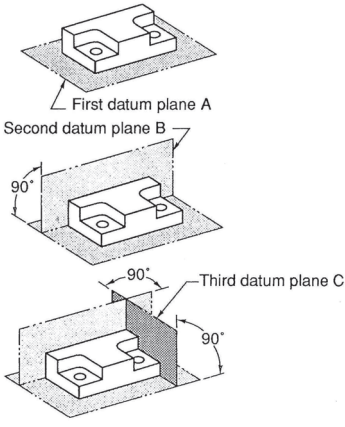


ESTABLISHING DATUM PLANES

1. The primary datum plane (A) is established with 3-point minimum contact.

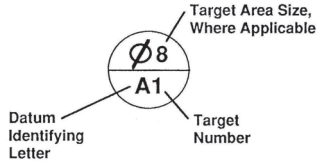
2. The secondary datum plane (B) is established with 2-point minumum contact.

3. The tertiary datum plane (C) is established with 1-point minimum contact.

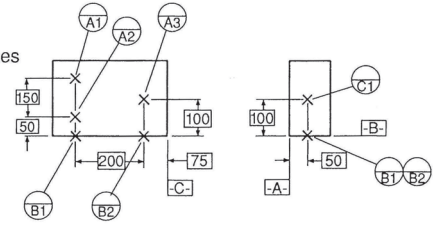


DATUM TARGETS

Datum targets are used on drawings to identify tooling points at which the part is to be supported during manufacturing. The targets can specify point, line, or area contact. Target points are indicated by the symbol "X".



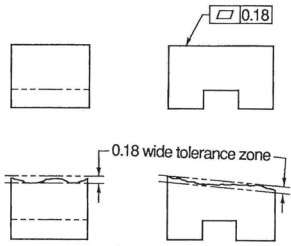
Datum points A1, A2, and A3 establish primary datum plane A. Points B1 and B2 establish second datum plane B perpendicular to A. Point C1 establishes datum plane C perpendicular to A and B.



FORM TOLERANCE

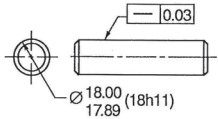
FLATNESS

Flatness is the condition of a surface having all points in one plane. A flatness tolerance specifies a zone defined by 2 parallel planes. In the example shown the surface must lie between 2 parallel planes 0.18mm apart and the surface must be within the specified size limits.



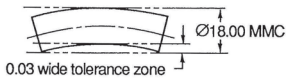
STRAIGHTNESS

Straightness is the condition where all the points on a surface or an axis are in a straight line. A straightness tolerance specifies a zone within which the surface or axis must lie. In the example the zone is bounded by two parallel lines 0.03mm apart.



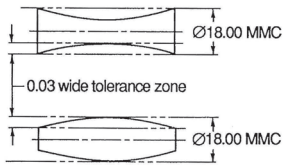
–	Ø0.03

When a diameter symbol is added to the tolerance the derived axis of the feature must lie within a cylindrical tolerance zone of 0.03mm diameter.



–	Ø0.03 Ⓜ

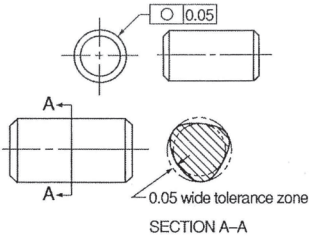
When a modifier Ⓜ is added, the tolerance zone is 0.03 diameter at 18mm diameter and the zone increases as the feature decreases from MMC.



ROUNDNESS

Roundness or Circularity is the condition where all the points on a surface are in a circle. Roundness tolerance specifies a zone bounded by 2 concentric circles within which the measured surface must lie.

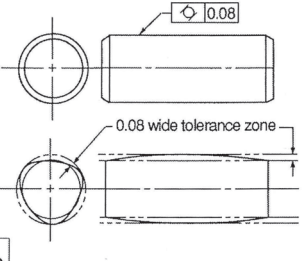
In the example shown each circular element must lie between 2 concentric circles, one having a radius 0.05mm larger than the other. Each circular element of the surface must also be within the specified limits of size.



CYLINDRICITY

Cylindricity is the condition where all points of a surface of revolution are equidistant from a common axis. Cylindricity tolerance specifies a zone bounded by 2 concentric cylinders within which the measured surface must lie.

The tolerance applies simultaneously to both circular and longitudinal elements.

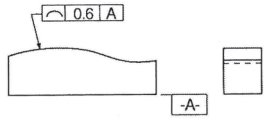


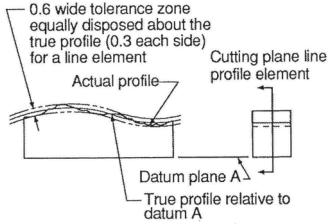
PROFILE TOLERANCE

LINE

A Profile is the outline of an object in a given plane. The tolerance zone established by the profile of a line tolerance is two dimensional extending along the length of the considered feature.

In this example each line element at any cross section must lie between 2 profile boundaries 0.6mm apart in relation to the datum plane A. Additionally the surface must be within any specified limits of size. The tolerance zone may be specified to be unilaterally disposed either inside or outside the true profile.

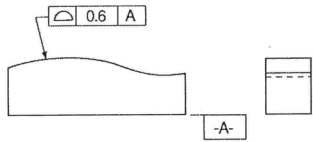


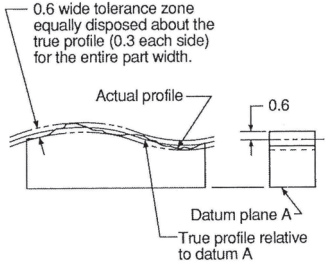


SURFACE

The tolerance zone established by the profile of a surface tolerance is three dimensional extending along the length and width (or circumference)of the considered feature.

In the example all points on the surface must lie between 2 profile boundaries 0.6mm apart in relation to datum plane A. Additionally the surface must be within any specified size limits.



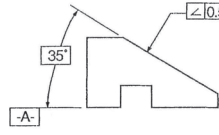


ORIENTATION TOLERANCE

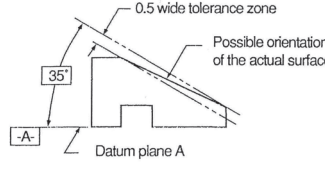
ANGULARITY

Angularity is the condition of a surface or axis at a specified angle (other than 90°) from a datum plane or axis.

The tolerance zone is defined by 2 parallel planes at the specified basic angle from a datum plane or axis. The surface or axis of the considered feature must lie within this zone.



In the example all points of the surface must lie within the 0.5mm wide tolerance zone.

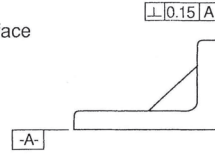


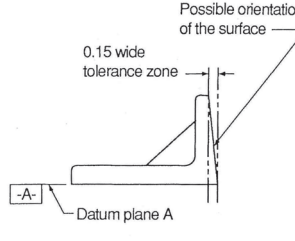
PERPENDICULARITY

Perpendicularity is the condition of a surface or axis at a right angle to a datum plane or axis.

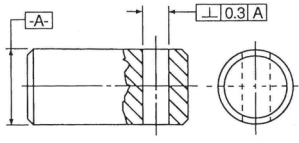
Perpendicularity tolerance specifies one of the following:

1. A zone defined by 2 parallel planes perpendicular to a datum plane or axis. In the example shown, the surface of the feature must lie within this zone which is 0.15mm wide and at right angles to datum A.

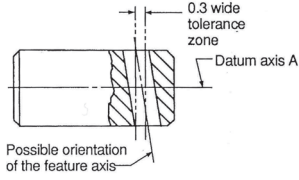




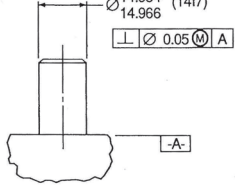
2. A zone defined by 2 parallel planes perpendicular to the datum axis. In this example, the axis of the hole must lie within the zone which is 0.3mm wide and at right angles to datum axis A. The feature axis must also be within the tolerance of location.



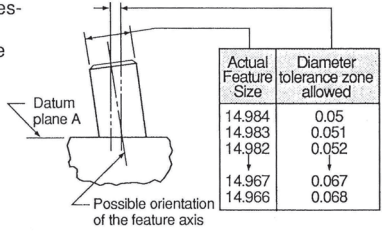
Note: The absence of a modifier indicates RFS applies.



3. A cylindrical tolerance zone perpendicular to a datum plane. In this example the axis of the part must lie within a cylindrical zone of 0.05mm diameter at right angles to datum A.



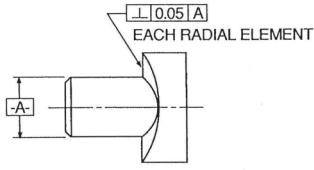
This tolerance applies at the maximum diameter of 14.984mm (MMC). As the feature size decreases from MMC, the perpendicularity tolerance zone is increased a corresponding amount. The feature axis must also be within the tolerance of location.

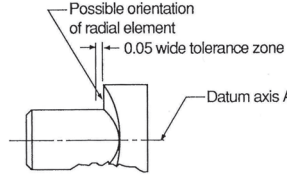


Actual Feature Size	Diameter tolerance zone allowed
14.984	0.05
14.983	0.051
14.982	0.052
↓	↓
14.967	0.067
14.966	0.068

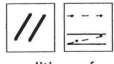
4. A zone defined by 2 parallel lines perpendicular to a datum plane or axis. In the example shown, each radial element of the surface must lie within this zone 0.05mm wide and at right angles to datum A.

   The surface must also be within the specified limits of size.





PARALLELISM

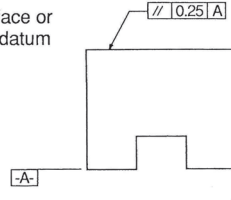


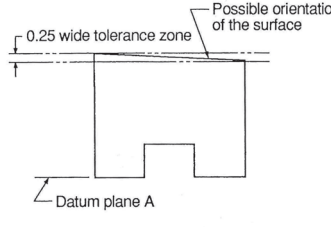
Parallelism is the condition of a surface or axis equidistant at all points from a datum plane or axis.

Parallelism tolerance specifies one of the following:

1. A zone defined by 2 planes or lines parallel to a datum plane or axis.

   In this example, the surface of the feature must lie within 2 planes 0.25mm apart which are parallel to datum A.

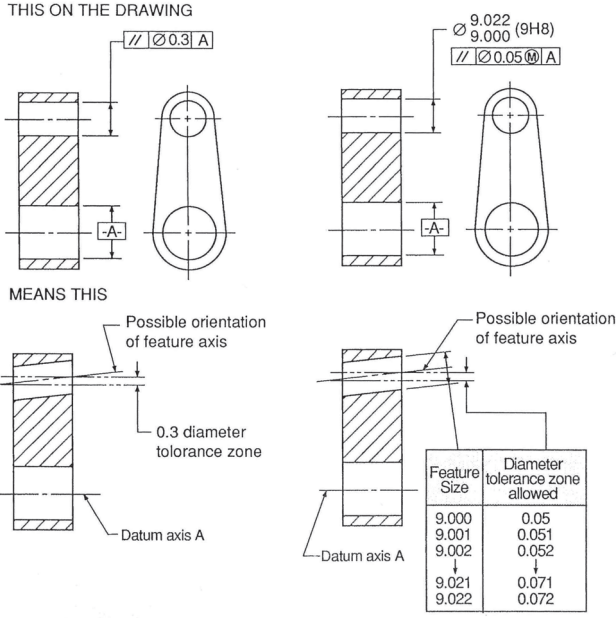




2. A cylindrical tolerance zone whose axis is parallel to a datum axis. In the first example the axis of the hole must lie within a cylindrical zone of 0.3mm diameter parallel to datum A.

   In the second example this tolerance applies at the minimum diameter of 9.000 (MMC). As the feature size increases from MMC, the parallelism tolerance zone is increased a corresponding amount.

   The feature axis must also be within the specified tolerance of location.



LOCATION TOLERANCE

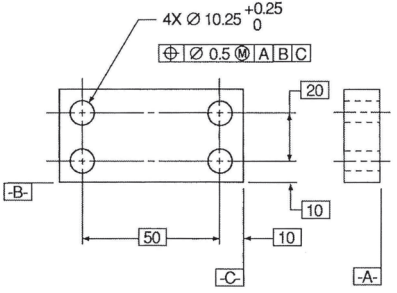
POSITION

If position tolerances are to be modified as features depart from maximum material condition, the MMC modifier must be specified on the drawing.

A positional tolerance defines a zone within which the center, axis or center plane of a feature of size is permitted to vary from the true (exact) position. Basic dimensions establish the true position.

In the example shown, the centers of the holes must lie within circles of 0.5mm diameter when the holes are at 10.25mm diameter.

As the diameters of the holes increase to 10.5mm diameter, the tolerance zones increase proportionately to 0.75mm diameter.

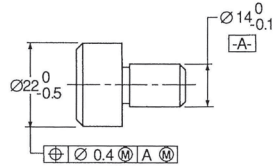


The tolerances, in this example, apply to the center distance between holes as well as the location of these features as a group from the datum planes (A-B-C).

POSITION - COAXIALITY

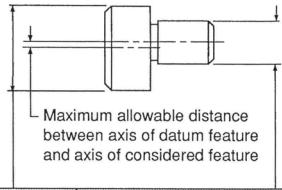
Coaxiality is the condition where the axes of two or more surfaces of revolution are coincident. The amount of permissible variation may be expressed by -

1. Positional tolerance
2. Runout tolerance
3. Concentricity tolerance



Where the surfaces are cylindrical and the control can be applied on a material condition, positional tolerancing is recommended.

In this example both the datum A and the feature size are specified on an MMC basis. As either feature or both features is reduced in size, the maximum allowable distance between feature axes is increased.

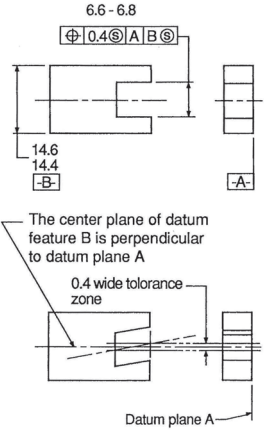


Considered feature sizes	Datum feature sizes					
	14	13.98	13.96	13.94	13.92	13.9
22	0.2	0.21	0.22	0.23	0.24	0.25
21.9	0.25	0.26	0.27	0.28	0.29	0.3
21.8	0.3	0.31	0.32	0.33	0.34	0.35
21.7	0.35	0.36	0.37	0.38	0.39	0.4
21.6	0.4	0.41	0.42	0.43	0.44	0.45
21.5	0.45	0.46	0.47	0.48	0.49	0.5

POSITION - SYMMETRY

Symmetry is the condition where a feature is symmetrically disposed about the center plane of a datum. Symmetry can be controlled by a positional tolerance.

In this example the positional tolerance and the datum references apply regardless of feature size (RFS). The center plane of the slot must lie between 2 parallel planes 0.4mm apart. Changes in feature sizes do not change the positional tolerance.



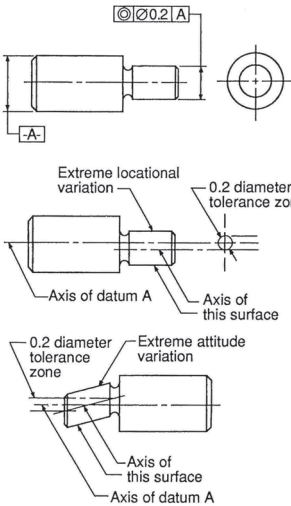
CONCENTRICITY



Concentricity is the condition where the axes of all cross sectional elements of a surface of revolution are common to the axis of a datum feature.

Concentricity tolerance specifies a cylindrical tolerance zone whose axis coincides with the datum axis.

In this example, the zone has a diameter of 0.2mm and the feature axis must lie within this zone.

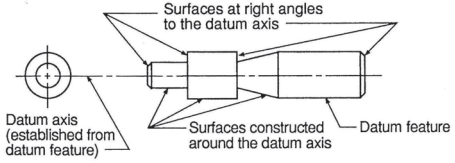


RUNOUT

CIRCULAR RUNOUT

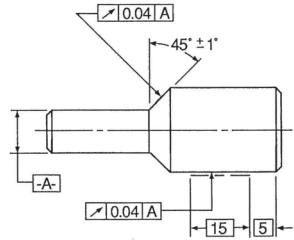


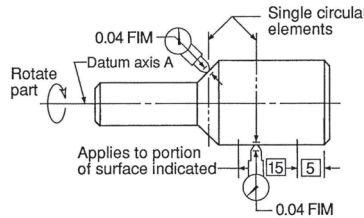
Runout is a composite tolerance used to control the relationship of one or more features to a datum axis. The illustration shows the types of features that can be controlled by runout tolerances.



Circular runout provides control of circular elements of a surface. It can be used to control the cumulative variations of circularity (roundness) and coaxiality.

In the example shown, each circular element of the surfaces toleranced must fall within 0.04mm (Full Indicator Movement) when the part is rotated 360° about the datum axis.

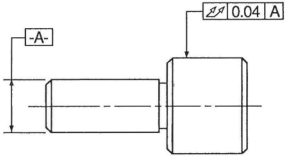




TOTAL RUNOUT

Total runout provides composite control of all surface elements. For surfaces around a datum axis this includes:

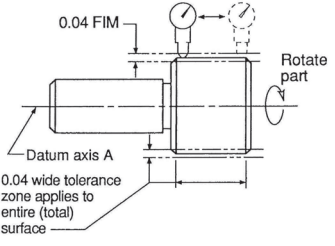
- Circularity (Roundness)
- Straightness
- Coaxiality
- Angularity
- Taper
- Profile of a Surface



For surfaces perpendicular to a datum axis it includes:

- Perpendicularity
- Flatness

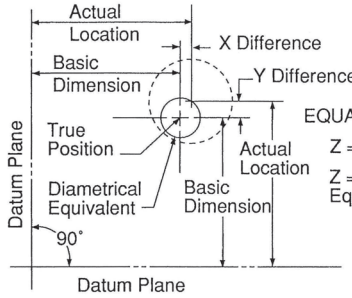
In the example shown, the entire surface must lie within the 0.04mm wide (Full Indicator Movement) tolerance zone when the part is rotated 360° about the datum axis.



CONVERSION OF COORDINATE MEASUREMENTS TO POSITION LOCATION

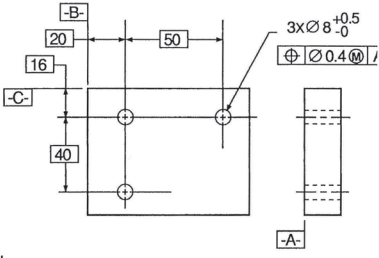
Z Values

Y \ X	0.01	0.02	0.03	0.04	0.05	0.06	0.07	0.08	0.09	0.10	0.11
0.20	0.400	0.402	0.404	0.408	0.412	0.418	0.424	0.431	0.439	0.447	0.456
0.19	0.380	0.382	0.385	0.388	0.393	0.398	0.405	0.412	0.420	0.429	0.439
0.18	0.360	0.362	0.365	0.369	0.374	0.379	0.386	0.394	0.403	0.412	0.422
0.17	0.340	0.342	0.345	0.349	0.354	0.360	0.368	0.376	0.385	0.394	0.405
0.16	0.321	0.322	0.325	0.330	0.335	0.342	0.349	0.358	0.367	0.377	0.388
0.15	0.301	0.303	0.306	0.310	0.316	0.323	0.331	0.340	0.350	0.360	0.372
0.14	0.281	0.283	0.286	0.291	0.297	0.305	0.313	0.322	0.333	0.344	0.356
0.13	0.261	0.263	0.267	0.272	0.278	0.286	0.295	0.305	0.316	0.328	0.340
0.12	0.241	0.243	0.247	0.253	0.260	0.268	0.278	0.288	0.300	0.312	0.325
0.11	0.221	0.224	0.228	0.234	0.242	0.250	0.261	0.272	0.284	0.297	0.311
0.10	0.201	0.204	0.209	0.215	0.224	0.233	0.244	0.256	0.269	0.283	0.297
0.09	0.181	0.184	0.190	0.197	0.206	0.216	0.228	0.241	0.254	0.269	0.284
0.08	0.161	0.165	0.171	0.179	0.189	0.200	0.213	0.226	0.241	0.256	0.272
0.07	0.141	0.146	0.152	0.161	0.172	0.184	0.198	0.213	0.228	0.244	0.261
0.06	0.122	0.126	0.134	0.144	0.156	0.170	0.184	0.200	0.216	0.233	0.250
0.05	0.102	0.108	0.117	0.128	0.141	0.156	0.172	0.189	0.206	0.224	0.242
0.04	0.082	0.089	0.100	0.113	0.128	0.144	0.161	0.179	0.197	0.215	0.234
0.03	0.063	0.072	0.085	0.100	0.117	0.134	0.152	0.171	0.190	0.209	0.228
0.02	0.045	0.056	0.072	0.089	0.108	0.126	0.146	0.165	0.184	0.204	0.224
0.01	0.028	0.045	0.063	0.082	0.102	0.122	0.141	0.161	0.181	0.201	0.221

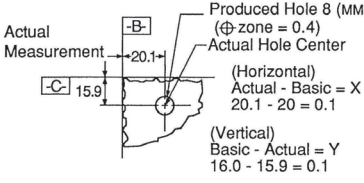


COORDINATE MEASUREMENT CONVERSION

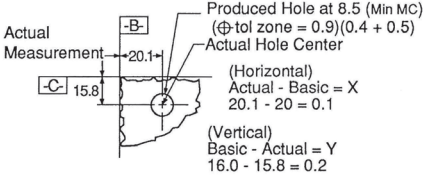
EXAMPLE



CONVERSION



From chart on Page 20, 0.1 (X) and 0.1 (Y) give 0.283 diameter hole location. This location is within the 0.4 diameter specified location.



From chart on Page 20, 0.1 (X) and 0.2 (Y) give 0.447 diameter hole location. This location is within the 0.9 diameter specified location with hole at high limit.

SI UNITS

Dividing and Multiplying Base Units

Multiples and divisions of Base Units are simply related by powers of 10 and expressed by adding a prefix to the name of the Base Unit. For example, **milli**meter for one-thousandth of a meter and **kilo**gram for one thousand grams.

MULTIPLES		DIVISIONS	
Prefix	**means**	**Prefix**	**means**
tera (10^{12})	One trillion times	deci (10^{-1})	One tenth of
giga (10^{9})	One billion times	centi (10^{-2})	One hundredth of
mega(10^{6})	One million times	milli (10^{-3})	One thousandth of
kilo (10^{3})	One thousand times	micro (10^{-6})	One millionth of
hecto (10^{2})	One hundred times	nano (10^{-9})	One billionth of
deka (10)	Ten times	pico (10^{-12})	One trillionth of

Note: In the expression of Metric values a space may be used in place of a comma. A zero always precedes the decimal point for values less than 1.

mega (M) 10^{6} = 1 000 000
micro (μ) 10^{-6} = 0.000 001

Common Length Measures and Abbreviations

1 micrometer	= 1/1,000,000 of 1 meter (10^{-6} meter)	= μm
1 millimeter	= 1/1000 of 1 meter (10^{-3} meter)	= mm
10 millimeters	= 1 centimeter (10^{-2} meter)	= cm
10 centimeters	= 1 decimeter (10^{-1} meter)	= dm
10 decimeters	= 1 meter (10^{0} meter)	= m
10 meters	= 1 dekameter (10^{1} meters)	= da
10 dekameters	= 1 hectometer (10^{2} meters)	= hm
10 hectometers	= 1 kilometer (10^{3} meters)	= km

Note: 1 meter = 1,650,763.73 Wavelengths of Krypton-86 Orange-Red Radiation.

SI SYSTEM OF METRIC CONVERSIONS

ENGLISH TO METRIC			METRIC TO ENGLISH	
inches (ins.)	x 25.4	= millimeters (mm)	x 0.04	= ins.
feet (ft.)	x 0.3	= meters (m)	x 3.3	= ft.
yards (yds.)	x 0.9	= meters (m)	x 1.1	= yds.
miles (mi.)	x 1.6	= kilometers (km)	x 0.6	= mi.
sq. inch (in^2)	x 6.5	= sq. centimeters (cm^2)	x 0.16	= in^2
sq. feet (ft^2)	x 0.09	= sq. meters (m^2)	x 11.0	= ft^2
sq. yard (yd^2)	x 0.8	= sq. meters (m^2)	x 1.2	= yd^2
acre (a)	x 0.4	= hectares (ha)	x 2.5	= a
cu. in. (in^3)	x 16.0	= cu. centimeter (cm^3)	x 0.06	= in^3
cu. ft. (ft^3)	x 0.03	= cu meters (m^3)	x 35.0	= ft^3
cu. yd. (yd^3)	x 0.8	= cu meters (m^3)	x 1.3	= yd^3
liq. quart (qt)	x 0.9	= liter (l)	x 1.05	= qt
gallon (gal)	x 0.004	= cu. meter (m^3)	x 264.2	= gal
ounce (oz-avdp)	x 28.3	= grams (g)	x 0.035	= oz
pound (lb avdp)	x 0.45	= kilogram (kg)	x 2.2	= lb
horsepower(h.p.)	x 0.75	= kilowatt (kW)	x 1.34	= h.p.
ft. per sec. (ft/s)	x 0.304	= met. per sec. (m/s)	x 3.28	= ft/s
ounce-force (ozf)	x 0.278	= newtons (N)	x 3.597	= ozf
pounds-force (lbf)	x 4.448	= newtons (N)	x 0.224	= lbf
foot-pounds (ft.lb)	x 1.355	= newton-meters (N.m)	x 0.737	= ft.lb
foot-pounds (ft.lb)	x 1.355	= joules (J)	x 0.737	= ft.lb
inch-pounds (in,lb)	x 0.112	= newton-meters (N.m)	x 8.850	= in.lb
lb. per ft (lb/ft)	x 14.593	= newton per met. (N/m)	x 0.068	= lb/ft
cycles per sec.(cps)	x 1.0	= hertz (Hz)	x 1.0	= cps
Brit.Therm Unit Btu)	x 1055.06	= joules	x 0.00094	= Btu
Degree Fah. (°F)	x 5/9 after subtract 32	= degree Celsius (°C)	x 9/5 + 32	= °F

Conversion from inches to millimeters (inches x 25.4) is exact. Conversion from millimeters to inches (mm x 0.04) is approximate; (mm x 0.039 370 1) is accurate to six significant figures for mm/in.

Converted units should be rounded off to values consistent with original accuracy.

DECIMAL EQUIVALENTS

Fractional inch	Decimal inch	mm	Fractional inch	Decimal inch	mm
1/64	0.015625	0.3969	33/64	0.515625	13.0969
1/32	0.03125	0.7938	17/32	0.53125	13.4938
3/64	0.046875	1.1906	35/64	0.546875	13.8906
1/16	0.0625	1.5875	9/16	0.5625	14.2875
5/64	0.078125	1.9844	37/64	0.578125	14.6844
3/32	0.09375	2.3813	19/32	0.59375	15.0813
7/64	0.109375	2.7781	39/64	0.609375	15.4781
1/8	0.125	3.175	5/8	0.625	15.875
9/64	0.140625	3.5719	41/64	0.640625	16.2719
5/32	0.15625	3.9688	21/32	0.65625	16.6688
11/64	0.171875	4.3656	43/64	0.671875	17.0656
3/16	0.1875	4.7625	11/16	0.6875	17.4625
13/64	0.203125	5.1594	45/64	0.703125	17.8594
7/32	0.21875	5.5563	23/32	0.71875	18.2563
15/64	0.234375	5.9531	47/64	0.734375	18.6531
1/4	0.25	6.35	3/4	0.75	19.05
17/64	0.265625	6.7469	49/64	0.765625	19.4469
9/32	0.28125	7.1438	25/32	0.78125	19.8438
19/64	0.296875	7.5406	51/64	0.796875	20.2406
5/16	0.3125	7.9375	13/16	0.8125	20.6375
21/64	0.328125	8.3344	53/64	0.828125	21.0344
11/32	0.34375	8.7313	27/32	0.84375	21.4313
23/64	0.359375	9.1281	55/64	0.859375	21.8281
3/8	0.375	9.525	7/8	0.875	22.225
25/64	0.390625	9.9219	57/64	0.890625	22.6219
13/32	0.40625	10.3188	29/32	0.90625	23.0188
27/64	0.421875	10.7156	59/64	0.921875	23.4156
7/16	0.4375	11.1125	15/16	0.9375	23.8125
29/64	0.453125	11.5094	61/64	0.953125	24.2094
15/32	0.46875	11.9063	31/32	0.96875	24.6063
31/64	0.484375	12.3031	63/64	0.984375	25.0031
1/2	0.5	12.7	1	1	25.4

Contact Hi-Tech Metrology

1800 283 095

Hi-Tech Metrology Pty. Ltd.

2 Carl Court Hallam VIC 3803 Australia

Ph: + 61 3 9702 3911 Fax: +61 3 9702 3933

www.hitechmetrology.com.au

HI-TECH METROLOGY