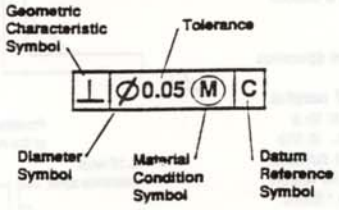
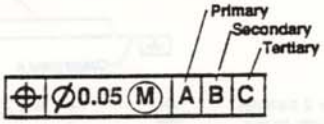
Geometric Tolerancing Explanations

(See also Kalpakjian pages 1057-1063)

FEATURE CONTROL FRAME



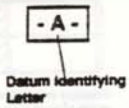
Control frame with 3 datum references



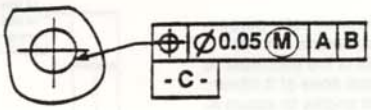
DATUMS

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.

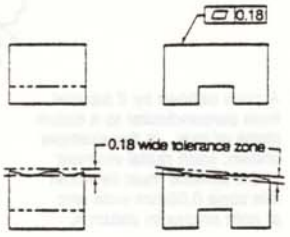


When a feature controlled by a geometric tolerance also serves as a datum, the control frame and datum are combined.



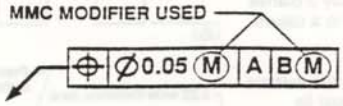
FLATNESS

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.



(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.



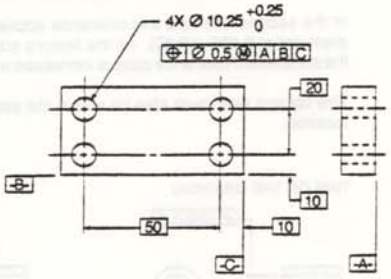
POSITION

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.

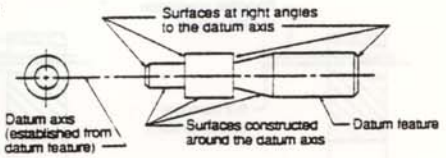
As the diameters of the holes increase to 10.5mm diameter, the tolerance zones increase proportionately to 0.75mm diameter.



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).

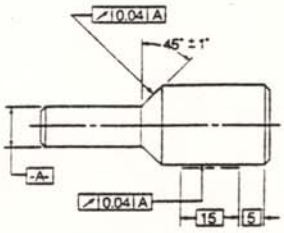
CIRCULAR RUNOUT

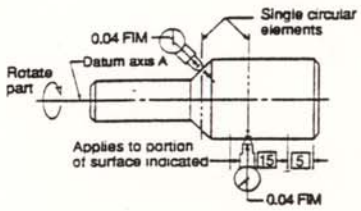
Runout is a composite tolerance used to control the relationship of one or more features to a datum axis. The illustration shows the types of features that can be controlled by runout tolerances.



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.





Geometric Tolerancing Explanations (cont.)

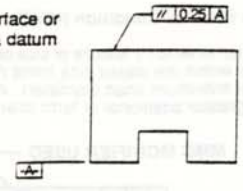
PARALLELISM

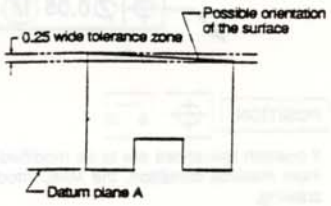
Parallelism is the condition of a surface or axis equidistant at all points from a datum plane or axis.

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.



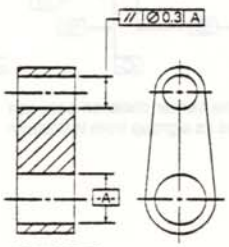


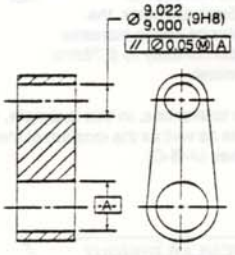
2. A cylindrical tolerance zone whose axis is parallel to a datum axis. In the first example the axis of the hole must lie within a cylindrical zone of 0.3mm diameter parallel to datum A.

   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.

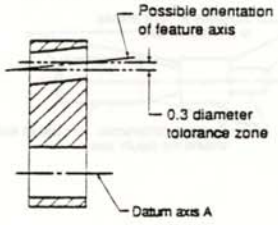
   The feature axis must also be within the specified tolerance of location.

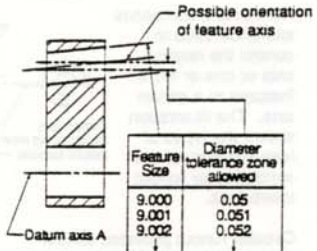
THIS ON THE DRAWING





MEANS THIS





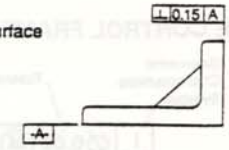
Feature Size	Diameter tolerance zone allowed
9.000	0.05
9.001	0.051
9.002	0.052
↓	↓
9.021	0.071
9.022	0.072

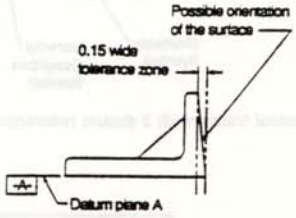
PERPENDICULARITY

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

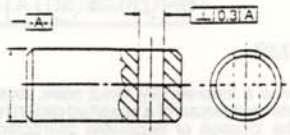
Perpendicularity tolerance specifies one of the following:

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.

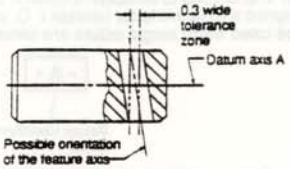




2. A zone defined by 2 parallel planes perpendicular to the datum axis. In this example, the axis of the hole must lie within the zone which is 0.3mm wide and at right angles to datum axis A. The feature axis must also be within the tolerance of location.

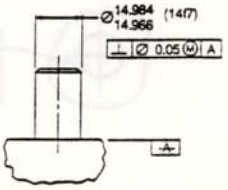


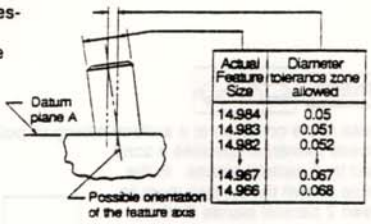
Note: The absence of a modifier indicates RFS applies.



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.