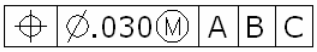
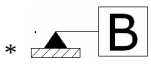
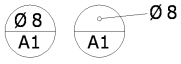
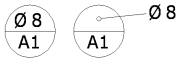
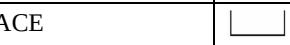
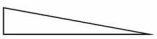
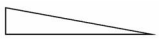
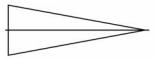
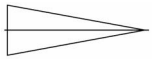
COMMON SYMBOLS/通用符号

Shown below are the most common symbols that are used with geometric tolerancing and other related dimensional requirements on engineering drawings. Note the comparison with the ISO standards. Most of the symbology is identical. There are a few symbols that are used in the ASME Y 14.5, 1994 standard that are being proposed for the ISO standards. The symbols marked with an "x" are new or revised from the previous Y14.5M, 1982 standard.

/以下显示的是最通用的符号，在工程图中，它和几何公差及其它相关的尺寸要求联合使用。下表同时列出 ISO 标准中相应的符号。绝大部分符号表示是一样的。一少部分 ASME Y 14.5, 1994 标准独有的符号，正被提议用于 ISO 标准。标记着"x"的符号是新的或者是由以前的 Y14.5M, 1982 标准修订的。

	SYMBOL（符号）	ASME Y14.5M	ISO
	FEATURE CONTROL FRAME 公差框格		
	DIAMETER/直径	Ø	Ø
	SPHERICAL DIAMETER/球直径	SØ	SØ
	AT MAXIMUM MATERIAL CONDITION 最大实体条件	Ⓜ	Ⓜ
	AT LEAST MATERIAL CONDITION 最小实体条件	Ⓛ	Ⓛ
	REGARDLESS OF FEATURE SIZE 忽视特征尺寸	NONE/什么也没有	NONE/什么也没有
	PROJECTED TOLERANCE ZONE 投影公差带	Ⓟ	Ⓟ
X	FREE STATE/自由状态	Ⓕ	Ⓕ
X	TANGENT PLANE 相切面	Ⓣ	Ⓣ (Proposed)（被提议的）
X	STATISTICAL TOLERANCE 统计公差	ST	NONE/什么也没有
X	RADIUS/半径	R	R
X	CONTROLLED RADIUS /约束半径	CR	NONE/什么也没有
	SPHERICAL RADIUS/球半径	SR	SR
	BASIC DIMENSION/基本尺寸 (theoretically exact dimension in ISO) （ISO 中理论上精确的尺寸）	50	50
X	DATUM FEATURE 基准特征		
	DATUM TARGET 基准目标		
	TARGET POINT/基准点	×	×
	DIMENSION ORIGIN 尺寸原点		
	REFERENCE DIMENDION/参考尺寸 (Auxiliary dimension in ISO) /ISO 中辅助的尺寸	(50)	(50)

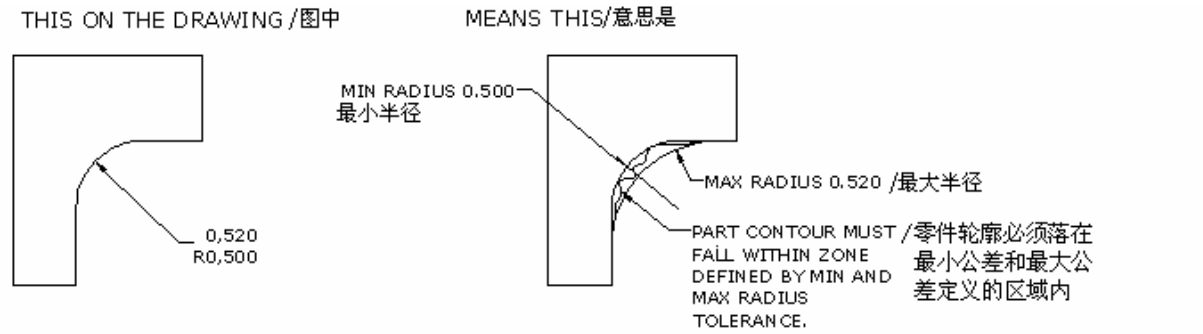
	NUMBER OF PLACES /处数	8x	8x
	COUNTERBORE/SPOTFACE 镗沉头孔/孔口平面		
	COUNTERSINK/钻沉头孔		
	DEPTH/DEEP/深度		
	SQUARE/正方形		
	ALL AROUND/全周		NONE
	DIMENSION NOT TO SCALE 尺寸没按比例		
	ARC LENGTH/圆弧长度		
X	BETWEEN/在两者之间		NONE
	SLOPE/斜面		
	CONICAL TAPER/锥形		
	ENVELOPE PRINCIPLE 包络线原理	NONE(implied)/（暗指）	

Note: * May be filled or not filled/注：*可能填实也可能不填实

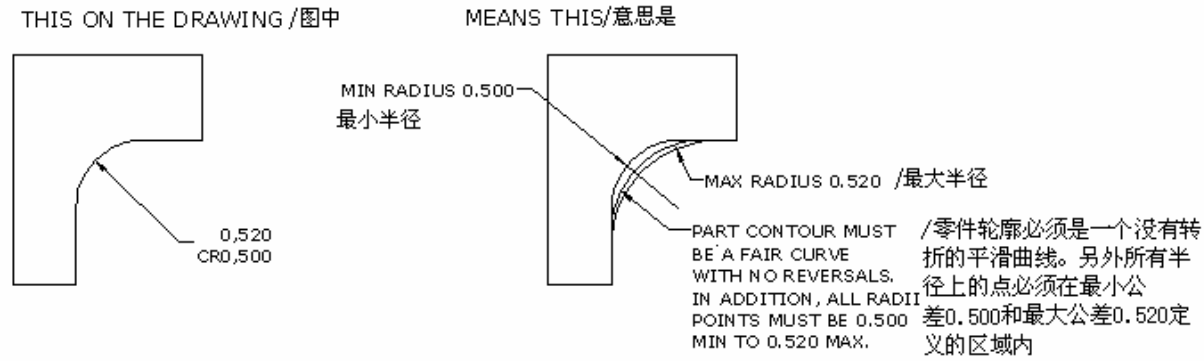
RADIUS, CONTROLLED RADIUS/半径，约束半径

There are two types of radii tolerance that can be applied, the radius and controlled radius. The radius (R) tolerance is for general applications. The controlled radius (CR) is used when it is necessary to place further restrictions on the shape of the radius, as in high stress applications.
/有两种类型半径公差能被应用：半径和约束半径。半径（R）公差是通用的。而约束半径(CR)则用于需要对半径的形状作进一步的约束时，作为一种高应力的应用。

Note: This is a change from the previous editions of the Y14.5 standard. The definition of the tolerance zone for the former term tangent radius, previously noted by the symbol R, is now meant to apply to a controlled radius (symbol CR).
/注：这个是从 Y14.5 标准以前的版本改变而来的。先前的术语相切半径的公差带定义，以前用符号 R 表示，现在用约束半径（符号 CR）表示。



CONTROLLED RADIUS, SYMBOL CR/受控半径，符号CR

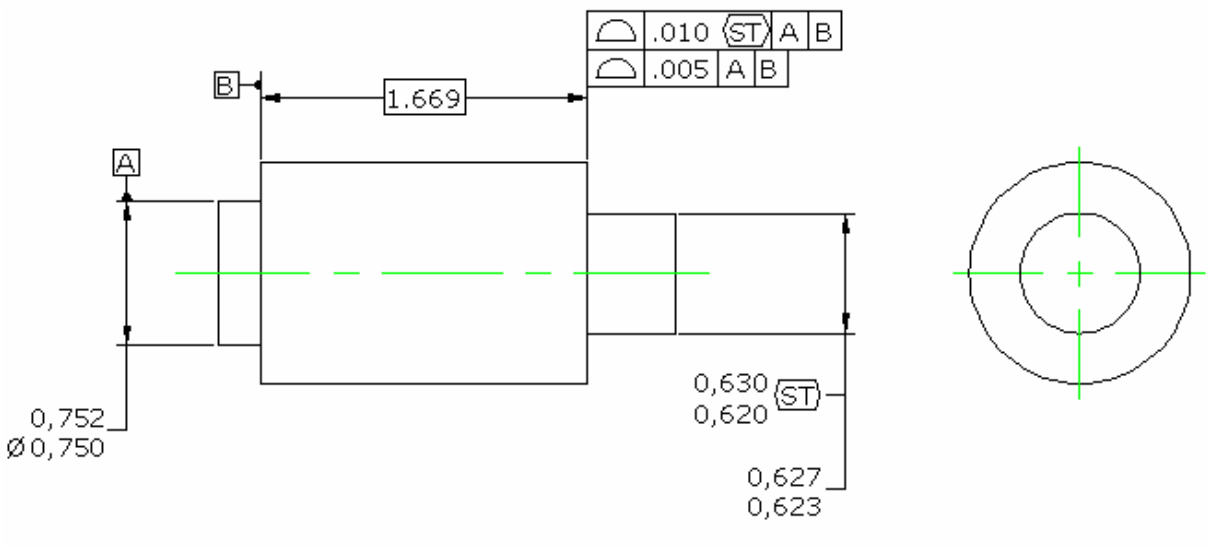


STATISTICAL TOLERANCING SYMBOL/统计公差符号

Often, tolerances are calculated on an arithmetic basis. Tolerances are assigned to individual features on a component by dividing the total assembly tolerance by the number of components and assigning a portion of this tolerance to each component. When tolerances are stacked up in this manner, the tolerance may become very restrictive or tight.
/公差经常基于算术基础来计算。通过根据部件的数量把总的装配公差分划，并把一部分公差指定给每一部件的方式，公差被指定给部件的个别特征。当公差以这种方式堆叠时，公差可能变得非常受限制或者紧密。

Statistical tolerancing is the assignment of tolerances to related components of an assembly on the basis of sound statistics. An example is: the assembly tolerance is equal to the square root of the sum of the squares of the individual tolerances.
/统计公差标注是在合理的统计学基础上，对相关装配部件的公差分配。例如：装配公差等于单个的公差平方的和的平方根。

Statistical tolerancing may be applied to features to increase tolerances and reduce manufacturing cost. To ensure compatibility, the larger tolerance identified by the statistical tolerance symbol may only be used where appropriate statistical process control will be used. A note such as the one shown below shall be placed on the drawing.
/统计公差可能应用到特征上，用以增加公差和减小加工费用。为保证兼容性， 被统计公差确定的较大公差可能只用于使用适当的统计工艺控制的地方。像下面所示的注释将被放置到工程图上。



Note:/注：

FEATURES IDENTIFIED AS STATISTICALLY TOLERANCED ST SHALL BE PRODUCED WITH STATISTICAL PROCESS CONTROLS, OR TO THE MORE RESTRICTIVE ARITHMETIC LIMITS.

/以统计公差 ST 识别的特征将在统计学工艺控制下生产，或者直到受更严格的算法约束为止。

In some cases, it may be desirable to state only the statistical tolerance and the arithmetic number will not be shown. In this case, a note such as the following must be placed on the drawing.
/某些情况下，可以只规定统计公差，而不显示算术数字。这种情况下，像下面所示的注释必须放在工程图中。

FEATURES IDENTIFIED AS STATISTICALLY TOLERANCED ST SHALL BE PRODUCED WITH STATISTICAL PROCESS CONTROLS.

/以统计公差 ST 识别的特征将在统计学工艺控制下生产。

For additional information on statistical tolerancing, see appropriate statistics or engineering design manuals.
/对于统计公差标注中的额外信息， 请参见适当的统计学或工程学设计手册。

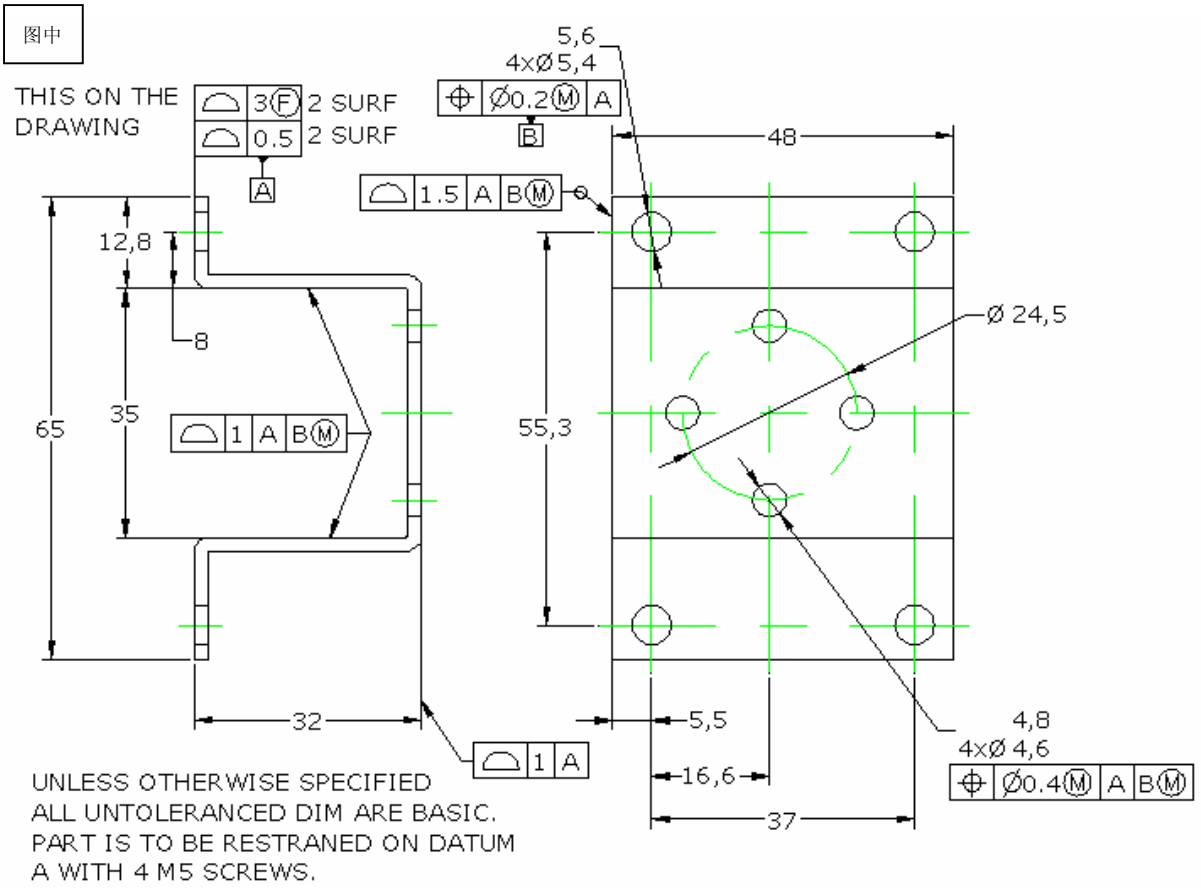
NONRIGID PARTS-FREE STATE CONDITION/非刚性零件 - 自由状态条件

Unless otherwise specified, all dimensioning and tolerancing applies in a Free State condition with no restraint. Some parts, such as sheet metal, thin metal, plastics and rubber are no rigid in nature. It may be necessary to specify design requirements on the part in a natural or Free State as well as in a restrained condition. The restraint or force on the nonrigid parts is usually applied in such a manner to resemble or approximate the functional or mating requirements.
/除非特别指定，所有尺寸标注和公差标注用于没有限制的自由状态。一些零件，诸如钣金件，薄金属件，塑料件和橡胶自然状态不是刚性的。在零件的自然状态或者自由状态以及受限制的条件下指定设计需求可能是必需的。对非刚性零件的限制或影响力通常以这样的方式达到相像或近似功能或匹配的需求。

A note or specification on the drawing should explain how the part is restrained and the force required to facilitate the restraint. A sample note can be found on the drawing below.
/一个图上的注释或规范应该解释零件如何被限制，和需要限制必需的力量。在下面的工程图中能发现一个示例注释。

If any of the part specifications are to be verified in a free state, the designer may specify this requirement with the words FREE STATE or the free state symbol. The free state symbol is a F in a circle. The free state symbol means that dimensions and tolerances that have the free state symbol applied are checked in the free state and not in the restrained condition.
/如果任何一个零件规范都以自由状态验证，设计者可以用“自由状态“一词或自由状态符号指定这个需求。自由状态符号是一个字母 F 在一个圆圈里。自由状态符号意思是应用自由状态符号的尺寸和公差以自由状态而不是以限制条件来查证。

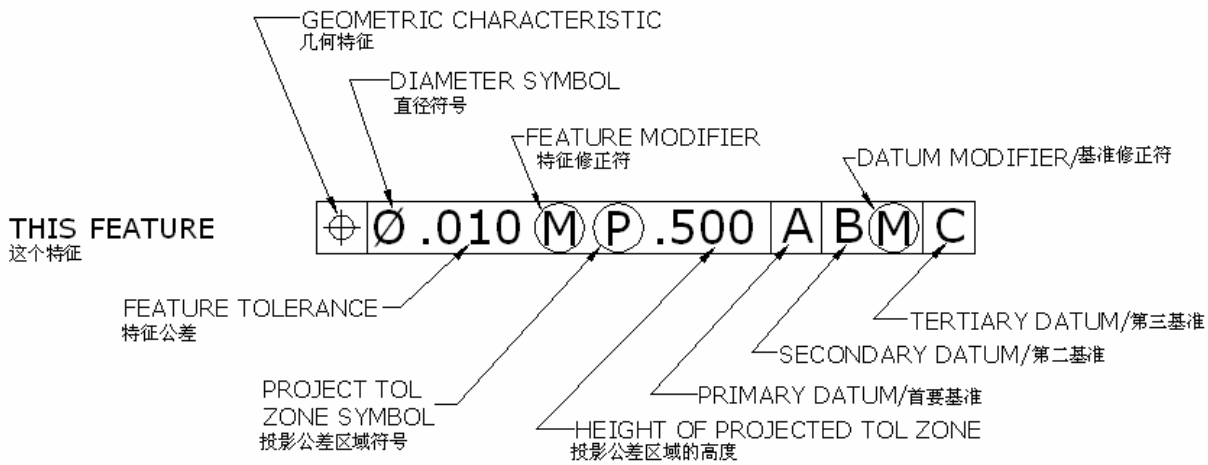
The free state symbol is applied by placing it next to or associating it with the required dimensions and tolerances. If it is applied in a feature control frame, it always follows the feature tolerance and any modifiers.
自由状态符号通过和要求的尺寸和公差紧挨着或者联合着放置应用到工程图中。如果被放在一个公差框格里，则总是跟在这个特征和任一修正符的后面。



/除非额外指定，所有未标公差的尺寸是基本的。零件使用 4 个 M5 螺钉限制在基准 A 上。

FEATURE CONTROL FRAME / 公差框格

A feature control frame states the requirements or instructions for the features to which it is attached.
/一个公差框格规定对所附着的特征的要求或指示。



SYMBOLS THAT CAN BE FOUND IN A FEATURE CONTROL FRAME/公差框格中的符号

All of the symbols below can be found inside a feature control frame. The symbols for projected tolerance zone, free state, tangent plane and statistical tolerance always follow the material condition modifier。 The minimum height of the projected tolerance zone can be specified in the feature control frame or in the view on the drawing in which it applies. If it is applied in the feature control frame, it follows the projected tolerance zone symbol.
/以下所有符号都能在公差框格中找到。投影公差带，自由状态，相切面和统计公差总是跟在材料条件修正符之后。投影公差带的最小高度可能在公差框格中被指定，或者在图中它应用到的视图中被指定。如果应用在公差框格中，则跟在投影公差带符号之后。

TERM/术语	SYMBOL/符号
FEATURE CONTROL FRAME/公差框格	⌖ ∅0.010 A B C
DIAMETER/直径	Ø
SPHERICAL DIA/球直径	sØ
MAXIMUM MATERIAL COND/最大实体条件	Ⓜ
LEAST MATERIAL COND/最小实体条件	Ⓛ
PROJECTED TOL ZONE/投影公差带	Ⓟ
FREE STATE/自由状态	Ⓕ
TANGENT PLANE/相切面	Ⓣ
STATISTICAL TOLERANCE/统计公差	(ST)
REGARDLESS OF FEATURE SIZE*/忽略特征尺寸*	Ⓢ

**The RFS symbol has been eliminated from the ASME Y14.5M, 1994 standard. It is applicable in earlier versions.*
*/*RFS 符号已经在 ASME Y14.5M, 1994 标准里消除了。它应用在更早的版本中。*

DATUM FEATURE SYMBOL/基准特征符号

Y 14.5M, 1982 AND EARLIER
/Y 14.5M, 1982 和更早的

Y14.5M, 1994 AND ISO
/ Y14.5M, 1994 和 ISO

MATERIAL CONDITION MODIFIERS – DEFINITIONS/材料条件修正符-定义

In geometric tolerancing there is often a need to refer to a particular feature of size at its largest size, smallest size or regardless of feature size. The terms maximum material condition (MMC), least material condition (LMC) and regardless of feature size (RFS) allow us to do this.
/几何公差标注中，经常有这么一个需求，在最大尺寸，最小尺寸，或是忽略特征尺寸条件下谈到一个特别的尺寸特征。术语最大实体条件(MMC),最小实体条件(LMC)和忽略特征尺寸（RFS）允许我们这样做。

These terms can only be used when referring to features of size such as holes, slots, tabs, pins, etc. these terms have no meaning when applied to non-features of size such as plane surfaces. The application or implication of these material condition modifiers inside the feature control frame can have a substantial effect on the tolerance. See discussion on maximum material condition later in text.
/这些术语只用于当提及诸如孔，槽，突起，销子等等时。当用于诸如平面的没有特征尺寸的时候，这些术语没有什么意思。公差框格中材料修正符的应用或含义对公差有实质的影响。在本文稍后参见最大实体条件的讨论。

MAXIMUM MATERIAL CONDITION-ABBREVIATION (MMC) SYMBOL： Ⓜ

/最大实体条件-缩写（MMC） 符号： Ⓜ

The condition where the feature contains the maximum material within the stated limits of size – for example, the largest pin or smallest hole.
/特征在规定的尺寸极限内包含最大材料的情形 – 例如，最大的销子或者最小的孔。

LEAST MATERIAL CONDITION-ABBREVIATION (LMC) SYMBOL： Ⓛ

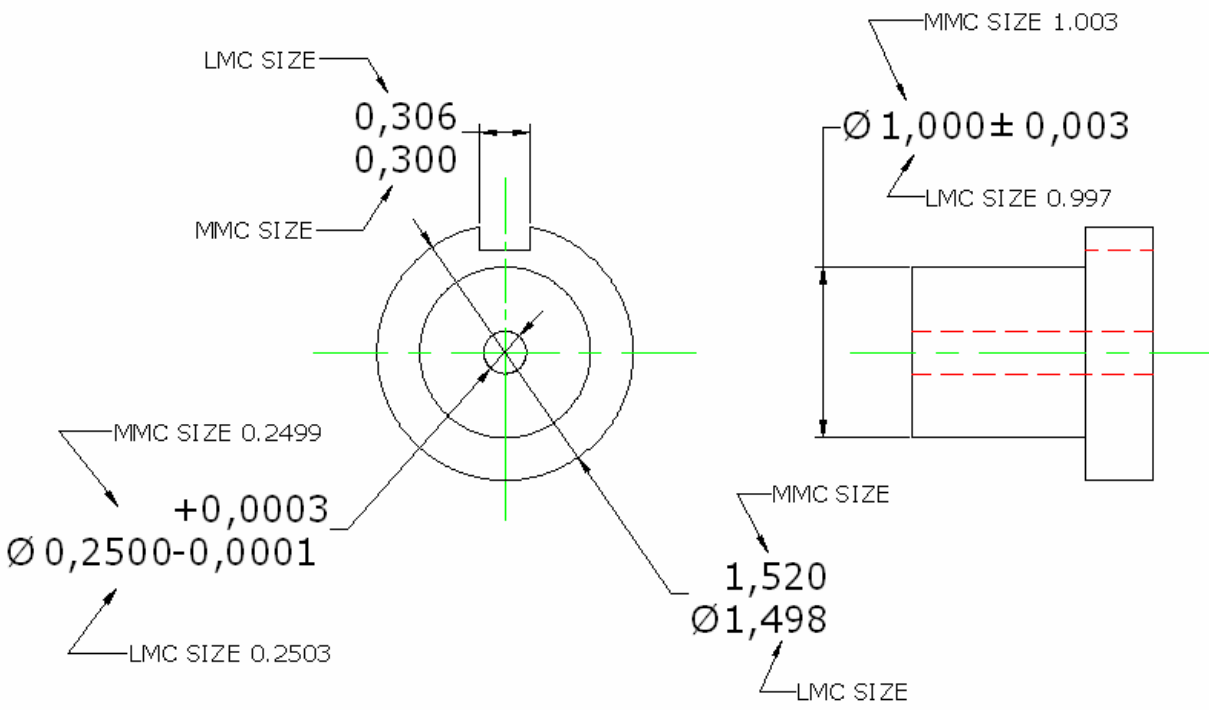
/最小实体条件 – 缩写（LMC） 符号： Ⓛ

The condition where the feature contains the least material within the stated limits of size-for example, the largest hole or smallest pin.
/特征在规定的尺寸极限内包含最小材料的情形 – 例如，最大的孔或者最小的销子。

REGARDLESS OF FEATURE SIZE-ABBREVIATION (RFS)/忽略特征尺寸 – 缩写（RFS）

The term used to indicate that a geometric tolerance applies at any increment of size of the feature within its size limits. /这个术语过去常常预示一个几何公差在它的尺寸极限内的任何特征尺寸增量。
In the current ASME Y 14.5M, 1994 standard there is no symbol for RFS. (Unless otherwise specified all geometric tolerances are implied RFS. See rule 2.) In previous editions of the Y14.5 standard the Ⓢ symbol was used for RFS.
/在现行的 ASME Y 14.5M, 1994 标准中，没有 RFS 的符号。（除非特别指定，所有几何公差暗含 RFS 的意思。参见规则 2。）在 Y14.5 标准的先前版本中符号Ⓢ用于 RFS 中。

MMC & LMC DEFINITION/MMC&LMC 的定义



图中 MMC SIZE 是最大实体尺寸，LMC SIZE 时最小实体尺寸。

EFFECT OF MAXIMUM MATERIAL CONDITION/最大实体条件的影响

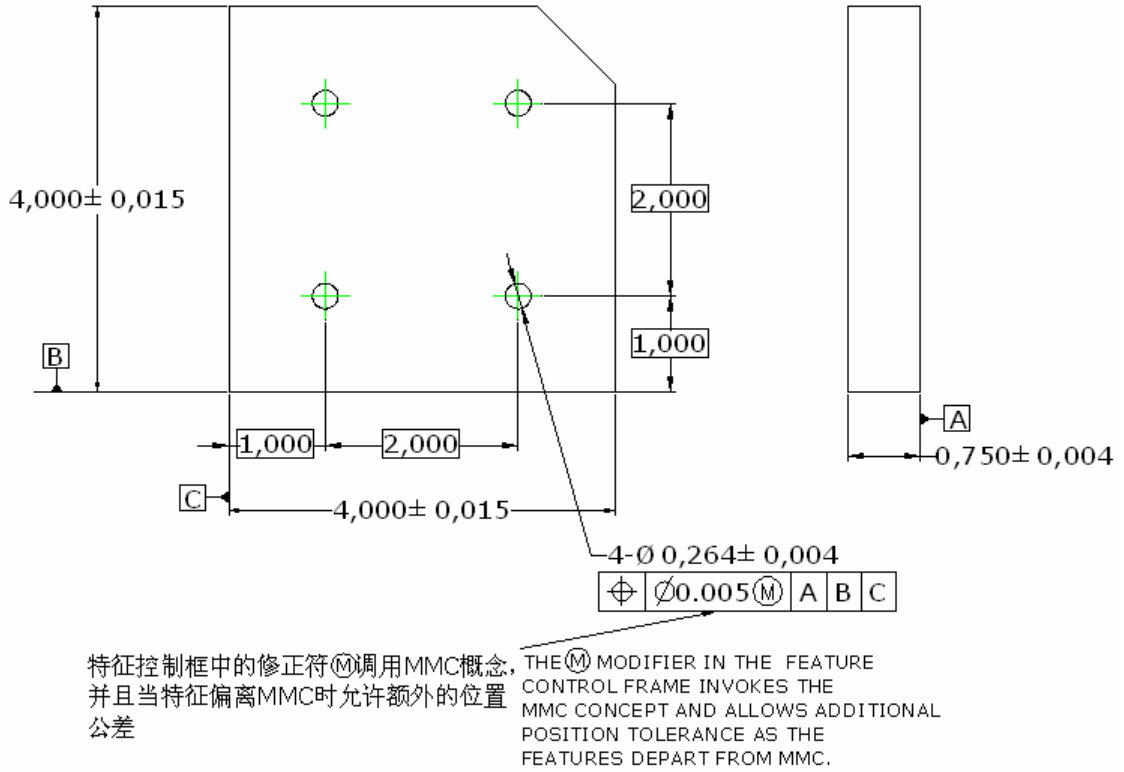
A geometric tolerance (size features only) may be applied on an MMC basis by placing the circle M symbol in the feature control frame following the feature tolerance. This will have a substantial effect on the allowable position tolerance. The allowable position tolerance is dependent on the actual mating size of the considered feature.
/一个几何公差（仅仅尺寸特征）可以通过在特征公差后的公差框格中放置一个带圆圈的 M 符号应用 MMC 概念。这对于允许的位置公差将有一个实质性影响。允许的位置公差依赖于所考虑特征的实际配合尺寸。

The allowable geometric tolerance for the feature applies when the feature is produced at its maximum material condition (smallest hole or largest pin). If the considered feature's size departs from its maximum material condition, an increase in the allowable geometric tolerance is permitted equal to the amount of the feature's

departure from MMC.
/当以最大实体条件（最小的孔或最大的销子）生产特征时，应用特征允许的几何公差。如果所考虑的特征的尺寸偏离它的最大实体条件，一个允许的几何公差的增量可以等于特征偏离最大实体条件的数量。

Consider the part shown below. The circle M modifier in the feature control frame states that the features must be positioned within a .005 diameter tolerance zone when the features are at their maximum material condition. The maximum material condition (MMC) for the holes is .260 dia. If the features depart from the .260 dia. size, they can have additional position tolerance equal to the amount of their departure from MMC.
/考虑以下所示零件。公差框格中的带圆圈的 M 修正符规定特征在最大实体条件时必须位于一个 0.005 直径公差带中。对于孔的最大实体条件是直径 0.260。如果特征偏离直径 0.260 尺寸，它们有另外的位置公差等于偏离 MMC 的数量。

MMC CONCEPT/最大实体条件概念



DIAMETER FEATURE SIZE /直径特征尺寸	DIAMETER TOL ZONE ALLOWED /允许的直径公差区域
0.260	0.005
0.261	0.006
0.262	0.007
0.263	0.008
0.264	0.009
0.265	0.010
0.266	0.011
0.267	0.012
0.268	0.013

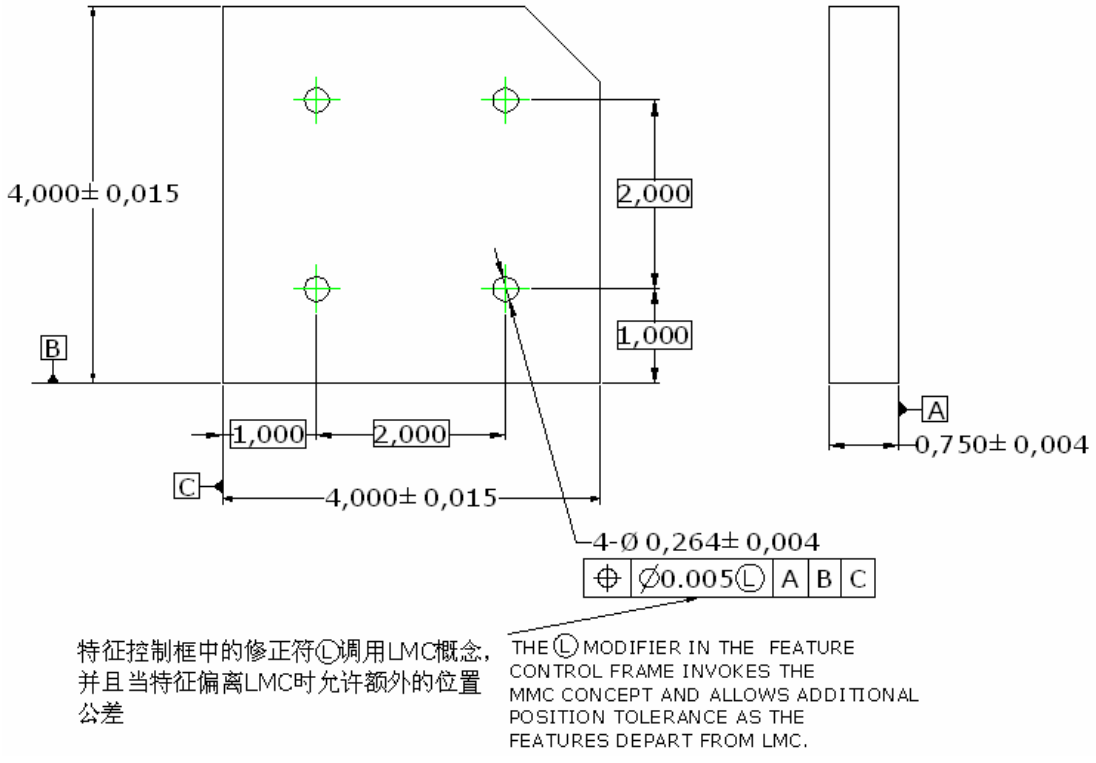
EFFECT OF LEAST MATERIAL CONDITION/最小实体条件的影响

A geometric tolerance (size features only) may be applied on an LMC basis by placing the circle L symbol in the feature control frame following the feature tolerance. This will have a substantial effect on the allowable position tolerance. The allowable position tolerance is dependent on the actual mating size of the considered feature.
/一个几何公差（仅仅尺寸特征）可以通过在特征公差后面的公差框格中放置一个带圆圈的 L 符号来应用 LMC 概念。这对于允许的位置公差将有一个实质性影响。允许的位置公差依赖于所考虑特征的实际配合尺寸。

The allowable geometric tolerance for the feature applies when the feature is produced at its least material condition (smallest pin or largest hole). If the considered feature's size departs from its least material condition, an increase in the allowable geometric tolerance is permitted equal to the amount of the feature's departure from LMC.
/当以最小实体条件（最小的销子或最大的孔）生产特征时，特征应用允许的几何公差。如果所考虑的特征的尺寸偏离它的最小实体条件，一个允许的几何公差的增量可以等于特征偏离最小实体条件的数量。

Consider the part below. The circle L modifier in the feature control frame states that the features must be positioned within a .005 diameter tolerance zone when the features are at their least material condition. The least material condition for the holes is .268 diameter. If the features depart from the .268 dia. size, they can have additional position tolerance equal to the amount of departure from LMC.
/考虑以下所示零件。公差框格中的带圆圈的 L 修正符规定特征在最小实体条件时必须位于一个 0.005 直径公差带中。对于孔的最小实体条件是直径 0.268。如果特征偏离直径 0.268 尺寸，它们有另外的位置公差等于偏离 LMC 的数量。

LMC CONCEPT/最小实体条件概念



DIAMETER FEATURE SIZE /直径特征尺寸	DIAMETER TOL ZONE ALLOWED /允许的直径公差区域
0.268	0.005
0.267	0.006
0.266	0.007
0.265	0.008
0.264	0.009
0.263	0.010
0.262	0.011
0.261	0.012
0.260	0.013

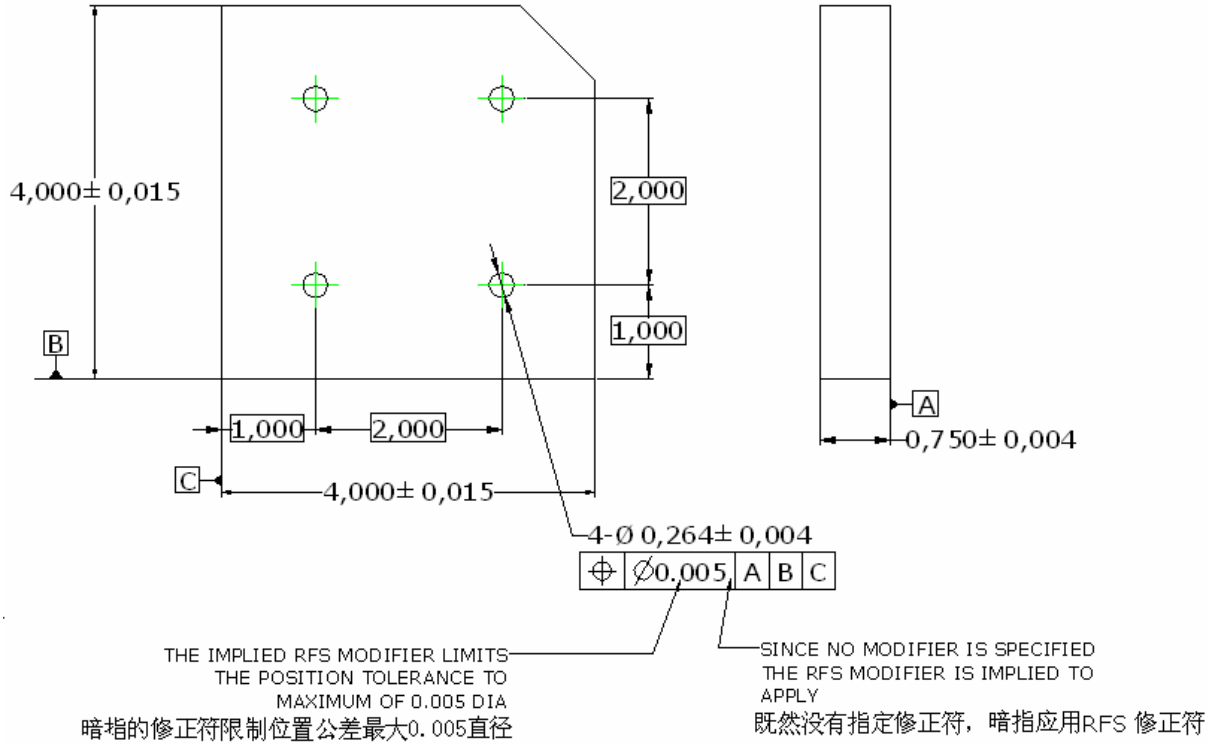
EFFECT OF REGARDLESS OF FEATURE SIZE – RFS/忽略特征尺寸的影响

Geometric tolerances (size features only) are implied on an RFS basis by implication. The modifier rule #2 states that unless otherwise specified, all geometric tolerances are by default implied to apply at RFS. Since all geometric tolerances apply at RFS, there is no need for an RFS symbol, and it has been eliminated in the ASME Y14.5M-1994 standard。In past editions of the Y14.5 standard, the RFS symbol, which is an S in a circle, was specified for position tolerances. See modifier rules for more information.
/几何公差（仅仅尺寸特征）暗含 RFS 的基本原则。除非特别指定，修正符规则 2 规定所有几何公差默认暗指 RFS. 既然所有几何公差应用 RFS, 就不需要符号 RFS 了，它已在 ASME Y14.5M-1994 标准里删除了，在 Y14.5 标准的过去的版本里, RFS 符号用一个 S 带一个圆圈表示，来指定位置公差。请参见修正符规则以获取更多的信息。

If a geometric tolerance is by implication applied RFS, the specified allowable geometric tolerance is independent of the actual size of the considered feature. The allowable geometric tolerance is limited to the specified value regardless of the actual size of the feature.
/如果一个几何公差暗指应用 RFS, 指定的允许几何公差是不受所考虑特征的确切尺寸约束的。允许的几何公差被限制在忽略特征确切尺寸的指定值里。

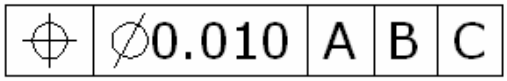
Consider the part below. Since no modifier is specified in the feature control frame following the feature tolerance, this tolerance is implied to apply RFS. The features must be positioned within a .005 dia tolerance regardless of their feature size. This means that regardless of the size of the features being positioned, they have a .005 dia position tolerance and no more. If the holes get large or small, the position tolerance remains .005 dia. The RFS condition is more restrictive than the MMC or LMC concept.
/考虑以下的零件。既然在跟随特征公差的公差框格里没有指定修正符，那么公差暗指应用 RFS 原则. 特征必须位于 0.005 直径公差内，忽略特征尺寸。意思是忽略正在决定位置的特征尺寸，它们有一个 0.005 直径的位置度公差，不会更大了。如果孔变大或变小， 位置度公差仍然是直径 0.005。RFS 条件比 MMC 或 LMC 概念更有限制性。

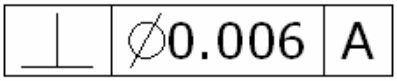
RFS CONCEPT /忽略特征尺寸的概念

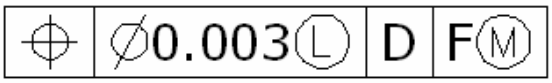


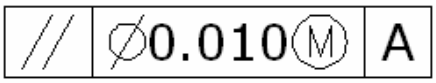
MODIFIER RULES – CURRENT AND FORMER PRACTICES/修正符规则 – 现行的和从前的惯例

Current ASME Y14.5M, 1994 and ISO – rule #2(old rule #3eliminated)
/现行的 ASME Y14.5M, 1994 和 ISO – 规则 2 （旧规则 3 已删除）

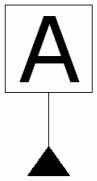






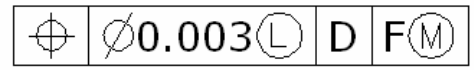


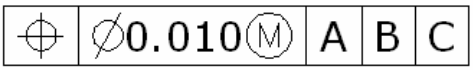
RFS applies for all geometric tolerances with respect to the individual tolerance, datum reference, or both, where no modifying symbol is specified. MMC and LMC must be specified where required.
/在没有指定修正符号的地方，RFS 应用于关于单独的公差，基准参照，或两者都有关的所有几何公差。MMC 和 LMC 必须在需要的地方指定。



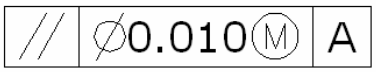
Datum feature symbol 基准特征符号
Y14.5M, 1994 and ISO

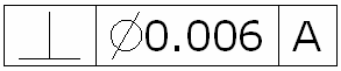
Former practice – ANSI Y 14.5M, 1982/从前的惯例 - ANSI Y 14.5M, 1982



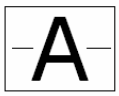


Rule #2 – MMC, LMC and RFS must be specified for individual tolerances and datum references for all position tolerances.
/规则 2 – 对于所有的位置公差，必须对单独的公差和基准参照指定 MMC, LMC 和 RFS。



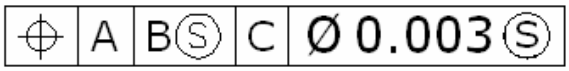


Rule#3- RFS applies for individual tolerances and datums on all other geometric tolerances. MMC and LMC must be specified where it is required.
/规则 3 – RFS 在所有的其它的几何公差上，应用于单独的公差和基准. MMC 和 LMC 必须在需要处指定。

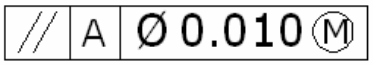


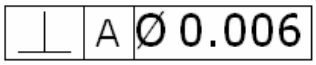
Datum feature symbol 基准特征符号
Y14.5M, 1982 and earlier/和更早

Former practice – ANSI Y 14.5M, 1973 / 更早的惯例



Rule #2 – MMC applies for individual tolerances and datum references for position. RFS and LMC must be specified where it is required.
/规则 2 –MMC 应用于单独的公差和位置的基准参照。 RFS 和 LMC 必须在需要处指定。



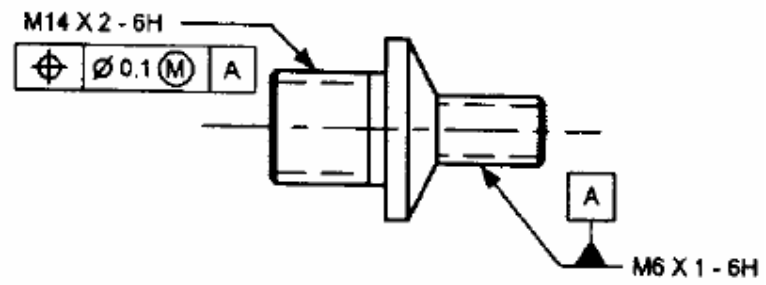


Rule #3 – RFS applies for individual tolerances and datums on all other geometric tolerances. MMC and LMC must be specified where it is required.
/规则 3 –RFS 在所有的其它的几何公差上，应用于单独的公差和基准. MMC 和 LMC 必须在需要处指定。

RULES FOR SCREW THREADS, GEARS AND SPLINES /对螺纹，齿轮和花键的规则

Each tolerance of orientation, position or datum reference for a screw thread applies to the axis of the thread derived from the pitch cylinder. Where an exception to this practice is necessary, the specific feature of the screw thread (such as MAJOR DIA or MINOR DIA) shall be stated under the feature control frame or adjacent to the datum feature symbol, as applicable.
/对于一个螺纹，每个方向，位置或基准参照公差应用到源自节圆的螺纹轴线上。当需要这个惯例有一个例外的时候，指定的螺纹（诸如大径或小径）的特征应该在公差框格下或者邻近基准特征符号处规定。

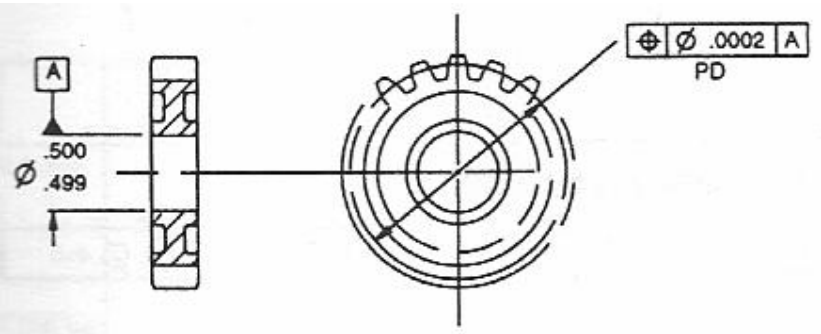
SCREW THREADS/ 螺纹



Unless otherwise specified, on screw threads, all geometric tolerances and datum references apply to the pitch diameter.
/除非特别指定，在螺纹上，所有几何公差和基准参照都应用到节圆上。

GEARS AND SPLINES /齿轮和花键

Each tolerance of orientation, position or datum reference specified for features other than screw threads, such as gears and splines, must designate the specific feature of the gear or spline to which each applies (such as MAJOR DIA, PITCH DIA, PD, or MINOR DIA)。 This information is stated under the feature control frame or under the datum feature symbol, as applicable.
/不同于螺纹，每个为特征指定的方向，位置或基准参照公差，例如齿轮和花键，必须指出应用到齿轮或花键的具体的特征（如大径，节圆 PD，或小径）。这个信息规定在公差框格的下面或基准特征符号的下面。



All geometric tolerances and datum references specified for gears and splines must designate the specific feature to which it applies.
/对于齿轮和花键，所有制定的几何公差和基准参照必须指出应用于具体的特征。

TYLOR PRINCIPLE – (RULE #1, ENVELOPE PRINCIPLE)/泰勒原则 - （规则 1 ，包络原则）

The Taylor principle is a very important concept that defines the size and form limits for an individual feature of size. The Taylor principle is widely accepted by the United States and the international standards organization (ISO). The ISO standards also allow the principle of independency which does not include form within the size limits. Depending on the particular standards invoked, either case may apply. In some cases, the circle E is used to designate features that must conform to the Taylor principle. See ISO standards ISO 8015 and ISO 1938 for additional information.
/泰勒原则是一个非常重要的概念，定义尺寸的一个个别特征的尺寸和形状界限。泰勒原则被美国和国际标准组织（ISO）广泛的接受。ISO 标准也允许不包括尺寸界限内形状的独立原则。依赖调用特别的标准，任一情形都可应用。在一些情形中，带圆圈的 E 用于指定必须和泰勒原则一致的特征。参见 ISO 标准 ISO

8015 和 ISO 1938 以获得额外的信息。

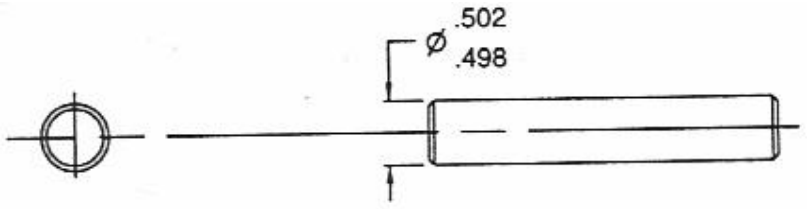
LIMITS OF SIZE /尺寸界限

Unless otherwise specified, the limits of size of a feature prescribe the extent within which variations of geometric form, as well as size, are allowed. This control applies solely to individual features of size.
/除非特别指定，一个特征的尺寸界限规定允许的几何形状和尺寸变化的程度。这个控制单独的应用到个别的尺寸特征。

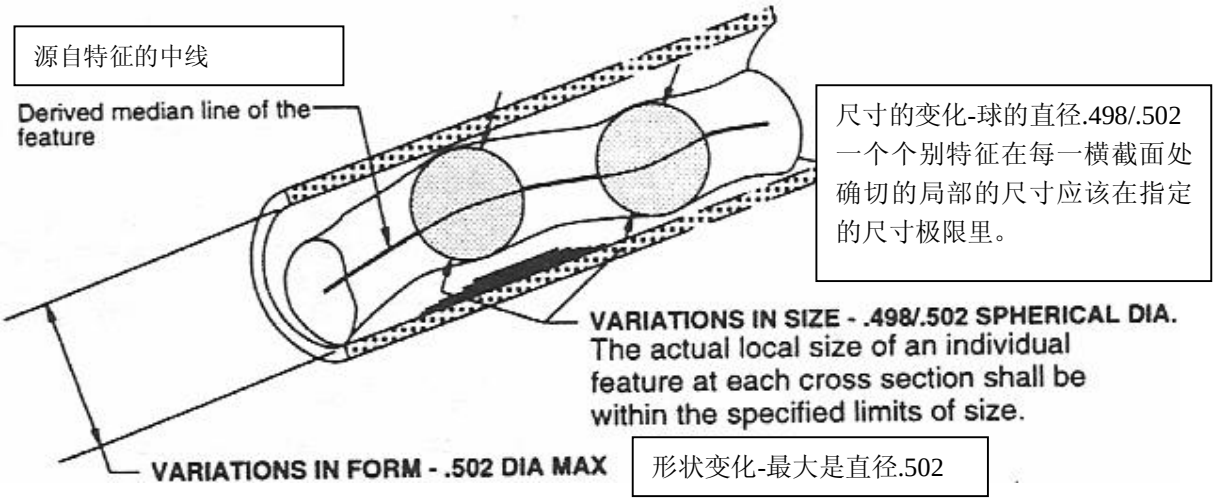
INDIVIDUAL FEATURE OF SIZE – TAYLOR PRINCIPLE (RULE #1)
/个别的尺寸特征 – 泰勒原则（规则 1）

Where only a tolerance of size is specified, the limits of size of an individual feature prescribe the extent to which variations in its geometric form, as well as its size, are allowed.
/只指定一个尺寸公差的地方，个别特征的尺寸界限规定允许的几何形状和尺寸的变化程度。

THIS ON THE DRAWING/图中



MEANS THIS /意思是这样



a. The surface or surfaces of a feature shall not extend beyond a boundary (envelope) of perfect form at MMC. This boundary is the true geometric form represented by the drawing. No variation in form is permitted if the feature is produced at its MMC limit of size.
/曲面或者一个特征的曲面不应该延伸到最大实体条件下的理想形状的边界（包络线）。这个边界是工程图体现的真正的几何形状。如果特征在尺寸界限的最大实体条件下生产，不允许形状变化。
b. Where the actual local size of a feature has departed from MMC toward LMC, a variation in form is allowed

equal to the amount of such departure.
/确切的局部特征尺寸已经偏离 MMC，而偏向 LMC 的地方，允许的形状变化等于这样偏离的数量。

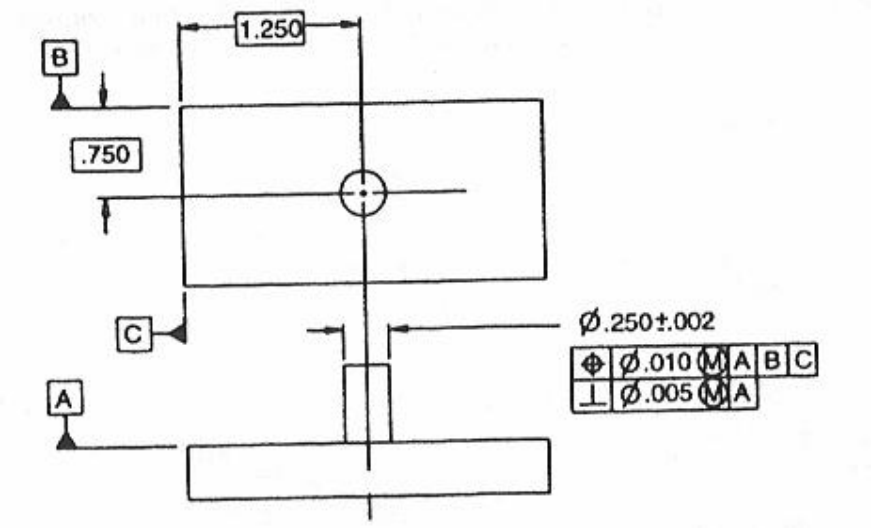
c. There is no requirement for a boundary of perfect form at LMC. Thus, a feature produced at its LMC limit of size is permitted to vary from true form to the maximum variation allowed by the boundary of perfect form at MMC.
/在最小实体条件下，没有对精确形状的边界的要求。 这样，一个在它的最小实体条件下生产的特征的尺寸界限允许从真实形状到在最大实体条件下精确形状的边界的最大允许变化的范围内变化。

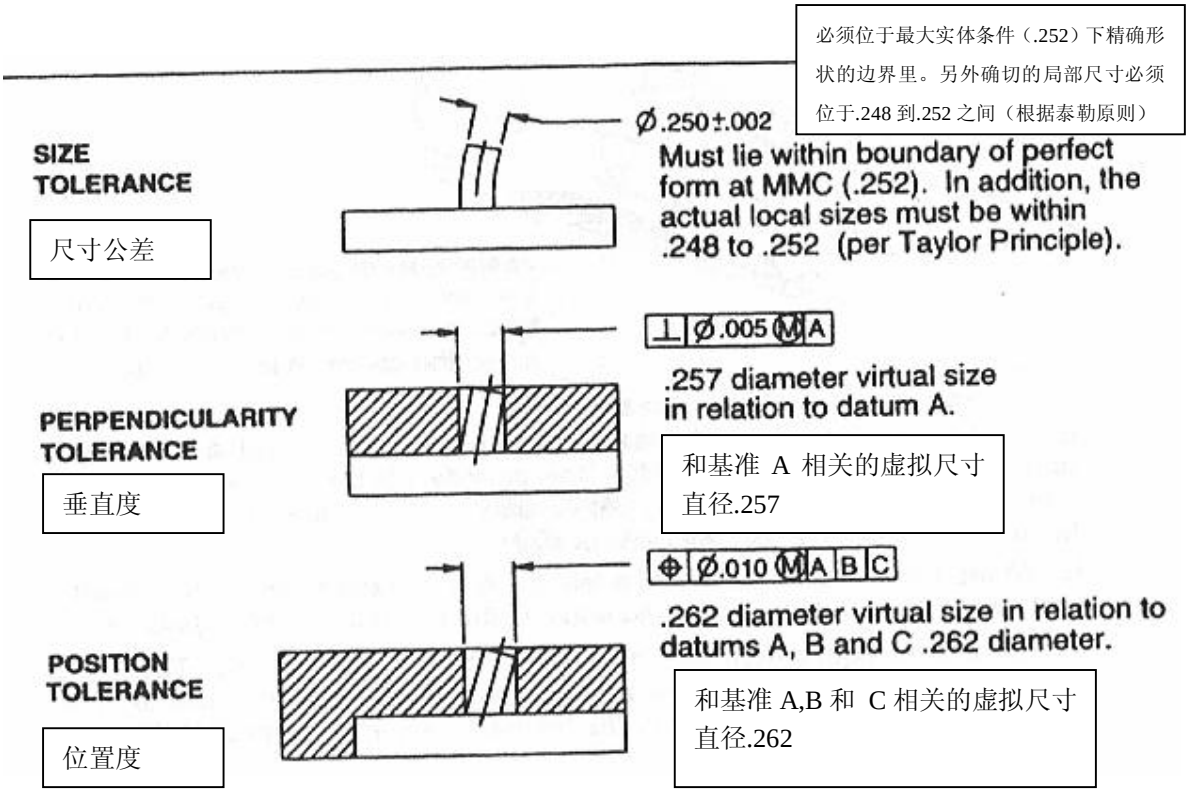
VIRTUAL CONDITION /虚拟条件

Depending upon its function, a feature may be controlled by tolerances such as size, form, orientation and location. Consideration must be given to the collective effects of these factors in determining the clearances between mating parts and in establishing gage feature sizes. The collective effect of these factors is termed virtual condition.
/ 一个特征依靠它的功能，可能被诸如尺寸，形状，方向和位置的公差所控制。在决定相配合零件之间的间隙和建立量规特征尺寸时就必须考虑这些因素的集体影响。把收集的这些因素的影响定义为虚拟条件。

Virtual condition is a constant boundary generated by the collective effects of a size feature's specified MMC or LMC material condition and the geometric tolerance for that material condition.
/虚拟条件是一个不变的边界，是由尺寸特征指定的 MMC 或 LMC 材料条件下 和那些材料条件下几何公差共同影响而生成的。

The pin in the illustration below has two virtual sizes. The .257 diameter virtual size is a result of the perpendicularity tolerance relative to datum A. the .262 diameter virtual size is a result of the position tolerance relative to datums A, B and C. the virtual sizes on this part could also be called outer boundary. It can also be viewed as a 3D solid.
/下图所示的销子有两个虚尺寸。直径 0.257 虚尺寸是一个和基准 A 相关的垂直度的结果。直径 0.262 虚尺寸是一个和基准 A,B,C 相关的位置度的结果。这个零件上的虚尺寸也能叫做外部边界。它能作为一个 3D 实体观看。



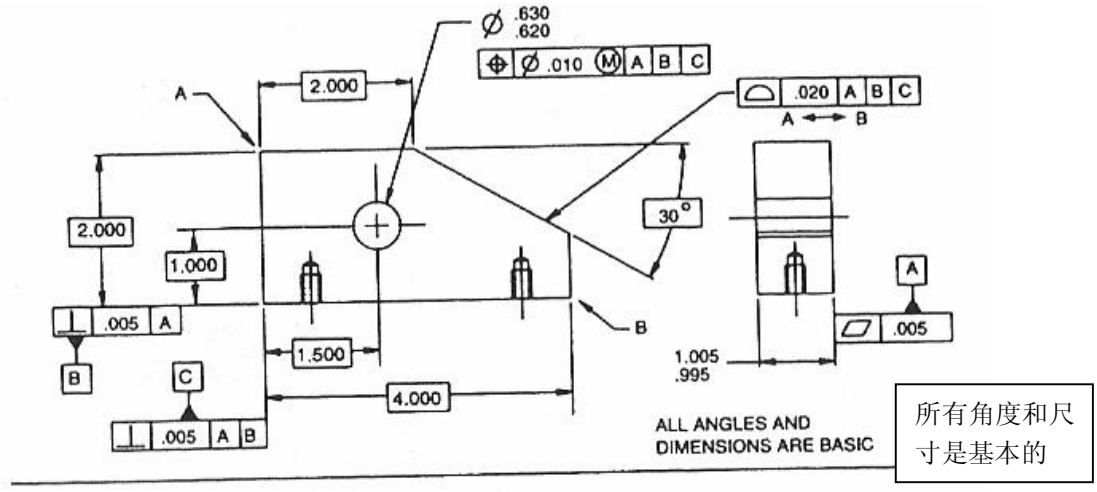


GEOMETRIC TOLERANCING APPLIED TO AN ANGLE BLOCK/应用到角度的几何公差

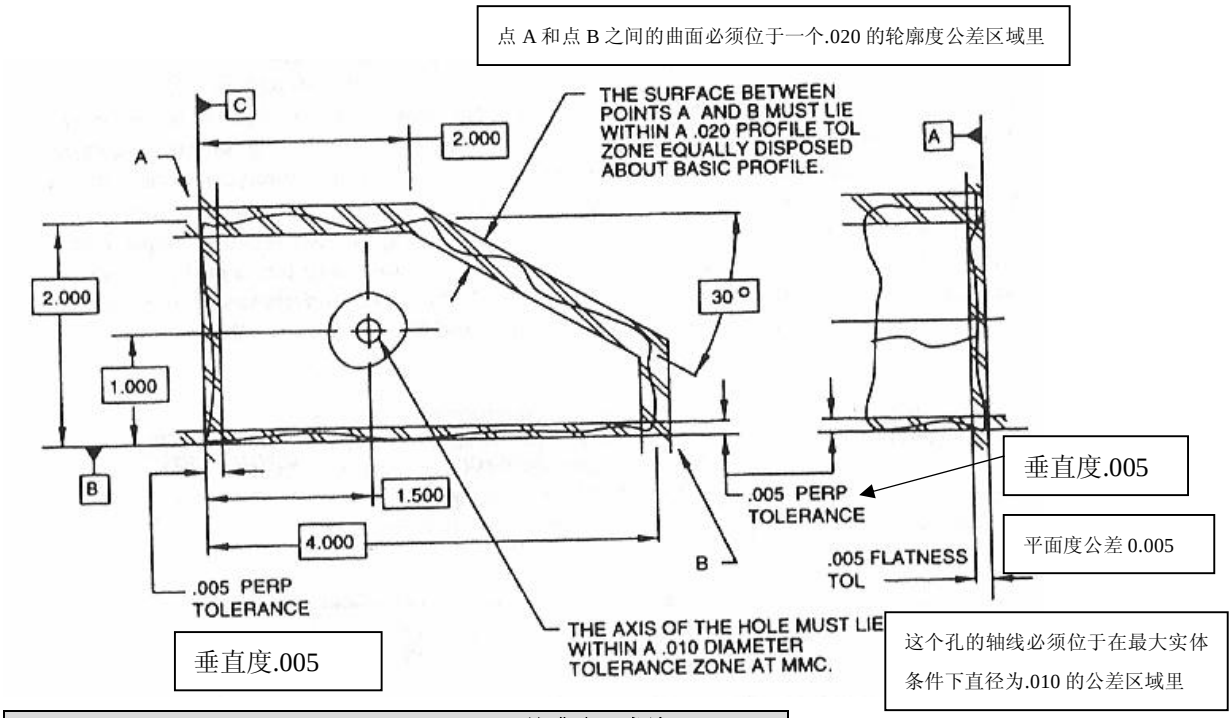
Geometric tolerancing is a very clear and concise three dimensional mathematical language for communicating product definition. A fully geometrically toleranced product drawing is shown in the top view. In the bottom view, the produced part is shown in the datum reference frame established by datum features A, B and C. the surfaces must lie within the specified tolerance zones.

/几何公差标注是一个在产品定义沟通方面非常清楚和简明的三维数学语言。一个完整的标注几何公差的产品图显示在顶视图中。在底视图中，在由基准特征 A,B,C 建立的基准参照框架里显示生产的零件。曲面必须位于指定的公差区域内。

THIS ON THE DRAWING/图中

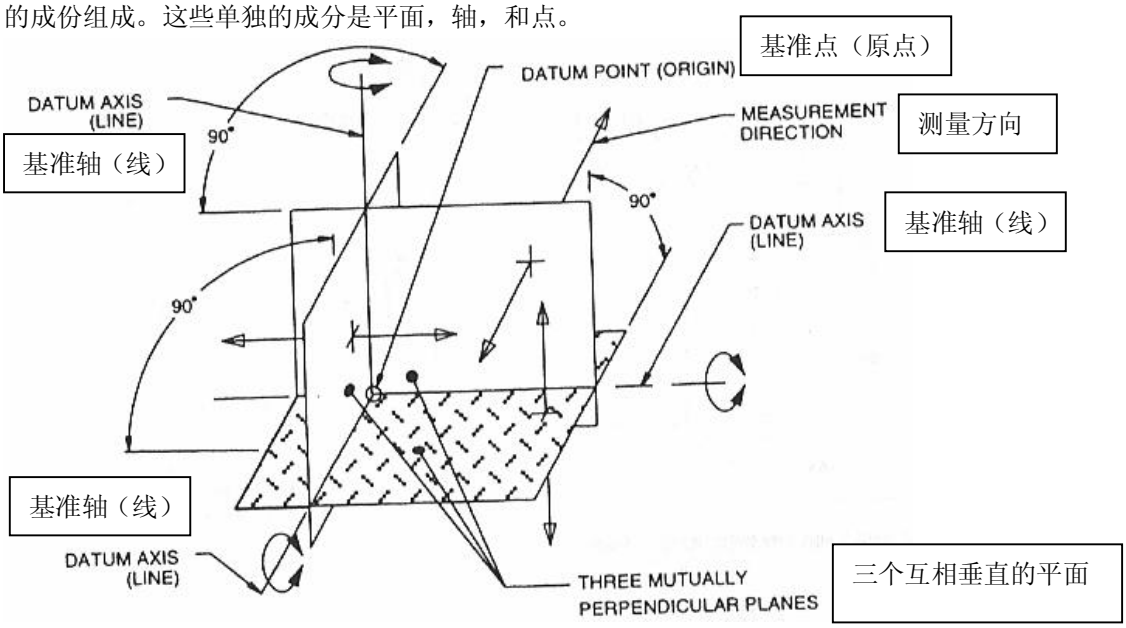


MEANS THIS ON PRODUCED PART/生产零件上的意思是



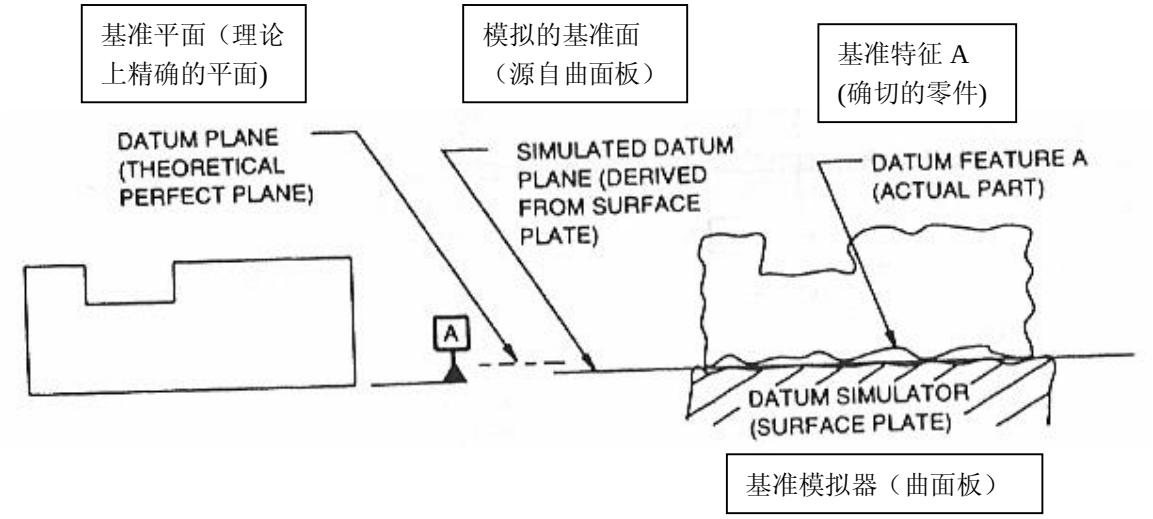
THE DATUM REFERENCE FRAME (DRF)/基准参照框架（DRF）

The datum reference frame is the 'frame of reference 'to which all the requirements are attached. In geometric tolerancing we can relate engineering, manufacturing and inspection together by using this datum reference frame. The three planes are called the primary plane, the secondary plane and the tertiary plane. The datum reference frame is made up of a series of individual components. The individual components are planes, axis, and points.
/基准参照框是所有要求要附在上面的'参照框'。在几何公差标注中，通过使用这个基准参照框，我们能使工程，加工制造和检测联系起来。这三个平面叫做第一面，第二面和第三面。基准参照框由一系列单独的成份组成。这些单独的成分是平面，轴，和点。



Datums and the datum reference frame indicate the origin of a dimensional relationship to a toleranced feature or features of a part. When a feature serves as a datum feature, its true geometric counterpart actually establishes the datum. Since measurements cannot be made from a true geometric counterpart, which is theoretical, simulated datums are assumed to exist and be simulated with our manufacturing, processing and inspection equipment such as the bed on a machine, a collet or chuck, gage pin, a surface plate, angle plate etc. measurements then originate from the simulated planes or axis that the manufacturing or inspection equipment simulates and not the features themselves.

/基准和基准参照框显示标注公差的特征或零件特征的空间关系的来源。当一个特征用作基准特征时，它的真正的几何副本实际上建立了基准。既然不能从真正的几何副本量得，理论上，假设模拟的基准存在并且模拟成我们的机械加工，工艺过程和诸如机床台面，夹头或卡盘，量规，平面，角板等的检测设备。那么，可以从模拟的平面或轴（加工或检测设备模拟的而且不是特征本身）为起点测量。



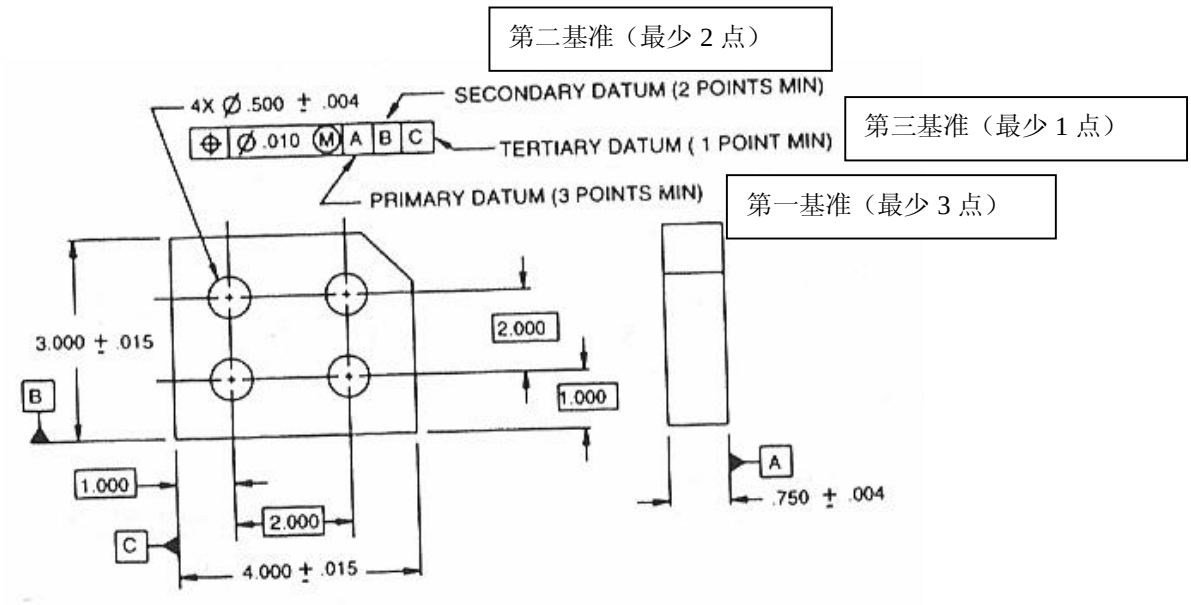
Datum plane A is established by the datum features surface contacting on the high points of a simulated perfect plane。 Measurements originate from the plane and not the part.

/基准 A 是通过和模拟的理想平面以高点接触的基准特征曲面建立的。从这个平面，而不是从零件测量。

SPECIFIED DATUMS CLEARLY DEFINE DESIGN INTENT/指定的基准明显定义设计意图

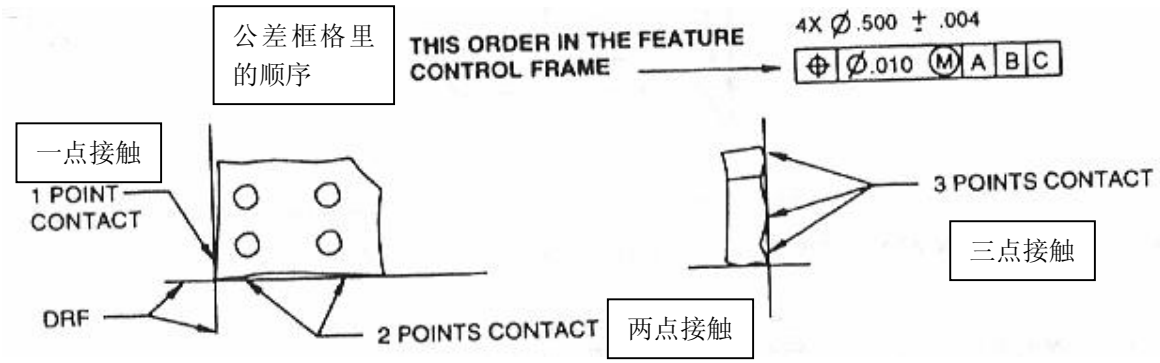
The part below has four holes located in relation to the datums. The requirements for the holes are shown in the feature control frame along with the specified datums. The datums identify how the part is located in the datum reference frame.

/下面的零件有四个孔，位置和基准相关。在公差框格连同指定的基准里显示出对孔的要求。基准识别零件怎样定位于基准参照框架里。



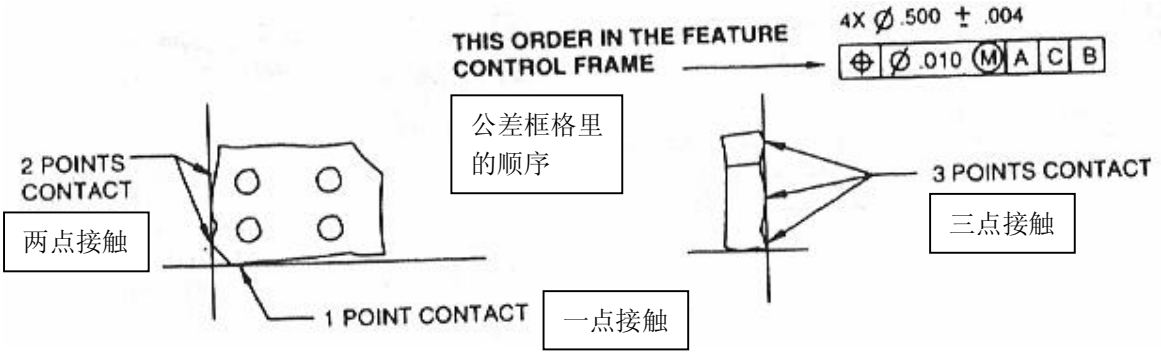
The order of the datums in the feature control frame specifies the order in which to load the imperfect part in the perfect datum reference frame. We then measure from the DRF and not the part.
/公差框格里基准的顺序指定在理想的基准参照框里加载给未完成的零件的顺序。然后我们从 DRF（基准参照框架），而不是从零件测量。

MEANS THIS /意思是这样



If the order of the datums in the feature control frame is changed, it will change the order in which the part is loaded in the feature control frame.
/如果公差框格里的基准顺序改变了，将改变零件加载在公差框架里的顺序。

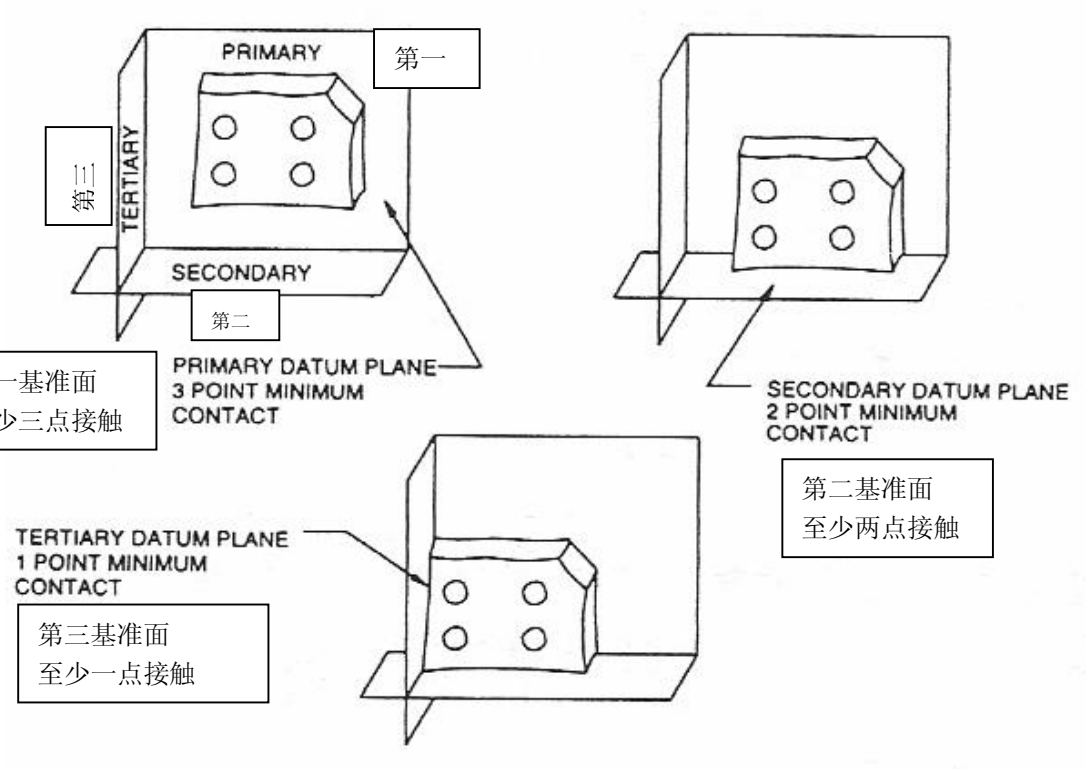
MEANS THIS /意思是这样



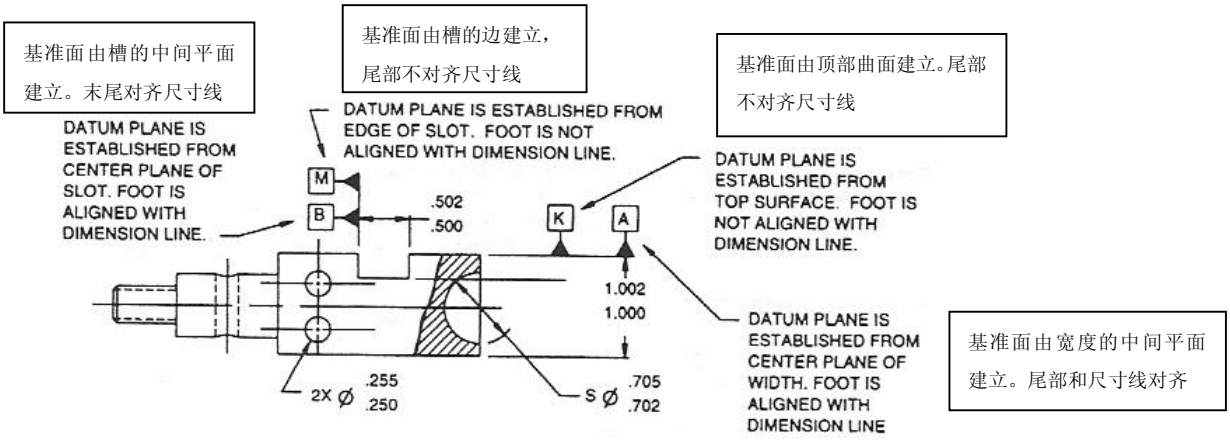
DATUM REFERENCE FRAME /基准参照框架

In geometric tolerancing we can relate engineering, manufacturing and quality together by placing parts in a datum reference frame. Each unsupported object or part has six degrees of freedom. The part must be fixed in relation to this datum framework in order to stop this freedom. The part is related to the primary datum plane by contacting 3 points. It is related to the secondary datum by contacting a minimum of two points. It is then related to the tertiary datum by contacting a minimum of one point.

/在几何公差标注中，通过把零件放在一个基准参照框架里，我们能和工程学，加工学和质量联系起来。每一个无支持的物体或零件有六个自由度。为了阻止这个自由度，这个零件必须固定并和这个基准框架相关。通过三点接触，这个零件和第一基准面发生关系。通过至少两点接触和第二基准发生关系。然后通过至少一点接触和第三基准发生关系。

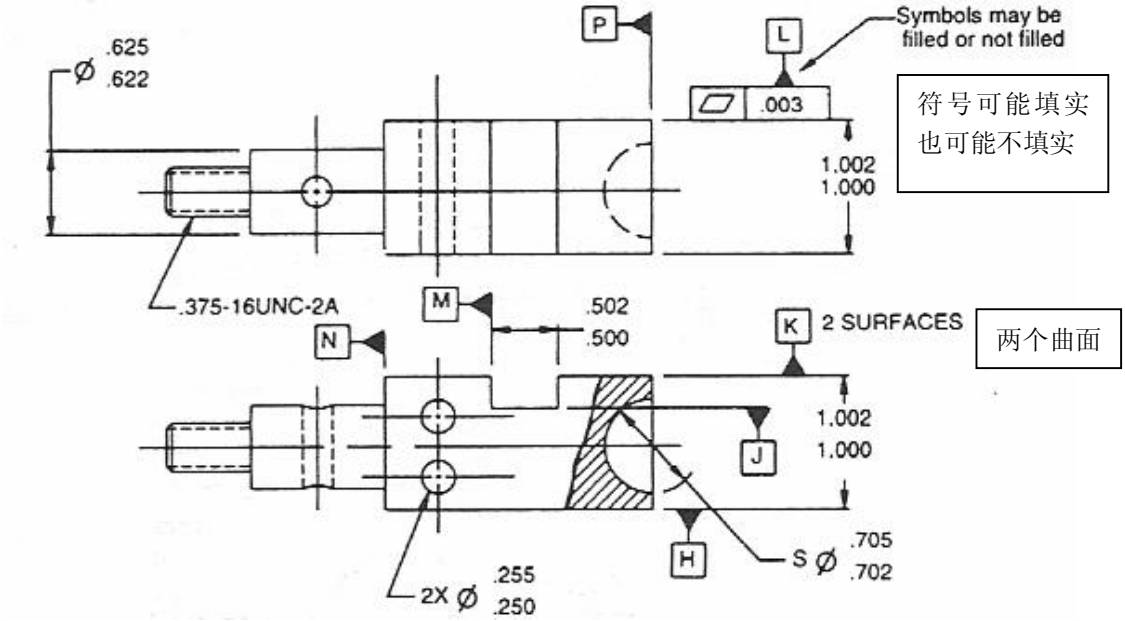


PLACEMENT OF DATUM FEATURE SYMBOL IS VERY IMPORTANT
/基准特征符号的布置是非常重要的

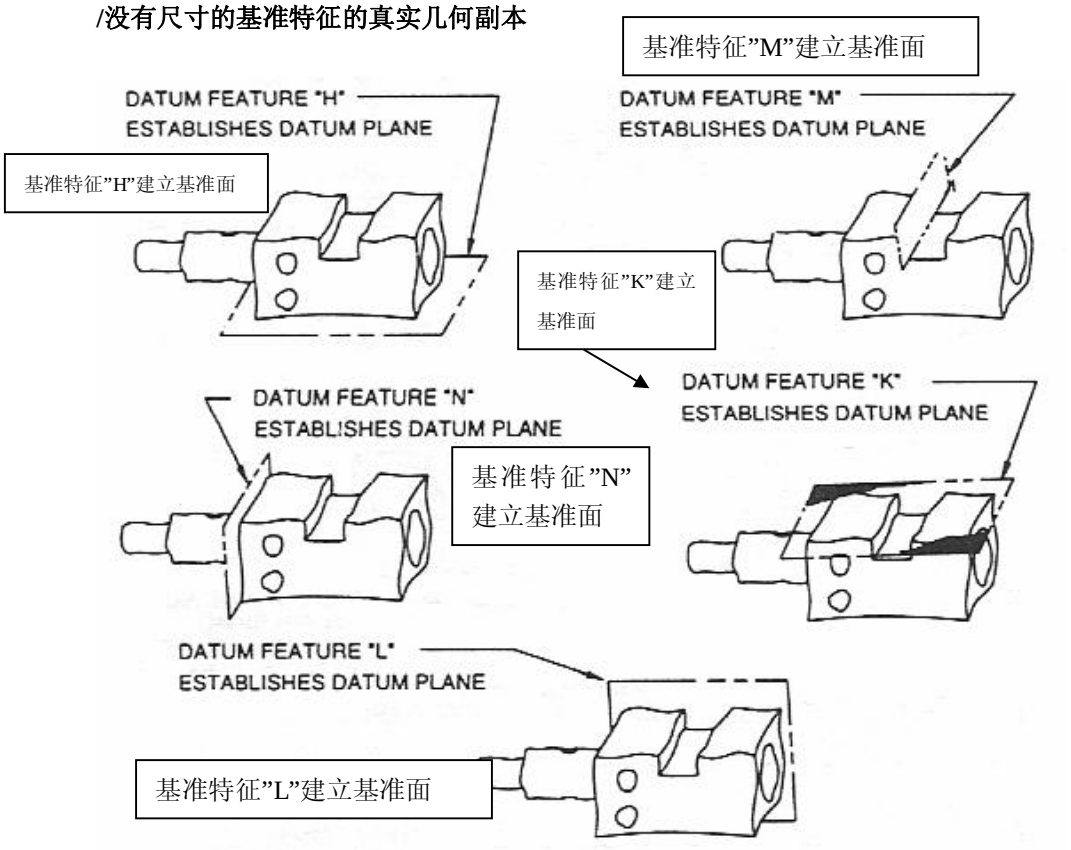


DATUM FEATURES WITHOUT SIZE/没有尺寸大小的基准特征

If the datum feature is a plane surface, the datum feature symbol is attached to the surface or an extension line of the surface as shown below. This will establish a datum plane.
/如果基准特征是一个平曲面，基准特征符号附在这个平曲面上或下图示的平曲面的延伸线上。这将建立一个基准面。

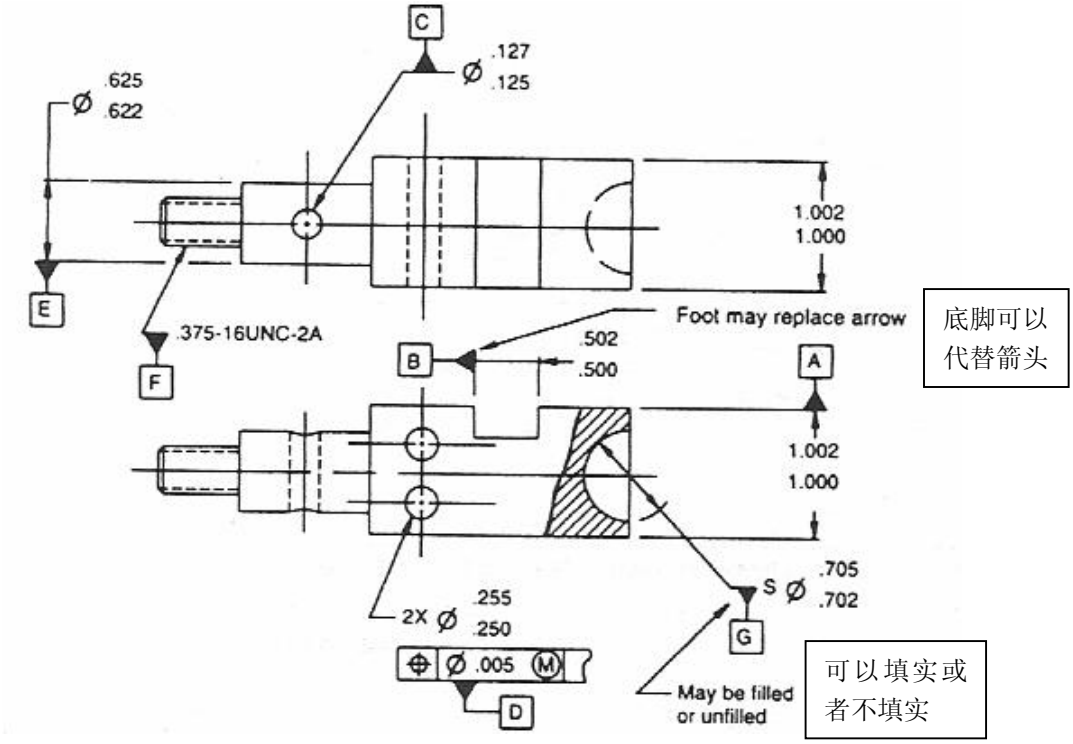


TRUE GEOMETRIC COUNTERPART OF DATUM FEATURES WITHOUT SIZE /没有尺寸的基准特征的真实几何副本

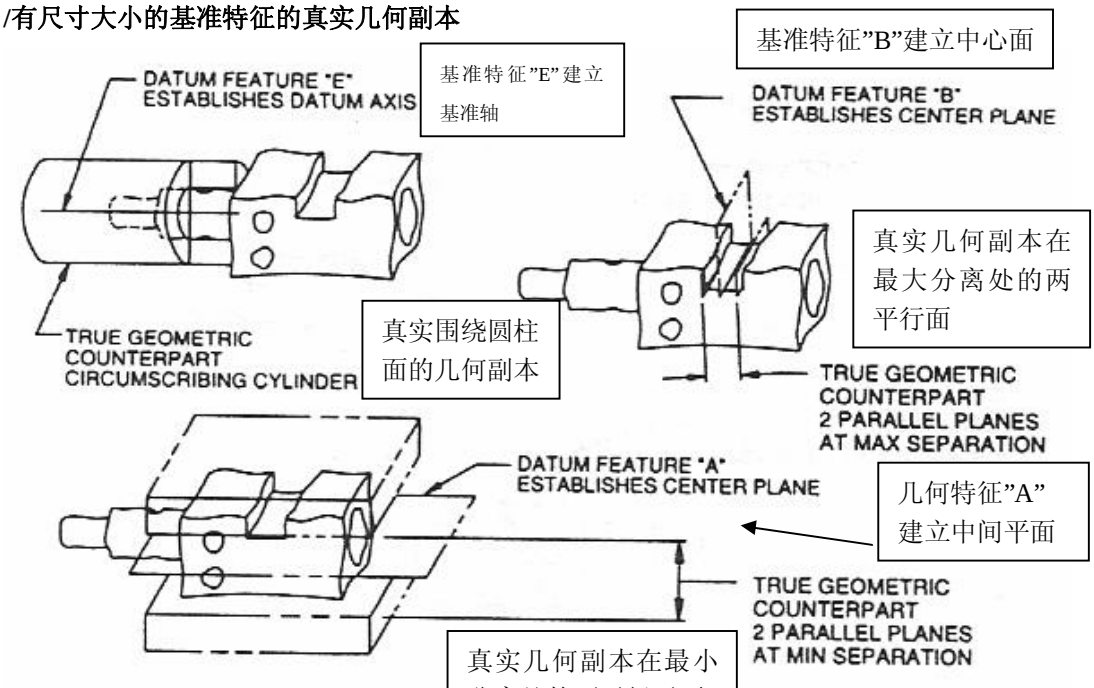


DATUM FEATURES WITH SIZE /带尺寸的基准特征

If the datum feature is a feature of size, the datum feature symbol is attached to or associated with the size dimension. By attaching or associating the datum feature symbol with a particular size dimension, it defines a specific point, axis or median plane that is derived from that feature as the datum.
/如果基准特征是一个尺寸特征，基准特征符号附在尺寸后或和尺寸相关联。通过和带有一个特定尺寸的基准特征符号相依附或相关联，定义一个源自那个特征的明确的点，轴或中间平面作为基准。



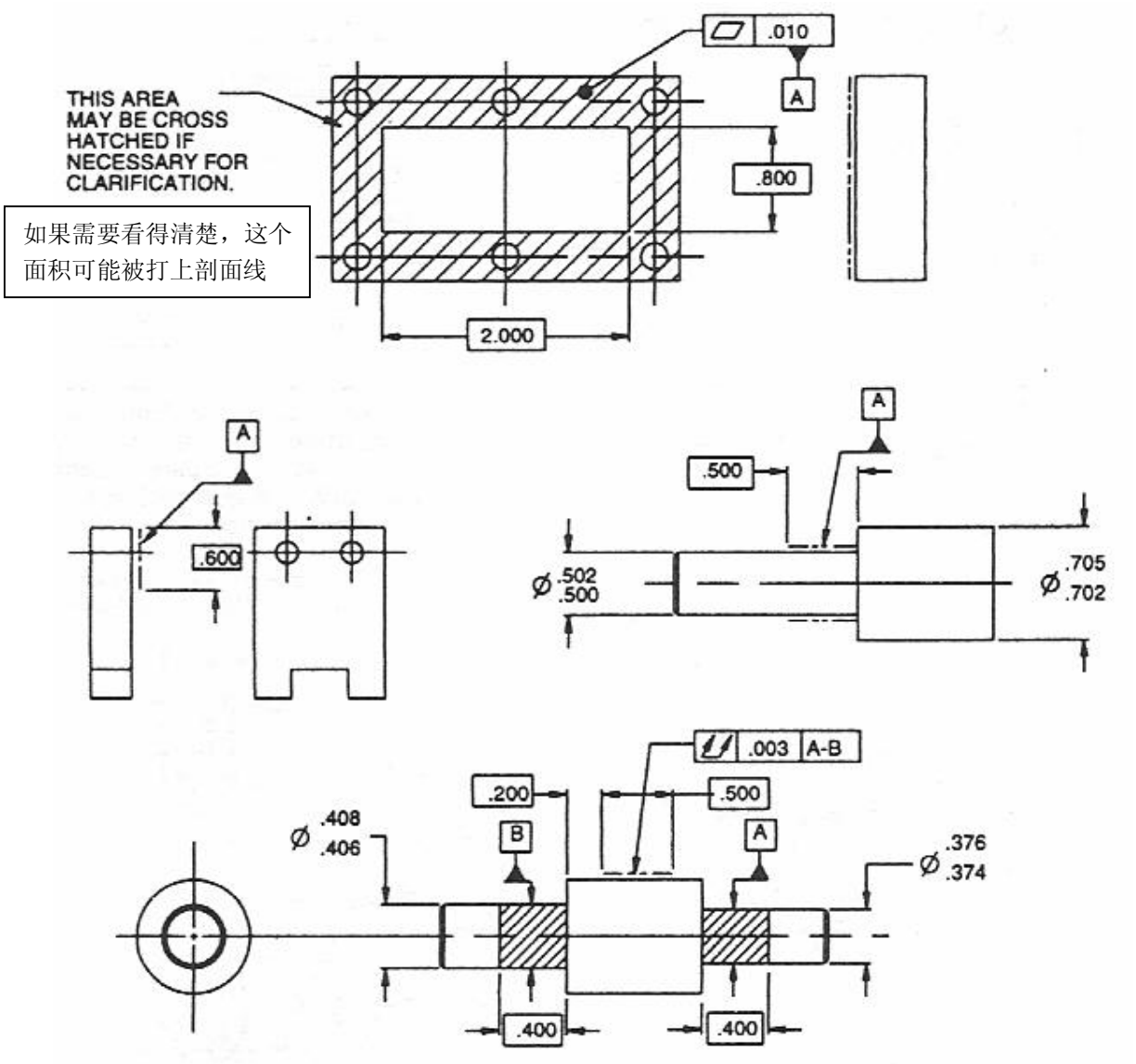
TRUE GEOMETRIC COUNTERPART OF DATUM FEATURES WITH SIZE
/有尺寸大小的基准特征的真实几何副本



PARTIAL DATUM FEATURES/局部基准特征

In some cases only a particular area of a feature will serve as a datum feature. This may be specified by the use of a thick chain line drawn parallel to the surface profile and dimensioned as to the required length and width. If the area needs clarification, it may by cross hatched. Partial datum may also be specified by means of a note or datum target. See examples below.

/在一些情况下，只是一个特别的特征区域将作为一个基准特征。这可能用画一个平行于曲面轮廓的一粗双点划线来指定，并且标上尺寸以显示所需要的长度和宽度。如果这个区域需要看得更清楚，可能会打上剖面线。局部基准也可能通过一个注释或基准目标来指定。参见下面的例子。



Where a datum is established by two datum features (two slots, two diameters etc.), both datum reference letters are entered in a single datum compartment in the feature control frame and separated by a dash.

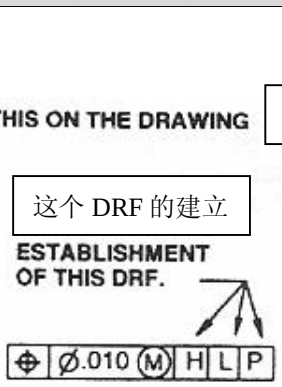
/在一个基准通过两个基准特征（两个槽，两个直径等等）创建的地方，两个基准参照字母加入到一个特征控制框里单一的基准间隔间里，并且用一个短线分开。

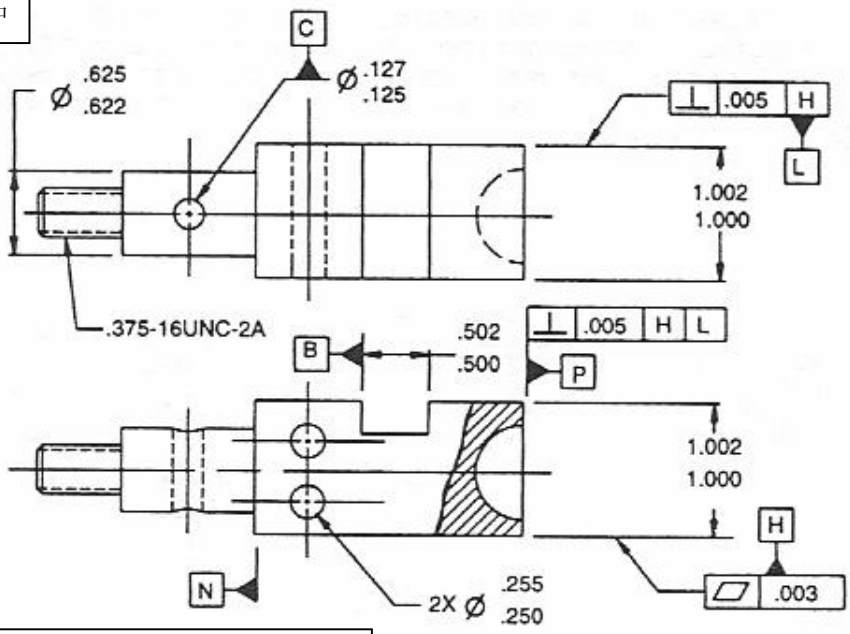
ESTABLISHING A DRF AND QUALIFYING DATUM FEATURES – PLANE SURFACES /建立一个 DRF 和限定基准特征-平曲面

THIS ON THE DRAWING 图中

这个 DRF 的建立

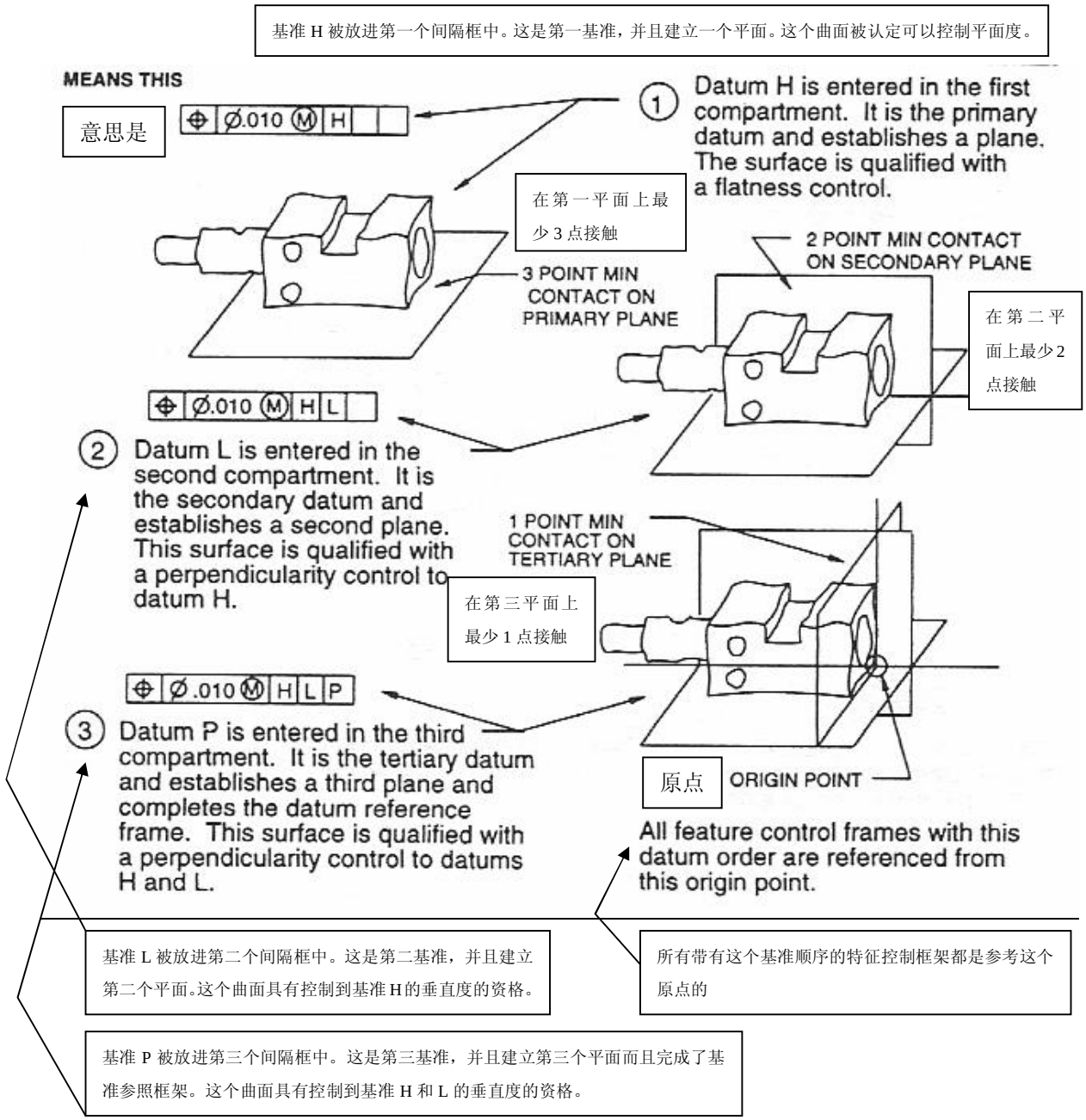
ESTABLISHMENT OF THIS DRF.





The qualification of the datum features with perpendicularity and flatness control the unstability of the imperfect part in the perfect DRF and, ultimately, in the functional assembly.

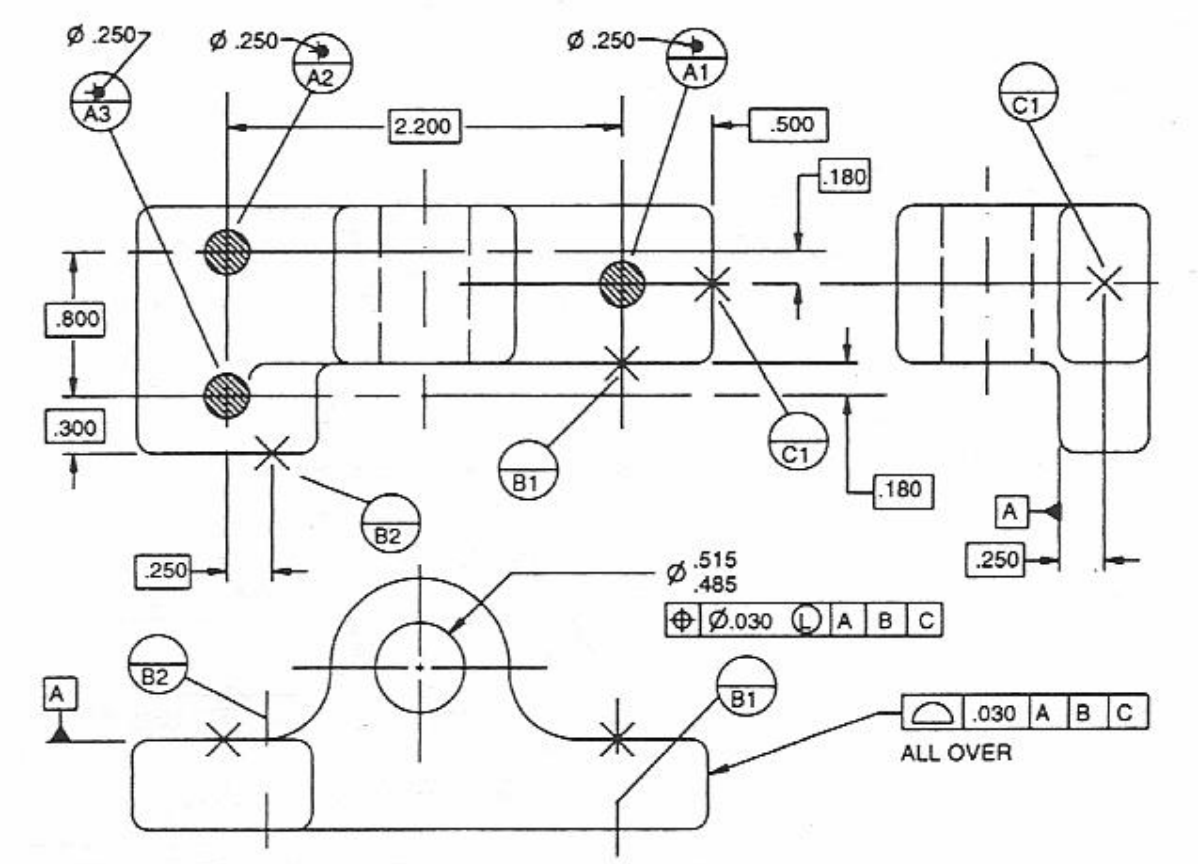
带有垂直度和平面度的基准特征的限定，控制在理想的 DRF 中和最终在功能装配中有缺点的零件的不稳定性。



DATUM TARGETS/基准对象

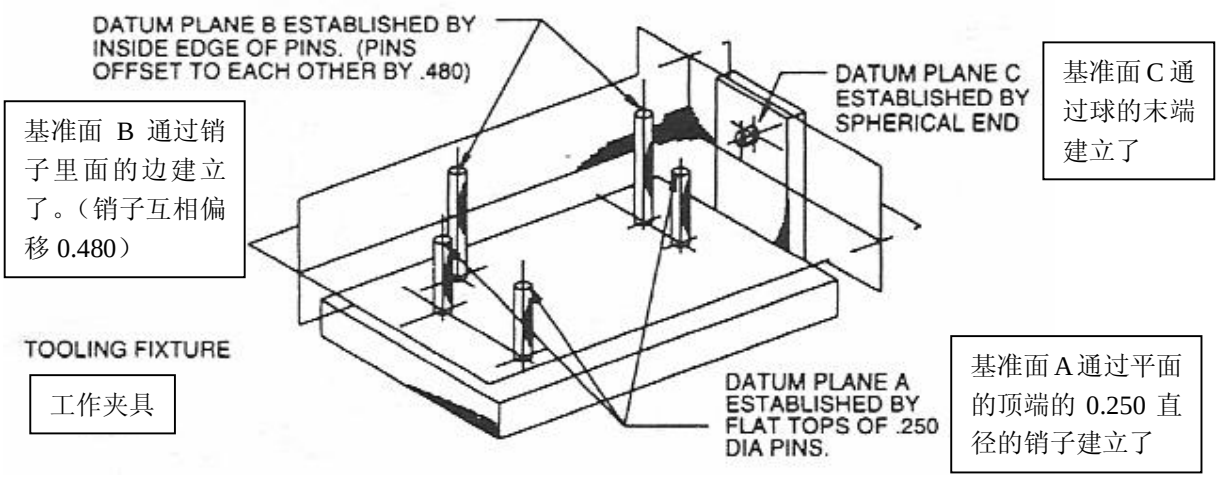
Datum targets may be used to establish a datum reference frame. This occurs because of manufacturing or fixturing concerns or part surface irregularities. Examples of this application might be castings, forgings, sheet metal parts, plastic parts and weldments.

/基准对象可以用于建立一个基准参照框。这个因为加工或涉及到的工作夹具或者零件曲面的不规则而发生。这个应用的例子可能是铸件，锻件，钣金件，塑料件和焊接件。



This is an example of a verification or set – up fixture illustrating the datum target concept. The datum reference frame is established by three target areas on the primary feature A, two target lines on datum B, and one target point on datum feature C.

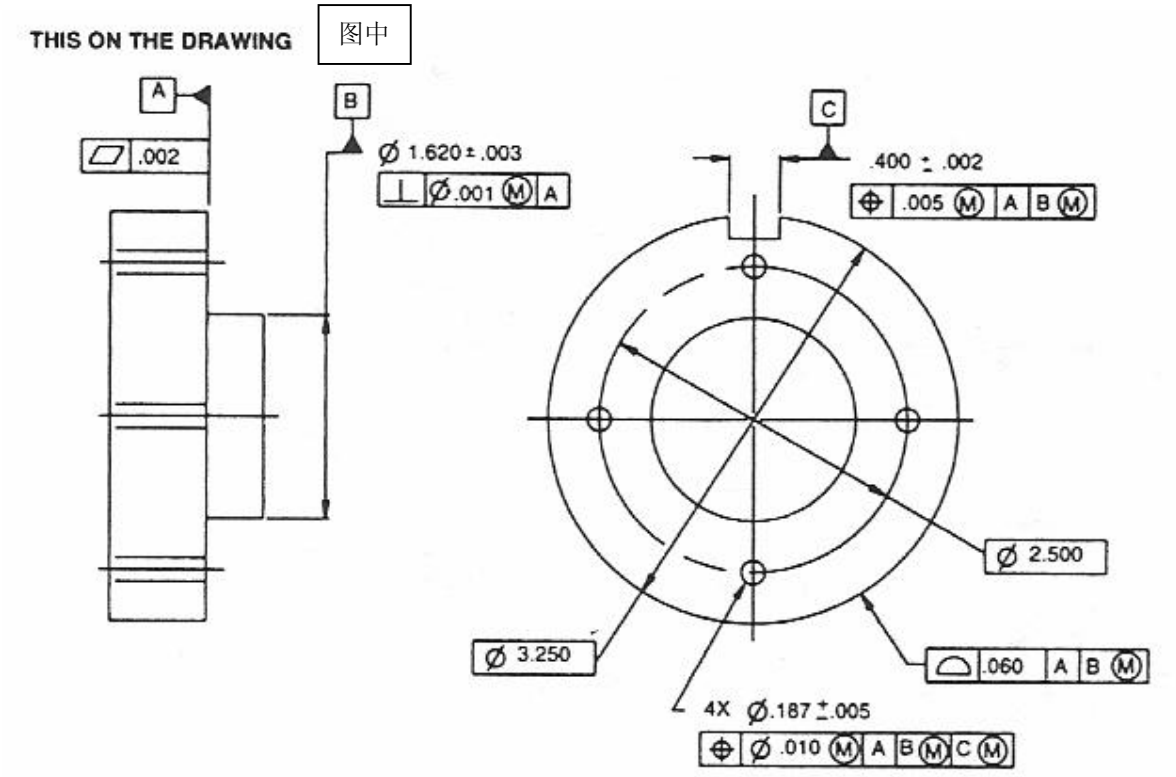
/这是一个例子，来确认或者建立图示基准对象概念的工作夹具。基准参照框通过三个目标区域建立，这三个目标区域是在第一特征 A,在基准 B 上的两条目标线，和一个在基准特征 C 上的目标点。



ROUND PART WITH SLOT /带有槽的圆形件

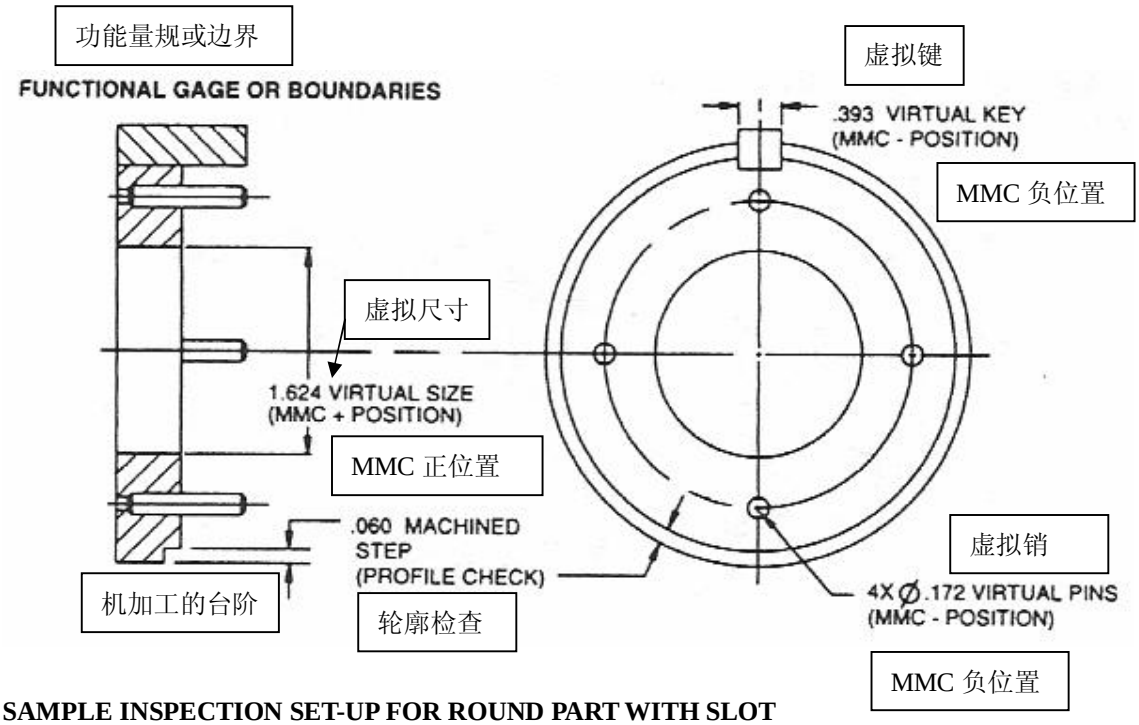
The part below mounts on the right hand face, the pilot and the key. These features are identified as datum features and are related to each other by the perpendicularity and position tolerances. The holes and outside contour have been related to the datum features by the position and profile tolerances. The .060 profile tolerance on the outside contour controls size, form, location and orientation.

/下面的零件以右边的面，引导销和键装配，。这些特征被看成基准特征，而且通过垂直度和位置度公差互相相关。孔和外面的轮廓已经通过位置度和轮廓度公差和基准特征相关。在外面轮廓上的 0.060 轮廓度公差控制尺寸，形状，位置和方向。

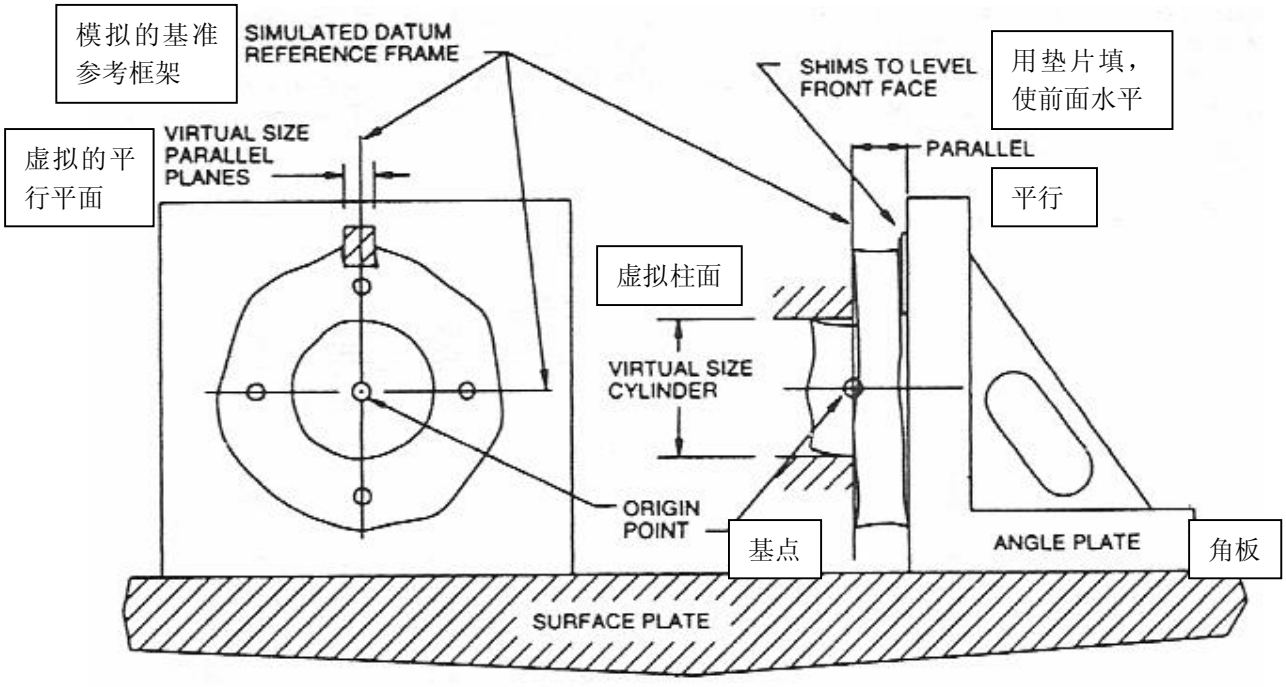


The functional gage or boundaries shown below reflect the geometric controls above. Since the features and datums are all applied at MMC, all the features may shift and/or displace as long as they meet the requirements set forth below.

/下面显示的功能量规或边界反映上面的几何控制。既然特征和基准都应用 MMC 条件，只要它们满足设定的要求，所有的特征可以转移或取代。



SAMPLE INSPECTION SET-UP FOR ROUND PART WITH SLOT
/带槽的圆形零件的样件检测设定



The illustration above is a sample open set-up inspection procedure for the verification of the round part in the previous example. Notice that the front face, datum feature A, is leveled or balanced with shims to make it parallel with the angle plate. The virtual size of the pilot is centered and the part is anti-rotated with the slot to set up the datum reference frame. All measurements are made from the origin point.
/以上图示是对于先前的例子中的圆形零件的确认的检测程序的例子。注意，前面（基准 A）用薄垫片调平，使它和角板平行。引导的虚拟尺寸居中，而且零件反转设定基准参照框架。所有尺寸测量全从基点开始。

This part can also be verified with a coordinate measuring machine (CMM). The front face is leveled, the origin is set at the pilot and the part is oriented by the slot.
/这个零件也能用坐标测量机（CMM）验证。前面被调平，基点设定在引导处，零件用凹槽定向。

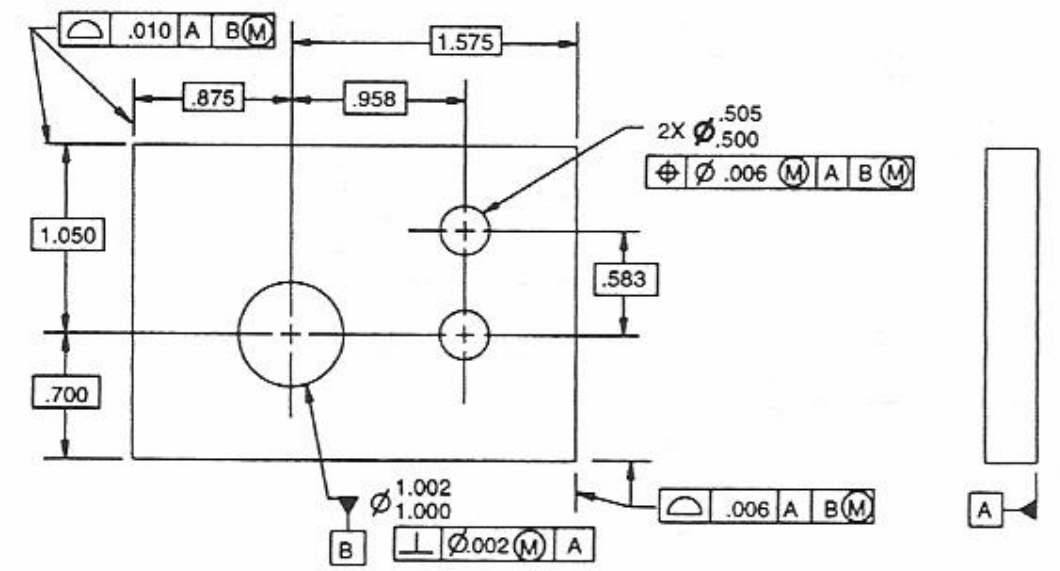
Since the MMC modifier is referenced on the features and datum features, additional tolerance is available as the datum features and features depart from virtual size.
/既然 MMC 修正符在特征和基准特征里被引用，基准特征和特征偏离虚拟尺寸可获得额外的公差。

The geometric tolerancing defines the product without defining the measurement procedure. All measurement procedures have associated uncertainty or risk. Some procedures will have more risk than others. This is called methods divergence. This part can also be verified with calipers to provide a rough check. The type of verification procedure is defined in the measurement plan. The measurement plan is designed by the quality engineer with knowledge of the manufacturing process.
/几何公差标注定义了产品，而没有定义测量程序。所有的测量程序全和不确定因素或风险相关联。一些程序将比其它的有更大的冒险。这叫做方法分歧。这个零件也能用卡尺验证，提供一个粗略的测量。验证程序的类型在测量计划里定义。测量计划是被有机械加工工艺知识的质量工程师设计的。

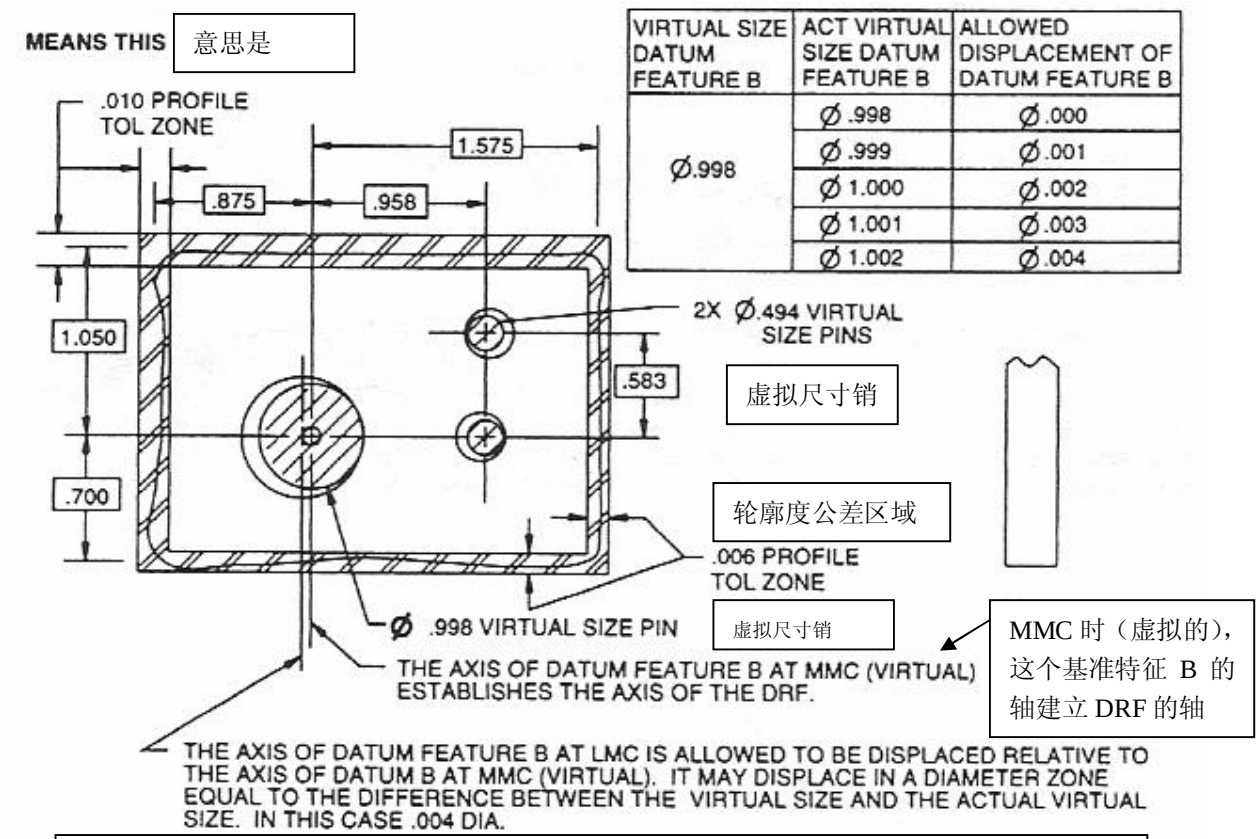
DATUM FEATURES OF SIZE – EFFECT OF MODIFIERS/尺寸的基准特征-修正符的影响

A group of features may be controlled relative to a datum feature at MMC as shown below. Datum feature B at MMC establishes the location of the axis of the datum reference frame (DRF) for the location of all the features. As datum feature B departs from MMC, its axis may be displaced relative to the datum B at MMC axis in a diameter zone equal to the difference between the virtual size and the actual virtual size.
/一组特征可以被控制，相关于在下图示的最大实体条件时的基准特征。为确定所有特征的位置，最大实体条件的基准特征 B 建立了基准参照框（DRF）的轴的位置。因为基准特征 B 偏离最大实体条件，它的轴可以被相关于最大实体条件时的基准 B 的轴置换，在直径区域里等于虚拟尺寸和实际虚拟尺寸之间的差异。

THIS ON THE DRAWING /图中



虚拟尺寸基准特征 B	确切虚拟尺寸基准特征 B	允许的基准特征 B 的移置

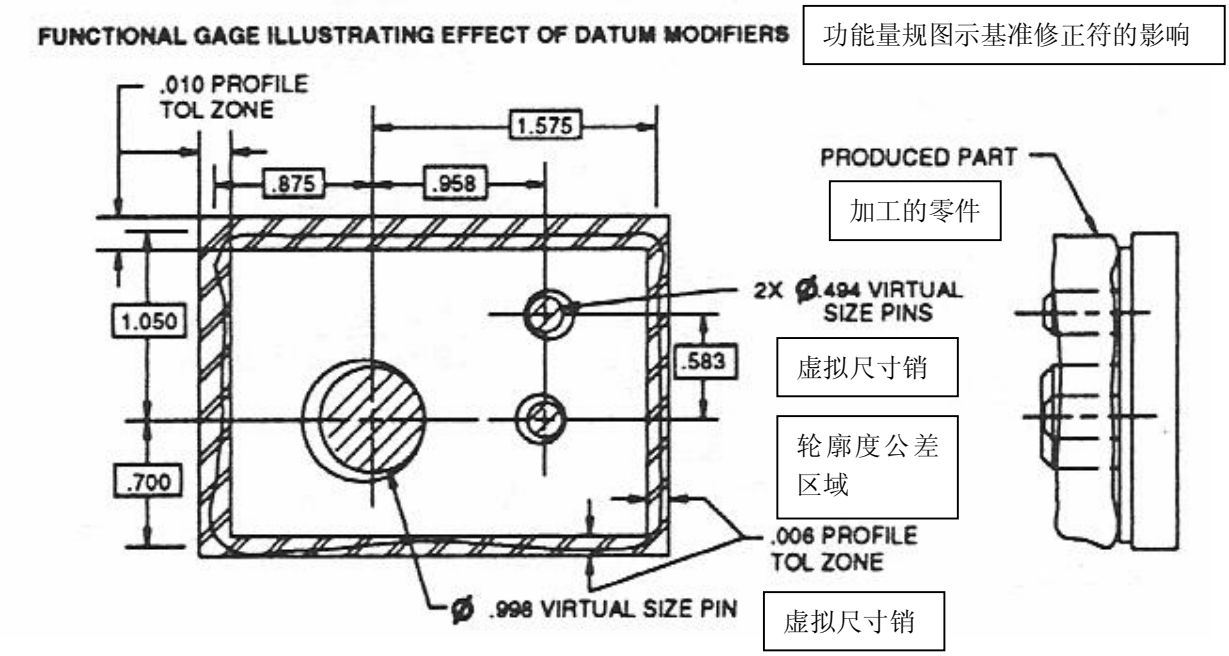


VIRTUAL SIZE DATUM FEATURE B	ACT VIRTUAL SIZE DATUM FEATURE B	ALLOWED DISPLACEMENT OF DATUM FEATURE B
Ø.998	Ø .998	Ø .000
	Ø .999	Ø .001
	Ø 1.000	Ø .002
	Ø 1.001	Ø .003
	Ø 1.002	Ø .004

最小实体条件时的基准特征 B 的轴允许被相关的最大实体条件的基准 B 的轴（虚拟的）代替。它可以代替在直径区域中等于虚拟尺寸和实际尺寸之间的差额。在这种情况下是直径 0.004。

The effect of a datum modifier is accommodated automatically in a functional gage as shown below. The datum feature and the related features must fall within the appropriate tolerance zone. As the datum feature and related features depart from their stated material condition they may displace and shift as long as they clear the virtual size pins or remains within the profile tolerance zones. Notice that the datum modifier has no effect on the relationship between the features. Unless otherwise stated there is an implied simultaneous requirement between all the features and the datum features.

/基准修正符的影响在下图示的功能量规里自动调和。基准特征和相关的特征必须属于适当的公差区域里。因为基准特征和相关的特征偏离它们的材料条件，在它们清除虚拟的尺寸销或轮廓公差区域里的残余时，它们可能代替并轮换。注意，基准修正符对于特征之间的关系没有影响。除非声明在所有特征和基准特征之间有一暗指的同时发生的要求。



If the produced part were to be evaluated using a CMM or open set-up inspection techniques the shift of the datum feature may also have to taken into account. If inspection zeroed in on the datum feature and all the related features checked good the part would pass inspection. If the part checked bad the zero or origin could be reset in a diameter zone equal to the departure of the feature from its virtual size to its actual virtual size.
/如果生产的零件用坐标测量机或开式设置检测技术评估，基准特征的移位可能必须被考虑。如果对准基准特征检测并且所有相关的特征检测没问题， 这个零件会通过检测。 如果零件检测不好，零点或基点应该在直径区域等于特征虚拟尺寸偏离实际尺寸情况下重新设定。

It is often explained that the features shift as a group, but in actuality it can be seen that the actual datum features does the shifting or displacement relative to the group of related features. The datum shift can be easily calculated with the paper gage concept or accommodated with functional gaging techniques.
/经常有特征成组移位的解释，但是在实际中，能看到实际的基准特征移位和代替相关的特征组。基准移位用纸规概念或适合功能量规技术，可能容易计算。

DATUM FEATURES AT VIRTUAL CONDITION/虚拟条件的基准特征

A virtual condition exists for a datum feature of size where its axis or center plane is controlled be a geometric tolerance. In such cases, the datum feature applies at its virtual condition even though it is referenced at MMC or LMC. Where a virtual condition equal to the maximum material condition or least material condition is required, a zero tolerance at MMC or LMC is specified.
/一个虚拟条件因为一个基准特征而存在，它的轴或中心平面被几何公差控制，在这种情况下，尽管基准特征引用 MMC 或 LMC, 也应用虚拟条件。在要求虚拟条件等于最大实体条件或最小实体条件的地方，在 MMC 或 LMC 时指定零公差。

SIMULTANEOUS REQUIREMENT/同时发生的要求

The simultaneous requirement concept applies to both position and profile specifications. It does not apply to the lower segment in a composite tolerance.

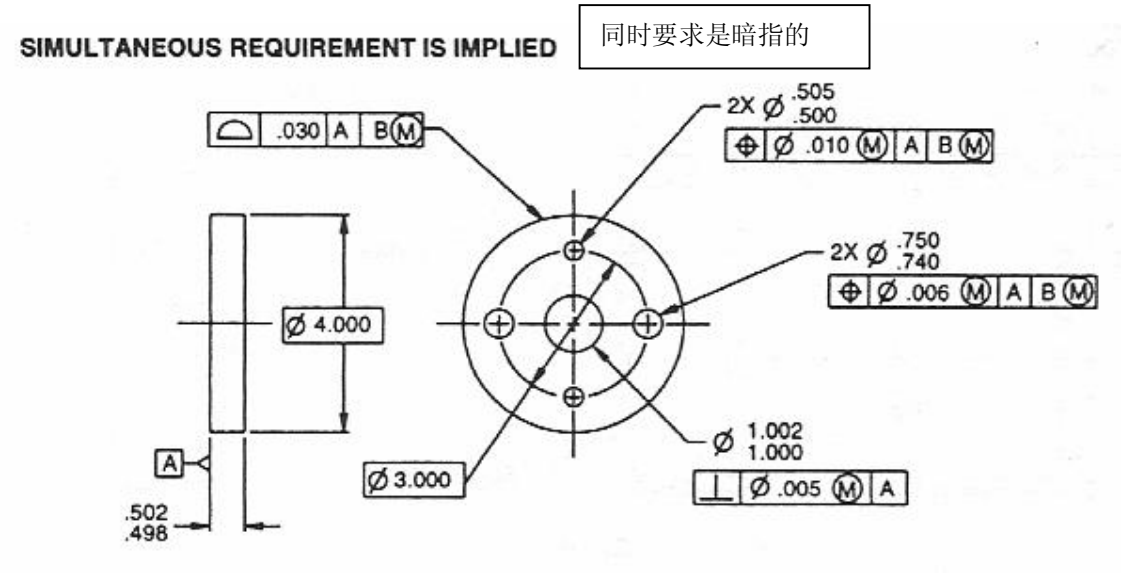
/同时发生的要求概念应用到位置度和轮廓度规范。不应用到复合公差里下面一段。

Multiple patterns of features, located by basic dimensions from common datum features of size, are considered a single composite pattern if their respective feature control frames contain the same order of precedence with the same material condition modifiers.

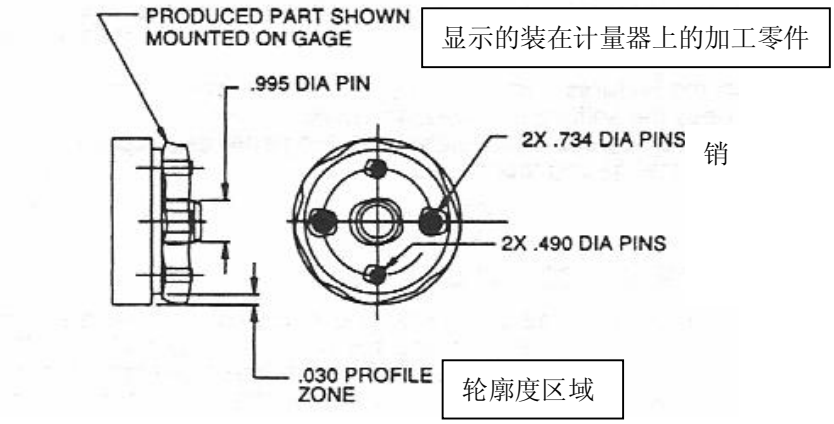
/特征的复式图形，由从尺寸的公共基准特征基本尺寸定位的，如果它们的各自的特征控制框包含同一材料条件修正符的优先顺序，被看作是一个单独的复合图形。

If such an interrelationship is not required between patterns of features, the notation SEP REQT is placed under the feature control frame. This allows each pattern of features to shift and/or rotate independently about the established datum reference frame.

/如果这样的干扰关系没有在特征式样之间要求，符号 SEP REQT 被置于特征控制框下。这允许每一特征式样关于建立的基准参照框独立移位或旋转。



FUNCTIONAL GAGE OR 3D BUNDARIES ILLUSTRATING VIRTUAL SIZES FOR SIMULTANEOUS REQUIREMENT/功能量规或 3D 边界图示同时要求的虚拟尺寸



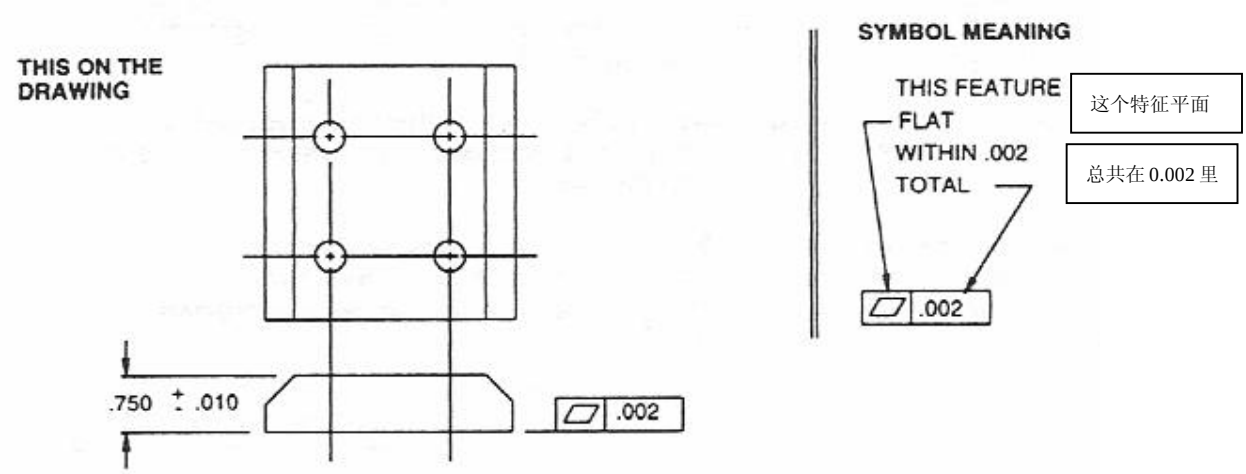
The sample functional gage shown above will verify all the features at once. The gage is for illustration only, as other methods could be used as well. In open set-up procedures, care should be taken when datum shift or feature rotation is factored into the acceptance criteria. Remember, datum shift and rotation must be factored to all the features as a group, rather than allowing each pattern of features to shift independently as the datum feature departs from MMC.

/上图示的样件功能量规将马上验证所有特征。量规仅仅为了图示，因为其它方法也能用。在开式设定程序

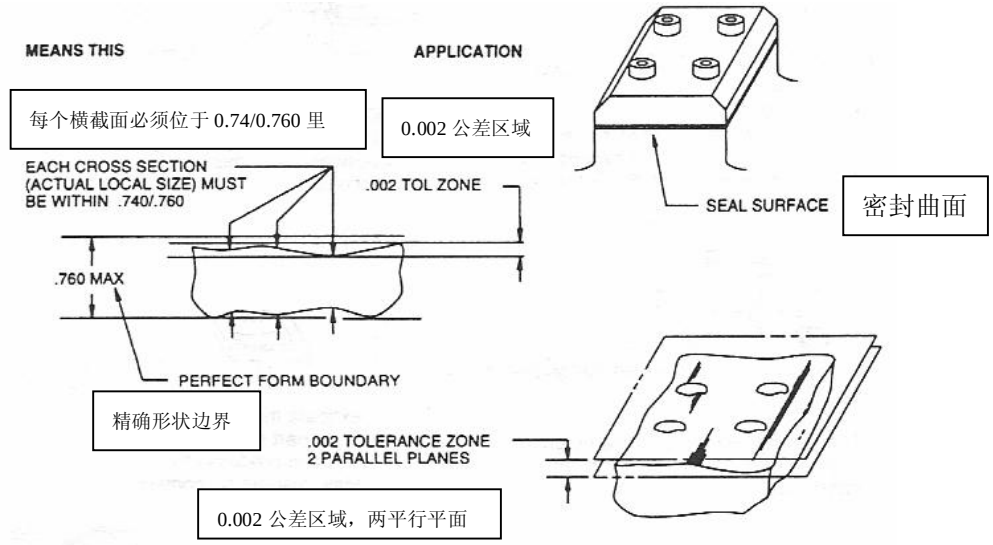
中，应该关心什么时候基准移位或特征旋转被表示为可接受的标准。记住，基准移位和旋转一定计算在所有的成组的特征里，而不允许每一特征图案 作为基准特征偏离 MMC 独立移位.

FLATNESS/平面度

Flatness is the condition of a surface having all elements in one plane. A flatness tolerance specifies a tolerance zone defined by two parallel planes within which the surface must lie.
/平面度是曲面的所有元素在一个平面里的条件。一个平面度公差指定一个公差区域，这个区域是曲面必须位于通过两个平行平面定义的。



The surface must lie between two parallel planes .002 apart. In addition, the surface must be within the specified limits of size or location.
/曲面必须位于两距离为 0.002 的平行平面之间。另外，曲面必须位于指定的尺寸或位置极限里。



Flatness is a 3D tolerance. Flatness is a form tolerance and therefore datums are not allowed. Flatness is a surface control so the modifiers MMC, LMC and RFS are not applicable.
/平面度是一个 3D 公差。 平面度是一个形状公差，因此不允许有基准。平面度是一个曲面控制，因此修正符 MMC, LMC 和 RFS 不适用。

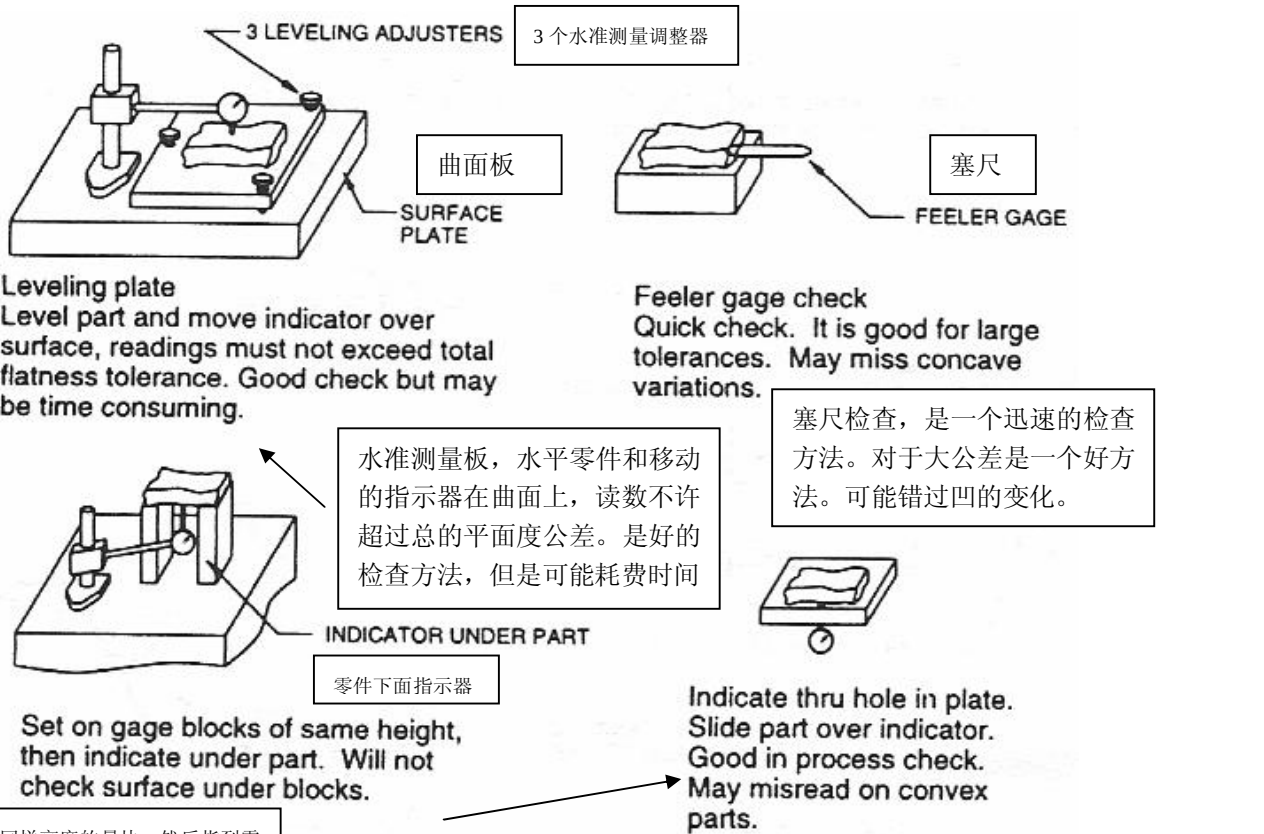
FLATNESS VERIFICATION/平面度验证

There are many ways to check a flatness specification. Some are better than others. Each method will give a different answer. This is called methods divergence. As with verifications for any geometric tolerance, the method or procedure used for verification will depend on many factors. How many parts are there to check? Is this the 1st part produced or the 1000th? Is the tolerance well within the process capability? Are statistical process controls being done? How tight is the tolerance? What kind of equipment is available? Is it an in-process check or a final check? How big is the budget?
/有许多方法检查平面度规范。有一些比其它的更好。每种方法有不同的答案。这叫方法分歧。因为对于任何几何公差的验证，用于验证的方法或程序将依赖许多因素。有多少零件要检查？这是生产的第一个零件还是第一千个零件？公差在工艺能力范围里是满意的吗？统计学工艺控制正在做吗？公差是多么严格？哪种设备可用？这是一个工艺过程中的检测还是最终检测？预算多少？

All of these factors and many more will have an effect on how the part is verified. The procedures for verification should be recorded in a dimensional measurement plan and coordinated with anyone who is involved with the part.
/所有这些因素和许多更多的因素将影响到零件是如何验证的。验证的程序应该被记录在一个尺寸测量计划里，并且列出谁和这个零件有关。

The measurement techniques and procedures shown below are for illustration and background information. These procedures are intended to assist the reader in understanding the concepts. In addition to the methods shown below, a coordinate measuring machine (CMM) can also be used.
/测量技术和下面显示的程序是为了图示和显示一些背景信息。这些程序意图是帮助读者理解这个概念。另外下面显示的方法，坐标测量机也能用。

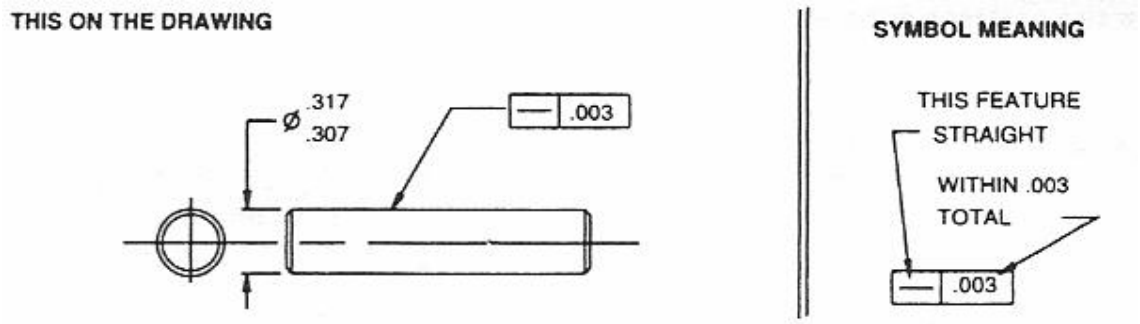
VERIFICATION METHODS FOR FLATNESS/平面度的验证方法



STRAIGHTNESS – LINE ELEMENTS/直线度-线元素

Straightness, line elements is a condition where an element of a surface is a straight line. The tolerance specifies a zone within which the considered line element must lie. A straightness tolerance is applied in the view where the elements to be controlled are represented by a straight line. Note the feature control frame is directed to the surface.

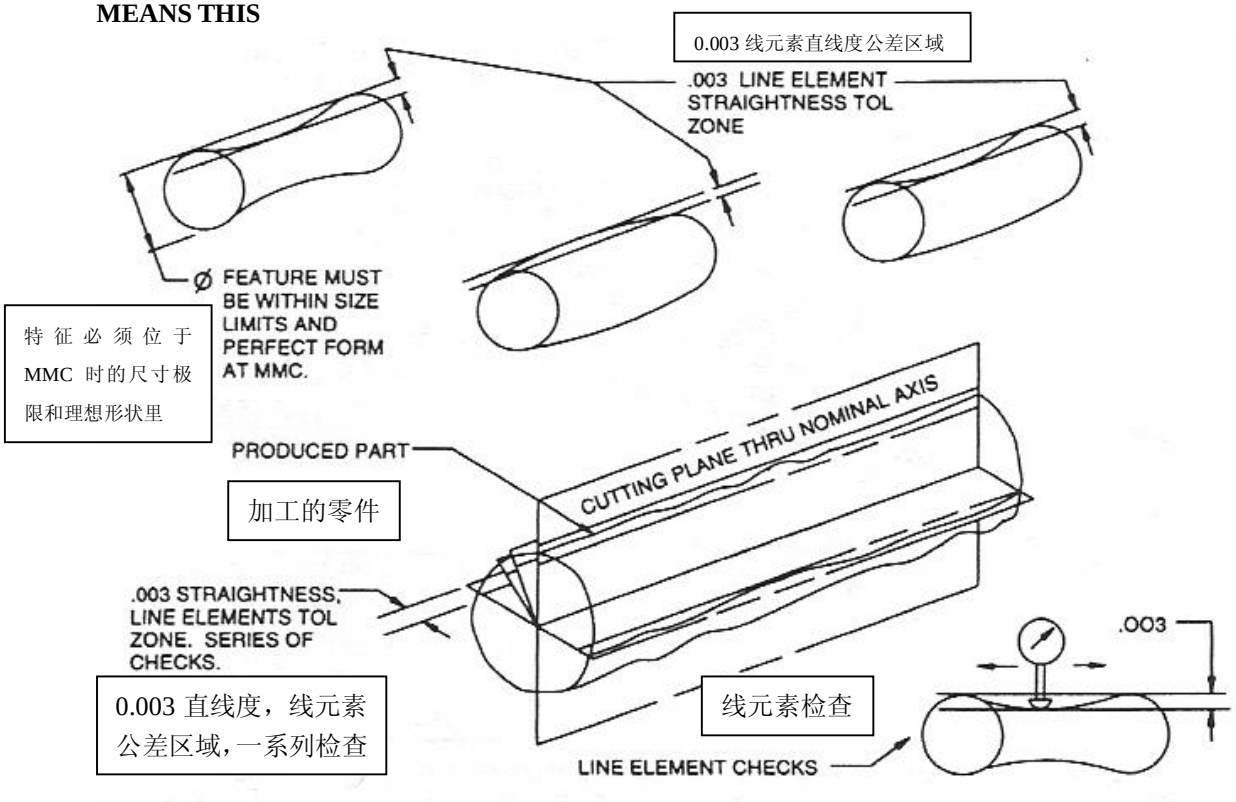
/直线度，线元素是曲面的元素是一直线的条件。公差指定一个区域，考虑的线必须位于这个区域里。 一个直线度公差应用在试图里，被控制的元素由一个直线表示。注意，特征控制框直接指向曲面。



Each longitudinal element of the surface must lie between two parallel lines .003 apart where the two lines and the nominal axis of the part share a common plane. In addition, the feature must be within the specified limits of size and the boundary of perfect form at MMC. Straightness, line elements will control waisted, barreled, and bent shapes, it does not control taper.

/每个曲面的纵向元素必须位于距离 0.003 的两平行线之间，这两条线和名义上的零件的轴位于一个公共平面里。另外，特征必须位于指定的尺寸极限和最大实体条件下的理想边界里。直线度，线元素将控制蜂腰，筒状，和弯曲形状，但不控制锥形。

MEANS THIS



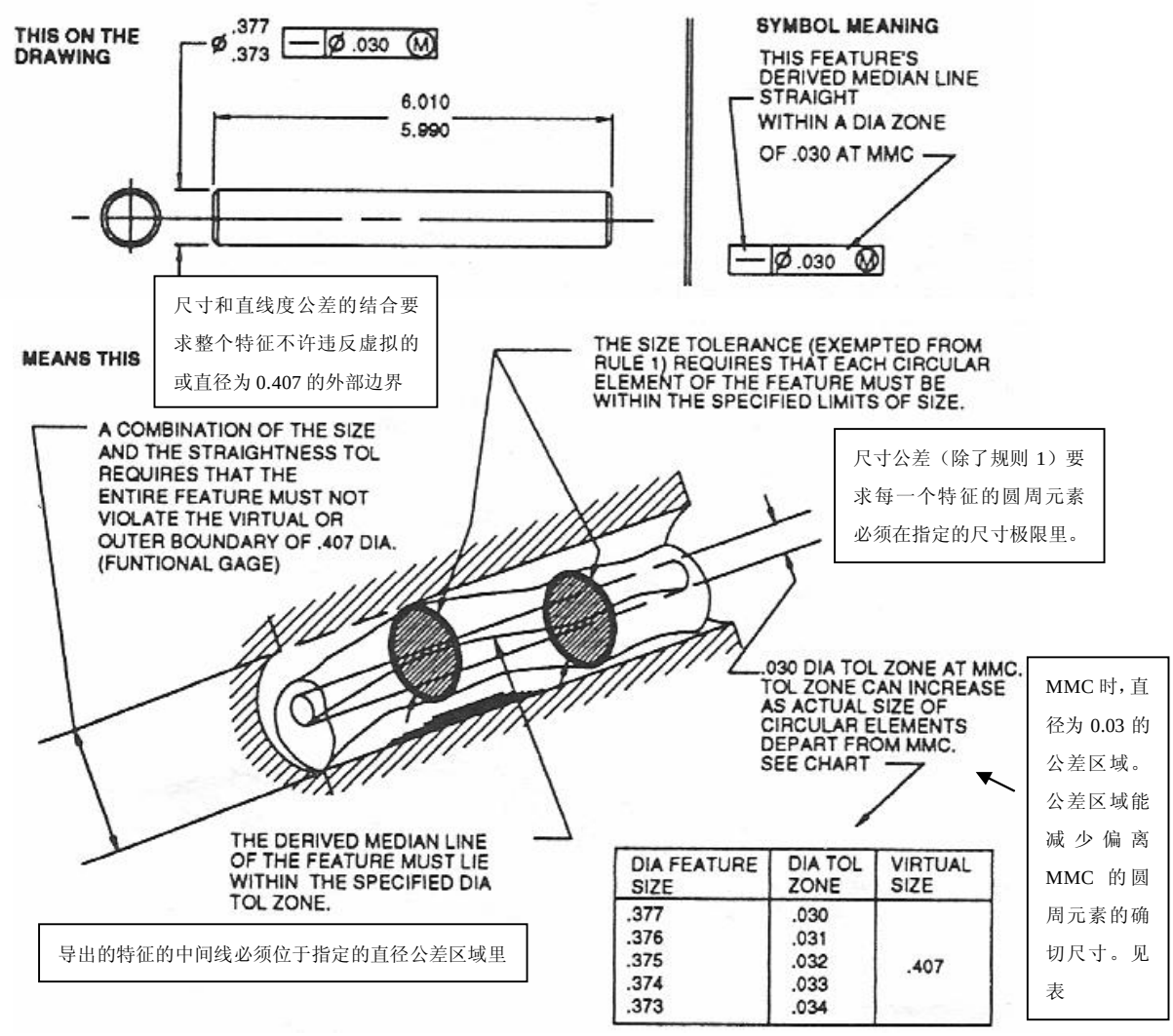
Straightness, Line elements controls the longitudinal elements of the feature. Straightness is a form tolerance, therefore datums are not allowed. Straightness, line elements is a surface control so the modifiers, MMC. LMC and RFS are not applicable. Straightness, line elements is used to refine the size requirements and is always a smaller value than the size tolerance.

/直线度， 线元素控制特征的纵向元素。直线度是一个形状公差，因此，不允许有基准。直线度，线元素是一个曲面控制，因此不应用修正符 MMC，LMC 和 RFS。直线度，线元素用于定义尺寸要求，并且和尺寸公差相比，总是一个最小的值。

STRAIGHTNESS- AXIS/直线度-轴

Straightness of an axis is a condition where an axis is a straight line. The tolerance specifies zone within which the derived median line of the feature must lie. This type of control is used where the size of the pin is important but the part can “bow” or be bent beyond the perfect form limits of size. Note the feature control frame is associated with the size tolerance of the feature.

/轴的直线度是轴是直线的条件。公差指定一个区域，偏离的特征中央线必须在这个区域里。这种控制用于销子的尺寸重要的地方，而零件在理想的形状极限尺寸周围能弓或弯曲。注意，特征控制框和特征的尺寸公差相关联。



The derived median line of the feature's actual local sizes must lie within a cylindrical tolerance zone of .030 diameter at MMC. As each local size departs from MMC, an increase in the local diameter of the tolerance cylinder is allowed which is equal to the amount of such departure. Each circular element of the surface must be within the specified limit of size.
/导出的特征的实际局部尺寸的中间线必须位于一个最大实体条件直径为 0.03 的圆柱公差区域里。因为每一局部尺寸偏离最大实体条件，局部公差圆柱的直径允许有一个增量，等于这样的偏离量。曲面的每一个圆形的元素必须在指定的尺寸极限里。

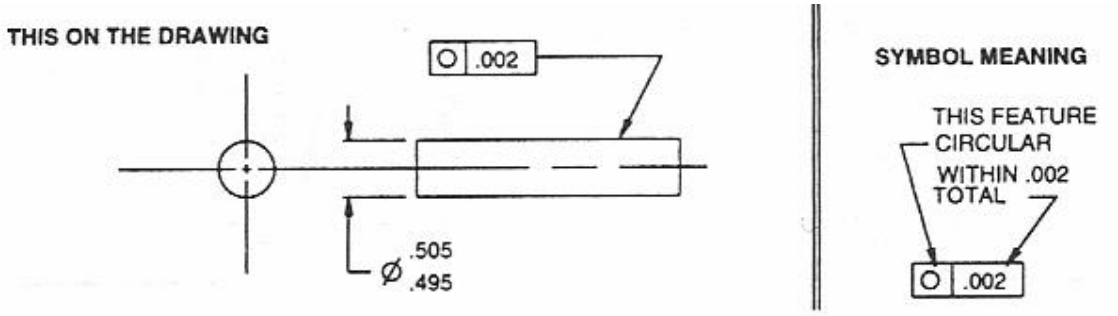
CIRCULARITY – ROUNDNESS/环状-圆度

Circularity is a condition of a surface where:

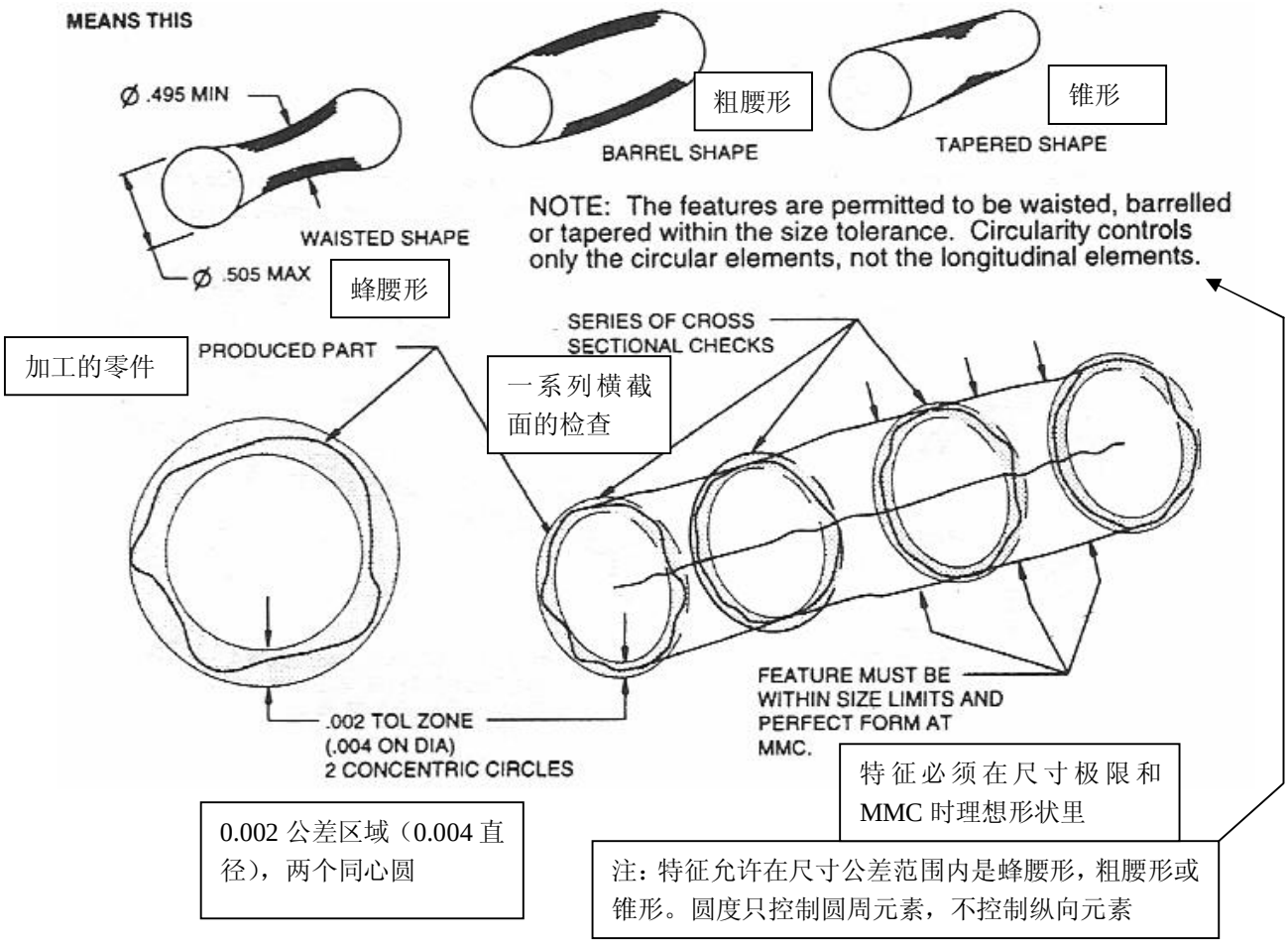
a. For a feature other than a sphere, all points of the surface intersected by any plane perpendicular to an axis are equidistant from that axis.
b. For a sphere, all points of the surface intersected by any plane passing through a common center are equidistant from that center.

/圆度是一个曲面的条件：

a. 对于一个特征，除了球，被任何和一个轴正交分割的曲面的所有点到这个轴是等距离的。
b. 对于一个球，通过一个公共中心的平面分割的曲面的所有的点到这个中心是等距离的。



Each circular element of the surface in a plane perpendicular to an axis must lie between two concentric circles, one having a radius .002 larger than the other. In addition, the feature must be within the specified limits of size.
/每一个和一个轴正交的平面里曲面的圆形元素，必须位于两个同心圆之间，一个比另一个半径大 0.002。另外， 特征必须位于指定的尺寸极限里。



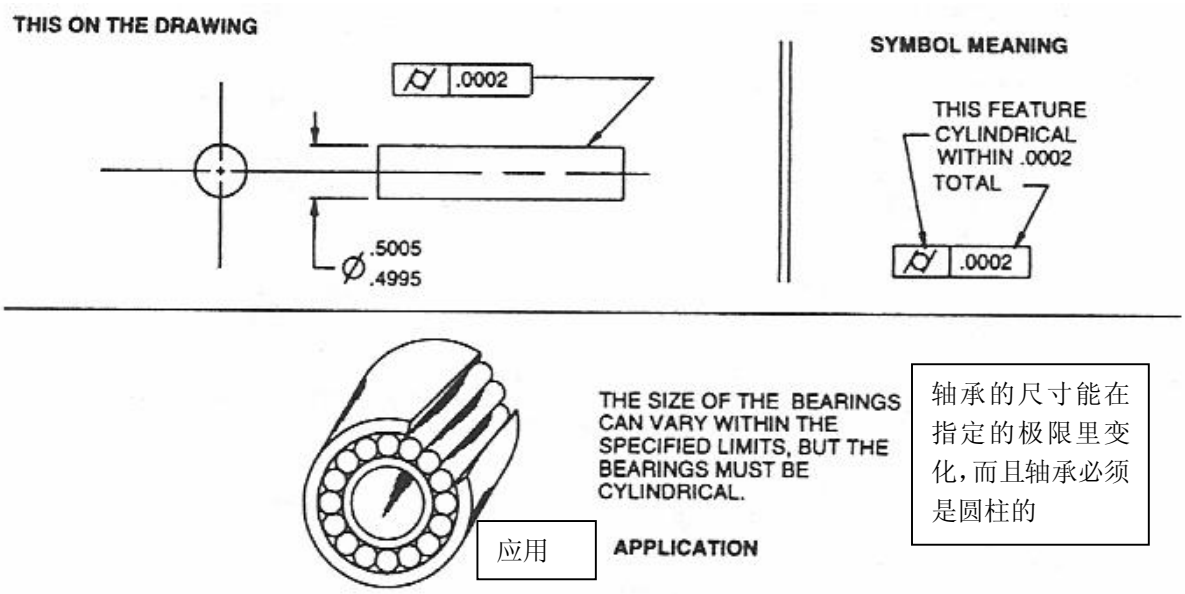
Circularity is a 2D tolerance; it controls circular elements only, not longitudinal elements. Circularity is a form tolerance and therefore, datums are not allowed. Circularity is a surface control so the modifiers MMC, LMC and RFS are not applicable. Circularity tolerance is used to refine the size requirements and is always less than the size tolerance. (NOTE: circularity applied in a Free State condition can exceed the size tolerance as the perfect form requirement of size is negated.) Examples of use: ball bearings, tubes, pipes and circular elements of tapered, barreled or waisted parts such as nose or tail cones, seals etc.

/圆度是一个 2D 公差，它仅仅控制圆形元素，不控制纵向元素。圆度是一个形状公差，并且不允许有基准。圆度是一个曲面控制，因此修正符 MMC, LMC 和 RFS 不适用。圆度公差用于定义尺寸要求，并且总是比尺寸公差小。（注：圆度应用在自由状态条件时，能超过尺寸公差，因为尺寸的理想形状要求被忽略。）使用的例子：球轴承，管子，和锥形的，筒形的和蜂腰形的零件的圆形元素，如飞机或船的前端或尾翼，图章等。

CYLINDRICITY /圆柱度

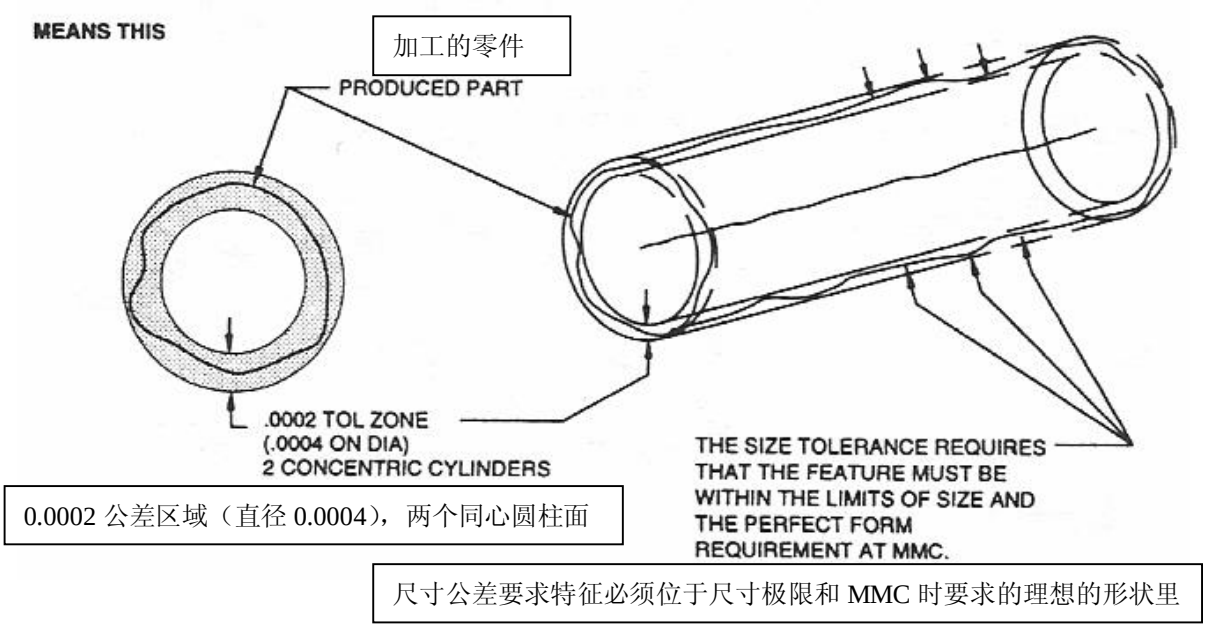
Cylindricity is the condition of a surface of revolution in which all points of the surface are equidistant from a common axis. The tolerance zone is two concentric cylinders within which the surface must lie.

/圆柱度是旋转的曲面的条件，曲面上所有的点对于一个公共轴等距离。公差区域是两个同心圆柱面，曲面必须位于其中。



The cylindrical surface must lie between two concentric cylinders, on having a radius of .0002 larger than the other. In addition, the feature must be within the specified limits of size.

/圆柱曲面必须位于两个同心圆柱面之间，一个比另一个半径大 0.0002。另外，特征必须位于指定的尺寸极限里。



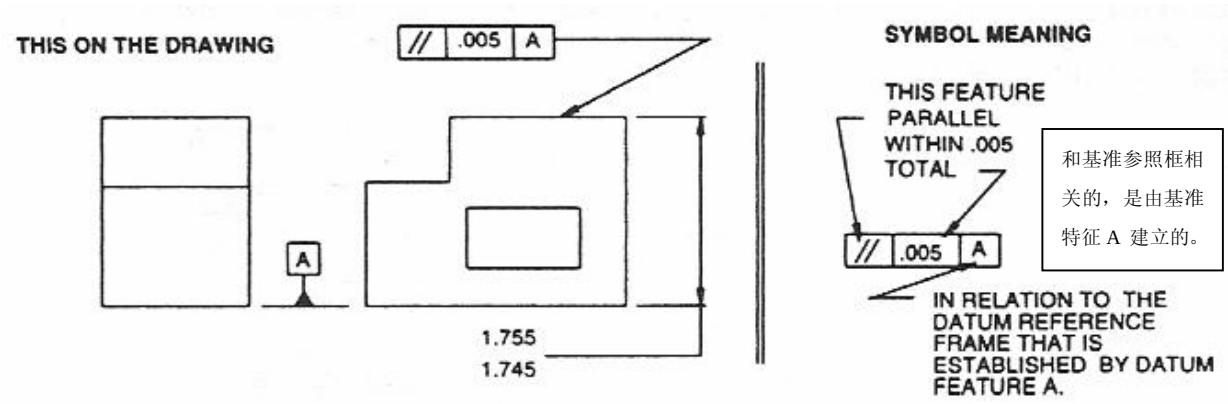
Cylindricity is a 3D tolerance; it controls both the circular and longitudinal elements of the feature. It includes circularity, straightness and taper. Cylindricity is a form tolerance, therefore datums are not allowed. Cylindricity is a surface control so the modifiers MMC, LMC and RFS are not applicable. Cylindiricity tolerance is used to refine the size requirements and is always a smaller value than the size tolerance. Examples of use: bearings as shown above, bearing journals, cylinders etc. Cylindiricity can be compared to flatness wrapped around the surface.

/圆柱度是一个 3D 公差，它控制特征的圆周和纵向的元素。它包括圆度，直线度和锥度。圆柱度是一个形状公差，因此不允许有基准。圆柱度是一个曲面控制，因此修正符 MMC,LMC 和 RFS 不适用。圆柱度公

差用于定义尺寸要求，并且总比尺寸公差值小。使用的例子：以上所示的轴承，圆柱体等。圆柱度能和环绕曲面的平面度比较。

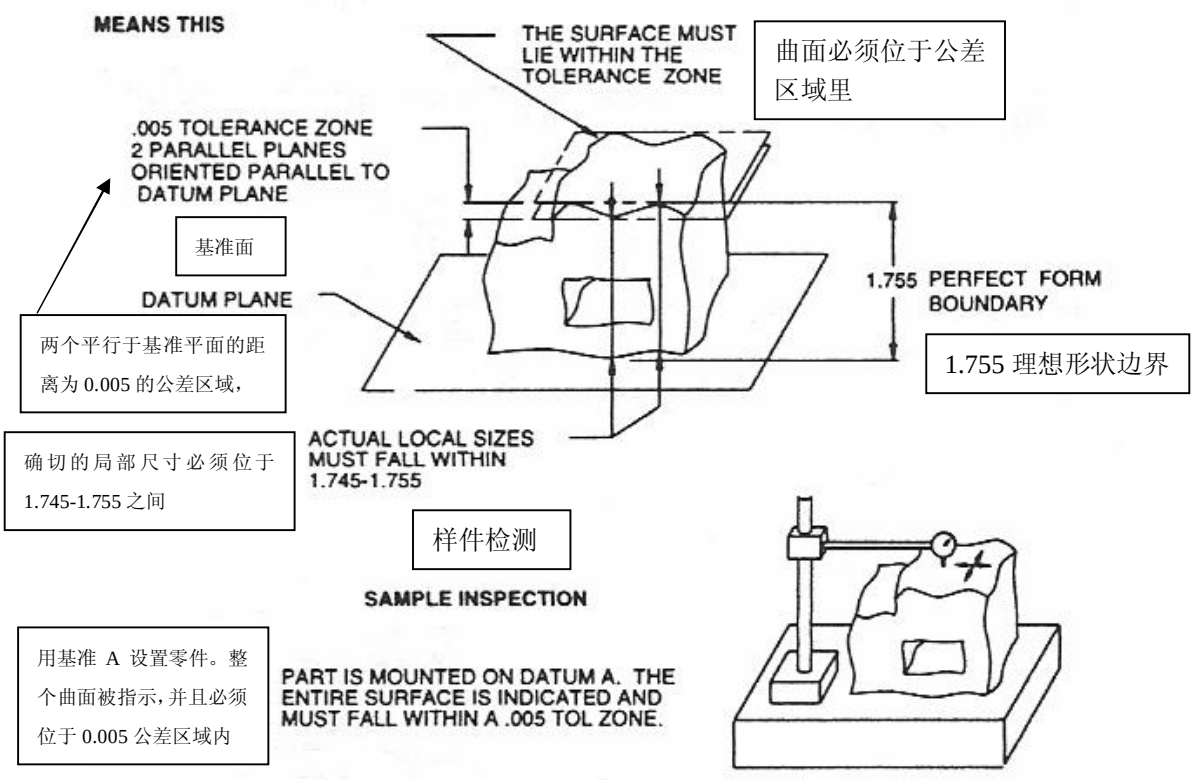
PARALLELISM-SURFACE/平行度-曲面

Parallelism is the condition of a surface or center plane, equidistant at all points from a datum plane: or an axis, equidistant along its length from one or more datum planes or a datum axis.
/平行度是一个曲面或中间平面的条件，所有点对于基准平面，或一个轴等距离，从一个或更多的基准面或基准轴，沿长度方向等距离。



和基准参照框相关的，是由基准特征 A 建立的。

The surface must lie between two parallel planes .005 apart which are parallel to datum plane A. in addition, the surface must be within the specified limits of size or profile.
/曲面必须位于距离为 0.005 的两平行平面之间，并且这两平面和基准面 A 平行。另外，曲面必须位于指定的轮廓尺寸极限内。



曲面必须位于公差区域里

基准面

两个平行于基准平面的距离为 0.005 的公差区域，

1.755 理想形状边界

确切的局部尺寸必须位于 1.745-1.755 之间

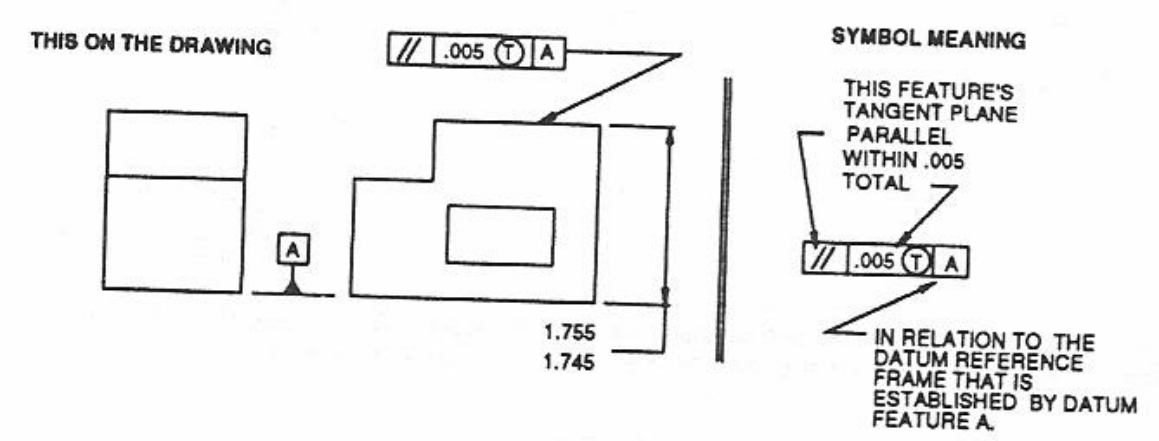
样件检测

用基准 A 设置零件。整个曲面被指示，并且必须位于 0.005 公差区域内

The above parallelism specification is a surface control. The MMC, LMC and RFS modifiers are not applicable. Parallelism tolerance applied to a plane surface also controls the flatness of that surface. Thus, if no flatness tolerance is specified, the flatness tolerance will be at least as close as the parallelism requirement.
/以上的平行度规范是一个曲面控制。 MMC, LMC 和 RFS 修正符不适用。平行度公差应用到平面也控制曲面的平面度。这样，如果没有指定平面度公差，平面度公差至少和平行度要求的接近。

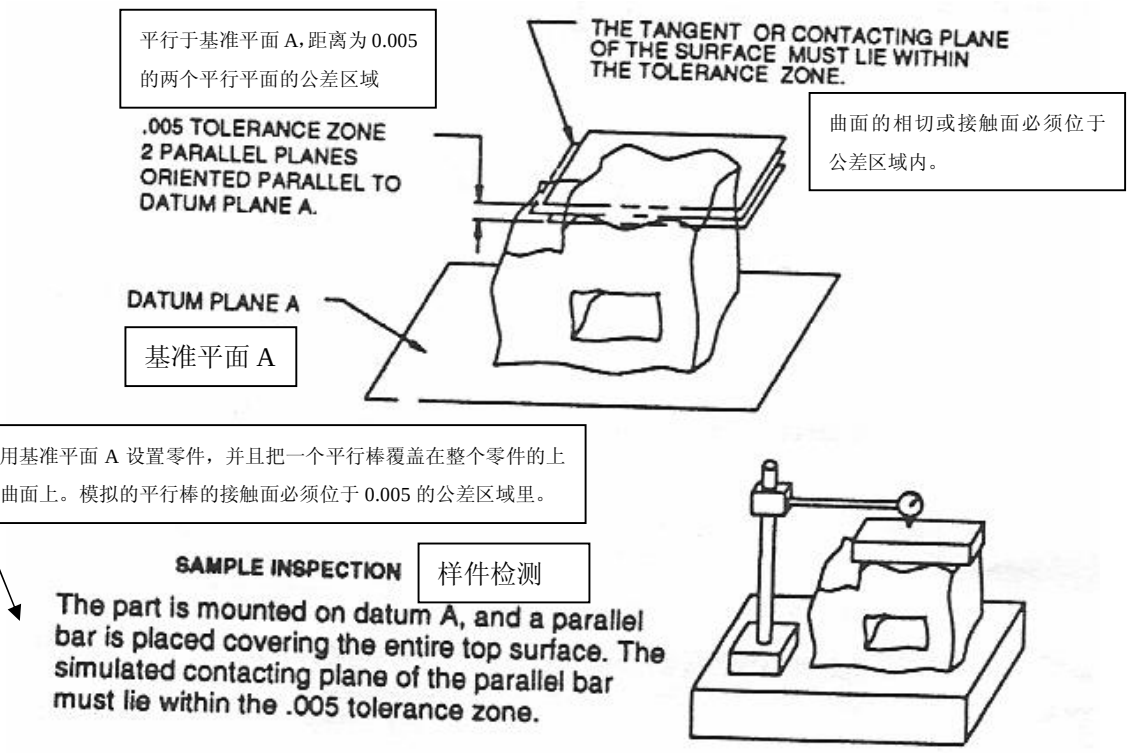
PARALLELISM-TANGENT PLANE/平行度-相切面

The tangent plane symbol can be applied to many of the geometric characteristics. The tangent plane symbol is shown below with a parallelism specification. A plane contacting the high points of the surface must lie within the parallelism tolerance. The tangent plane application does not control flatness of the surface. If flatness of the surface is of concern, a separate control must be specified.
/相切面符号能应用到许多几何特征。相切面符号在下面的平行度公差规范里显示。和曲面 的一个高点接触的一个平面必须位于平行度公差里。 应用相切面不控制曲面的平面度。如果关系到曲面的平面度，必须单独指定。



The tangent plane must lie between two parallel planes .005 apart which are parallel to datum plane A. In addition, the surface must be within the specified limits of size or profile.
/相切面必须位于距离为 0.005 的两平行平面之间，并且这两个平行平面平行于基准 A。 另外，曲面必须位于指定的轮廓或尺寸极限里。

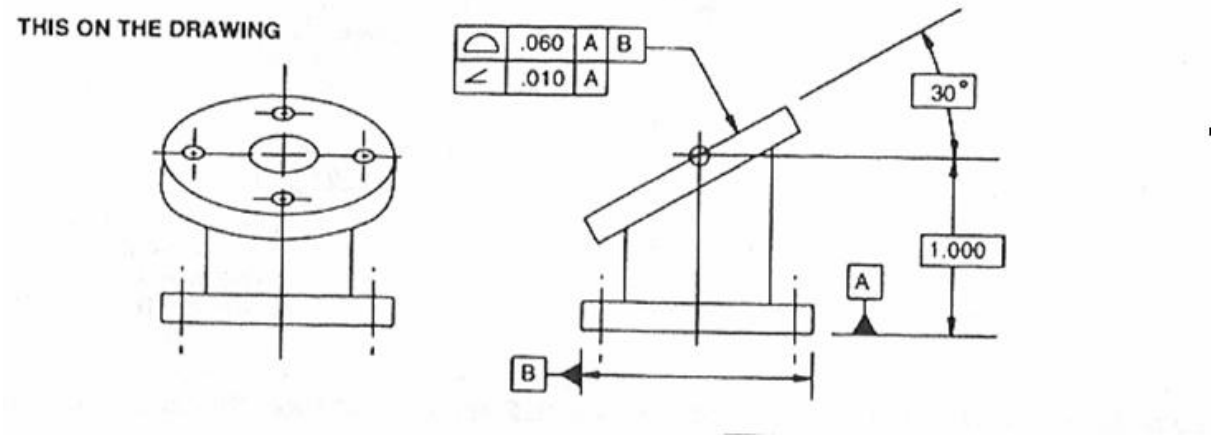
MEANS THIS



ANGULARITY/倾斜度

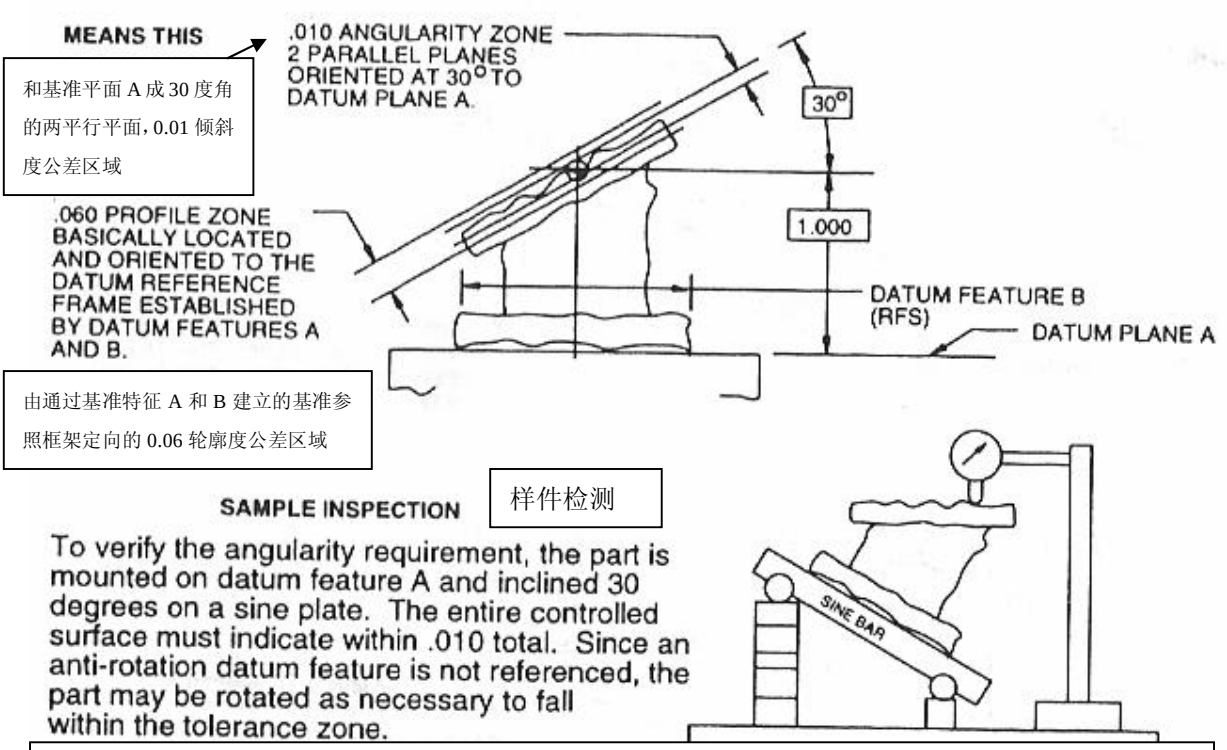
Angularity is the condition of a surface, center plane, or axis at a specified angle (other than 90 or 180) from a DRF. The example below illustrates a surface located with a profile tolerance and then refined for orientation with an angularity tolerance.

/倾斜度是一个基于 DRF 的曲面，中心平面，或指定角度的轴 （除了 90 度或 180 度）的条件。下面的例子图示了曲面和轮廓度公差一起，和一个倾斜度公差使方向精细化。



The surface must lie between two parallel planes .010 apart which are inclined at 30 ° to datum plane A. In addition, the surface must also fall within the profile zone.

/曲面必须位于两个距离为 0.010 的平行平面之间，这两个平面相对于基准平面 A 倾斜 30 度。另外，曲面也必须位于轮廓区域里。

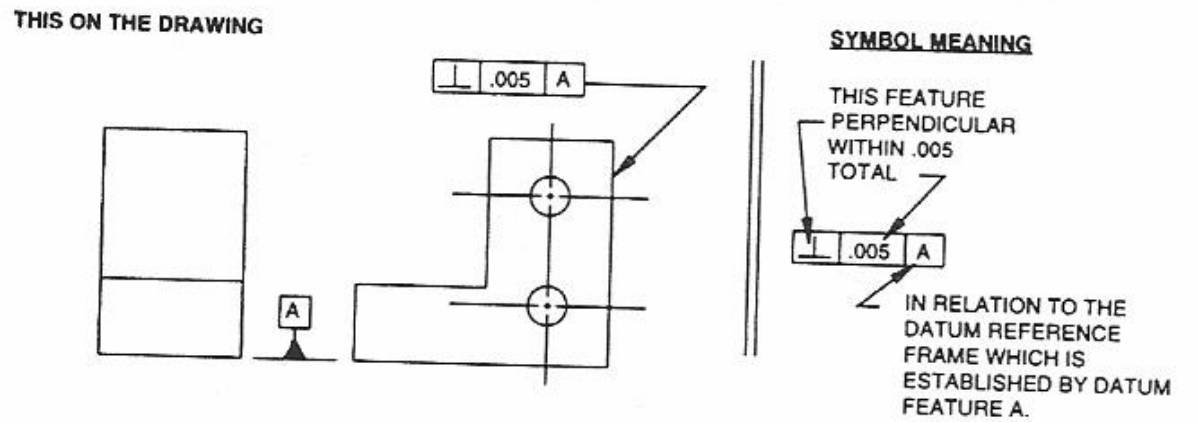


为了验证倾斜度要求，零件以基准 A 安装，并在正弦板上倾斜 30 度角。整个控制曲面必须在总共 0.01 范围内指示。既然一个反旋转的基准特征没有被参照，零件可以按需求旋转，并落在公差区域内。

Angularity is a 3D tolerance. Angularity tolerance applied to a plane surface also controls the flatness of that surface. The surface of the part above is not controlled for rotation relative to the DRF. If rotation is control is necessary, additional datums (possibly the pattern of holes on the bottom surface) can be specified.
/倾斜度是一个 3D 公差。倾斜度公差也适用到平曲面，控制那个曲面的平面度。以上这个零件的曲面相对于 DRF 旋转不受控制。如果旋转控制是必需的，可能指定另外的基准（可能在底面上的孔的阵列）。

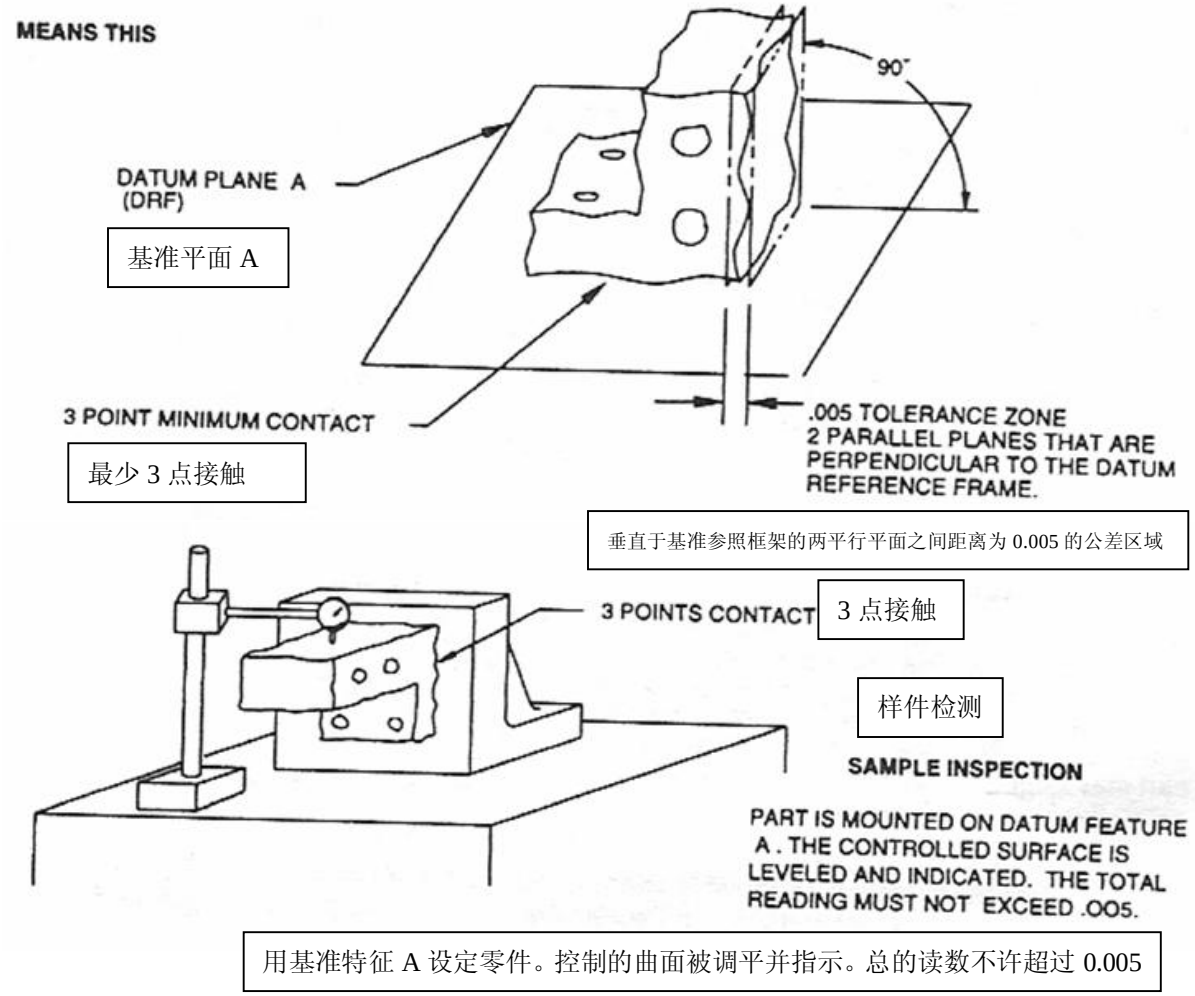
PERPENDICULARITY-SURFACE/垂直度-曲面

Perpendicularity is the condition of a surface, median plane or an axis at a right angle to datum plane(s) or axes.
/垂直度是曲面，中间面或一个轴在相对于基准面或轴的直角处的条件。



The surface must lie between two parallel planes .005 apart. In addition, the feature must be within the limits of

size or location.
/曲面必须位于两个距离为 0.005 的平行平面之间。另外，特征必须位于尺寸或位置的极限里。

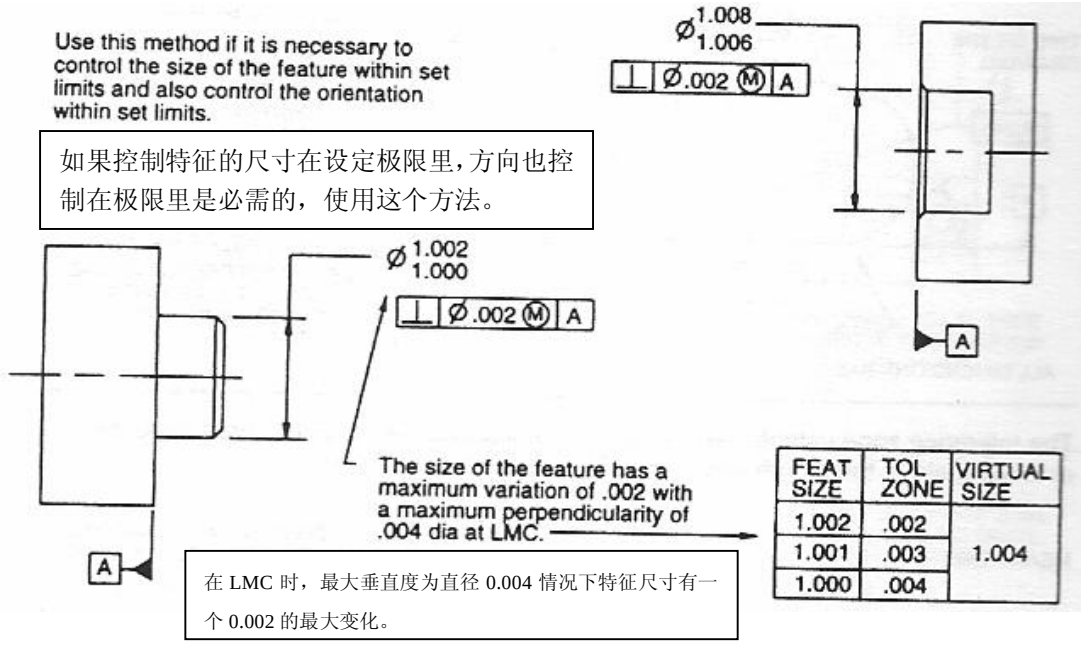


Perpendicularity is a 3D tolerance. Perpendicularity tolerance applied to a plane surface also controls the flatness of that surface. Thus, if no flatness is specified, the flatness tolerance will be at least as close as the perpendicularity requirement.
/垂直度是一个 3D 公差。垂直度公差也应用到一个平曲面，控制曲面的平面度。这样，如果没有指定平面度，平面度公差将至少和垂直度公差要求接近。

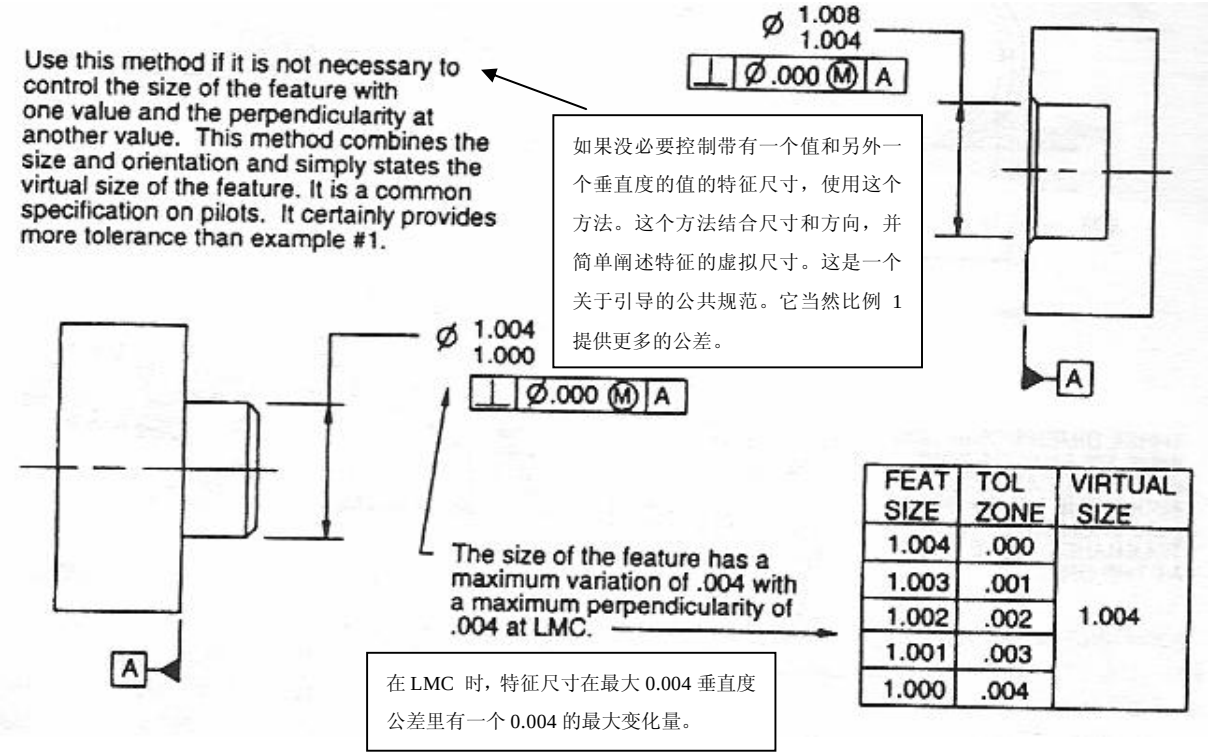
PERPENDICULARITY – ZERO TOLERANCING AT MMC 垂直度-MMC 时零公差

The two examples below represent two mating assemblies. Example#1 specifies a .002 at MMC perpendicularity requirement. Example #2 specifies a zero perpendicularity at MMC requirement. Notice that example #2 provides more tolerance (greater variation on size) while still maintaining the same virtual size as example #1. The zero tolerancing concepts can be applied to other geometric tolerances as well.
/下面的两个例子代表两个配合的装配。例子 1 指定在 MMC 时垂直度为 0.002。例子 2 指定一个在 MMC 时零垂直度要求。注意，例子 2 在还维持例子 1 中虚拟尺寸的同时，提供更多公差（更多尺寸变化）。零公差概念也能应用到其它几何公差上。

EXAMPLE #1 – PERPENDICULARITY .002 DIA AT MMC/例 1-MMC 时垂直度直径 0.002



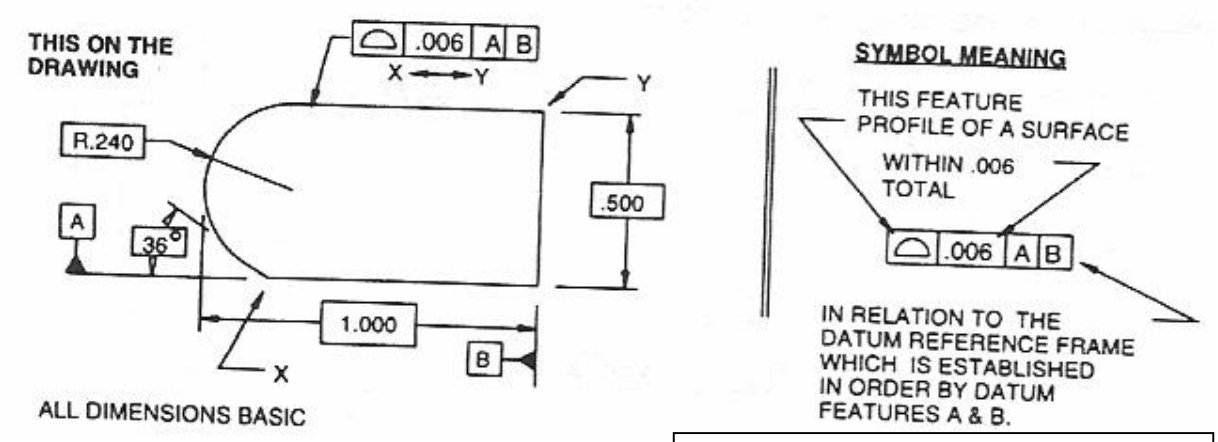
EXAMPLE #2 – PERPENDICULARITY .000 DIA AT MMC/例 2-MMC 时垂直度为直径 0



PROFILE OF A SURFACE – BILATERAL/面轮廓度-双面

Profile tolerancing specifies a uniform boundary along the true profile within which the elements of the surface must lie. It is used to control form or combinations of size, form and location. Profile tolerancing can be applied on a bilateral, unilateral or unequal distribution basis. Below is shown the bilateral distribution. The feature control frame arrow points directly at the surface.

/轮廓度公差指定一个统一边界，沿着曲面元素必须位于其中的真实轮廓。用于控制尺寸的形状或尺寸，形状和位置。轮廓度公差能应用在双面的，单面的或不均衡的分配。下面所示是双面分配。特征控制框箭头直接指向曲面。

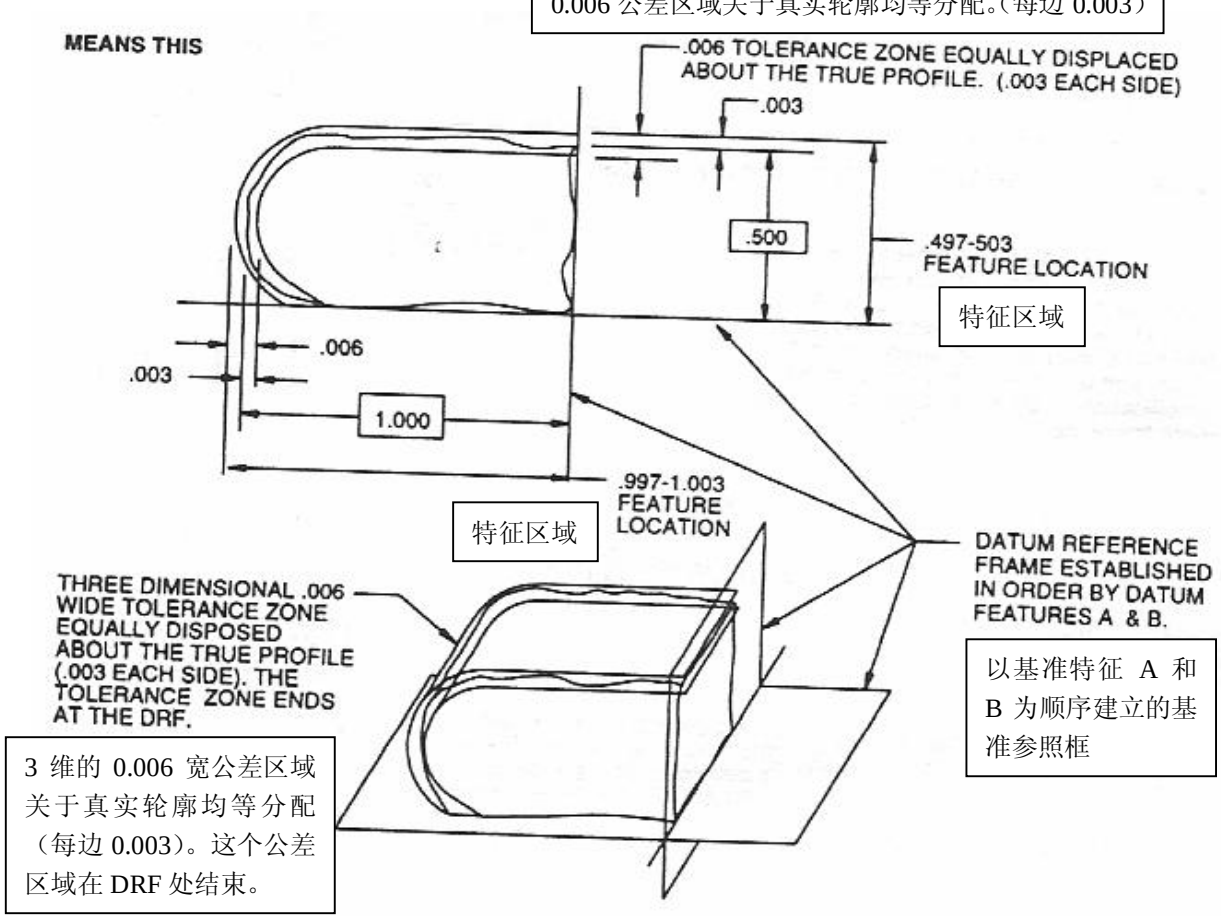


和由以基准特征 A 和 B 的顺序建立的基准特征框相关的

The tolerance zone established by the profile of a surface control is three dimensional, extending along the length and width of the considered feature.

/由曲面控制的轮廓建立的公差区域是三维的，沿着所考虑的特征的长度和宽度延伸。

0.006 公差区域关于真实轮廓均等分配。(每边 0.003)

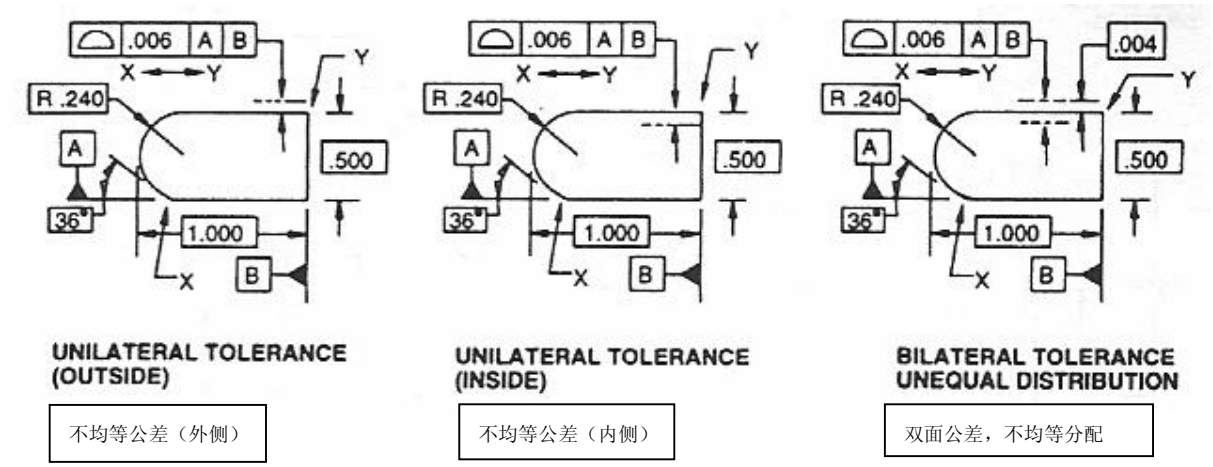


3 维的 0.006 宽公差区域关于真实轮廓均等分配（每边 0.003）。这个公差区域在 DRF 处结束。

PROFILE OF A SURFACE – UNILATERAL & BILATERAL UNEQUAL DISTRIBUTION /面轮廓度-单面&双面不均衡分配

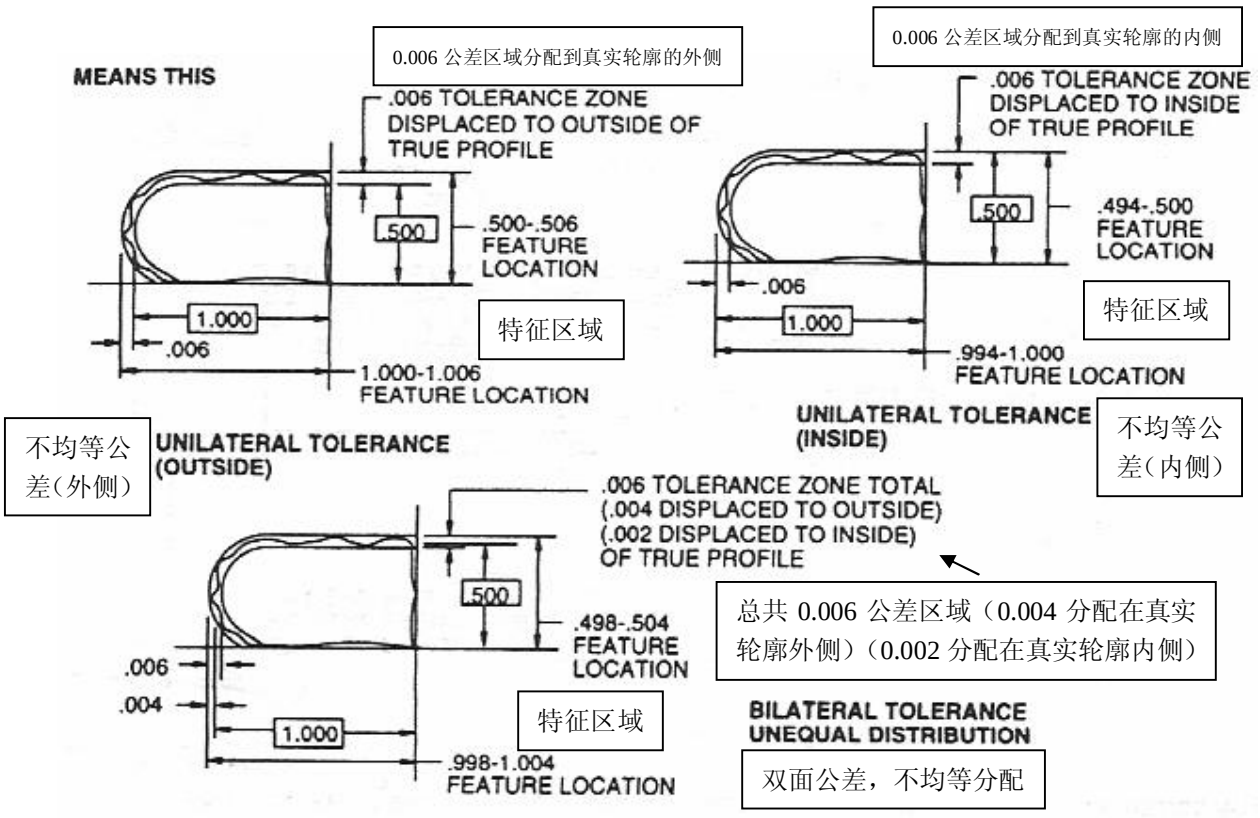
Profile tolerancing can be applied equally or unequally about the true profile. If the leader arrow from the feature control frame points directly at the surface the profile is equally distributed. If an unequal distribution is required two arrows are used to define the width or direction of the tolerance zone.
/轮廓度公差可以均等或不均等的应用到真实轮廓。如果从特征控制框引出的箭头直接指向曲面，轮廓是平等分配的。如果要求一个不平等的分配，用两个箭头定义公差区域的宽度或方向。

THIS ON THE DRAWING



不均等公差（外侧）　不均等公差（内侧）　双面公差，不均等分配

NOTE: TRUE PROFILE IS DEFINED BY BASIC DIMENSIONS/注：真实轮廓由基本尺寸定义

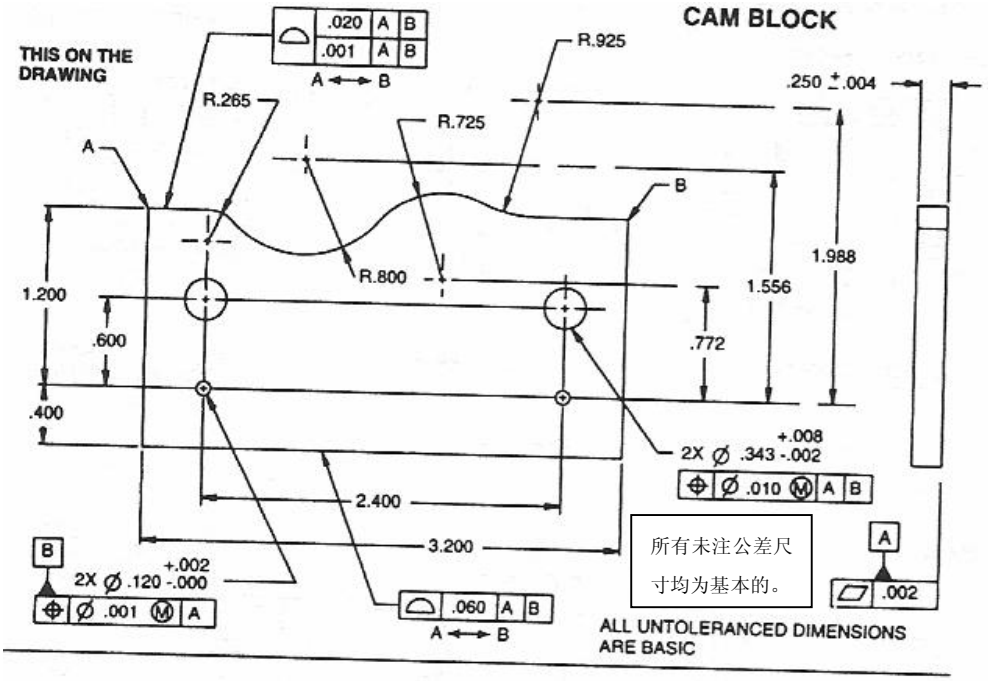


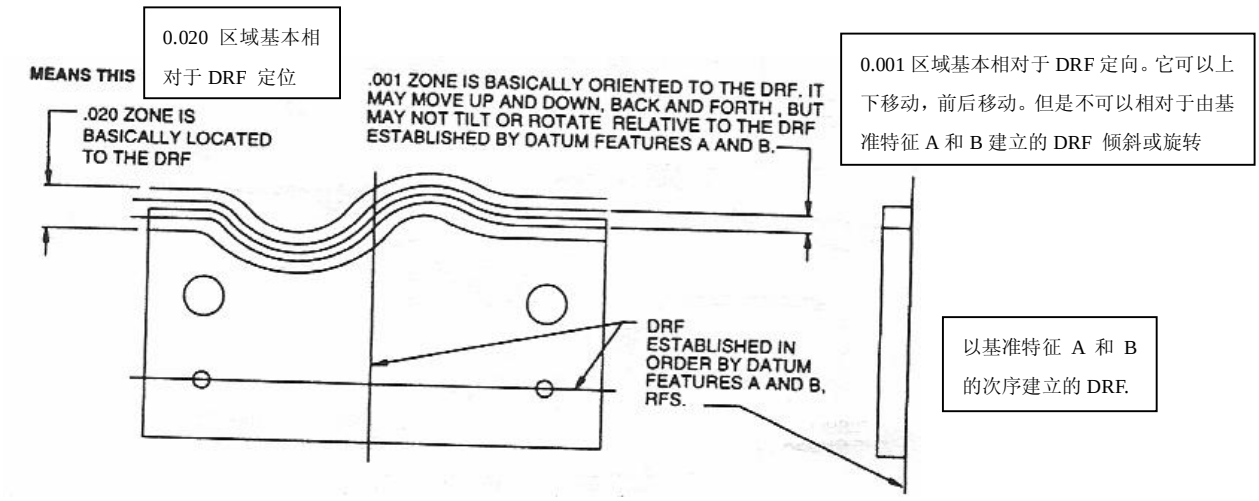
In some cases, where it is not clear as to the type of profile or the direction of the tolerance zone displacement a

note may be added under the feature control frame. The note may read UNILATERAL OUT, UNILATERAL IN OR UNEQUAL DISTRIBUTION. This method may be necessary when working with solid models, untoleranced drawings, sheet metal parts or where the drawing specification is unclear.
/在一些情况下，关于轮廓度的类型或公差区域移位的方向不清楚的地方，可以在特征控制框下加一个注释。注释可读作“单边外侧”，“单边里侧”，“不均等分配”。在实体模型，没有标公差的图，板件或图纸规范不清楚的地方，这个方法是必需的。

COMPOSITE PROFILE – TWO DATUM FEATURES/复合轮廓度-两个基准特征

The upper entry on composite tolerancing controls location to the DRF. The lower entry on composite tolerancing controls size/shape and orientation (perpendicularity, parallelism or angularity) to the DRF established by the specified datums.
/复合公差标注里的上面的条目控制 DRF 的位置。复合公差的下面的条目控制尺寸/形状和由指定基准建立的 DRF 的方位（垂直度，平行度或倾斜度）。





The composite tolerance specification above allows the .001 zone to “float “up and down and back and forth within the confines allowed by the .020 zone. The .001 tolerance zone, however, may not tilt or rotate. The datums in the lower entry control the orientation (parallelism/perpendicularity) of the .001 zone relative to the DRF established by datum features A and B. the surface of the part must lie in both zones simultaneously and meet both requirements.

/上面的复合公差规范允许在 0.02 的区域限制里有 0.001 的 上，下，前，后浮动。然而，0.001 的公差区域不会倾斜或旋转。下面条目的基准控制 0.001 区域相对于由基准 A 和 B 建立的 DRF 的方位（平行度/垂直度）。零件的曲面必须同时位于两个区域，并且两个要求全满足。

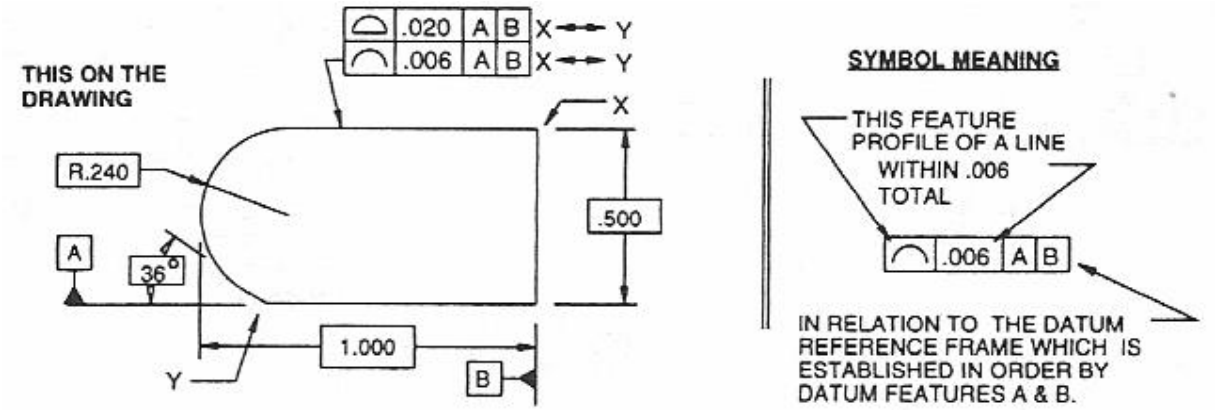
PROFILE OF A LINE – BILATERAL/线轮廓度-双面的

The concepts for profile of a surface and profile of a line are identical, with the exception that profile of a surface is three dimensional and profile of a line is two dimensional. Profile of a line is often used in conjunction with profile of a surface as shown below.

/除了曲面的轮廓度是三维的，而一条线的轮廓度是两维的以外，曲面的轮廓度概念和一条线的轮廓度概念是一样的。一条线的轮廓度经常和下面所示的曲面的轮廓度一起使用。

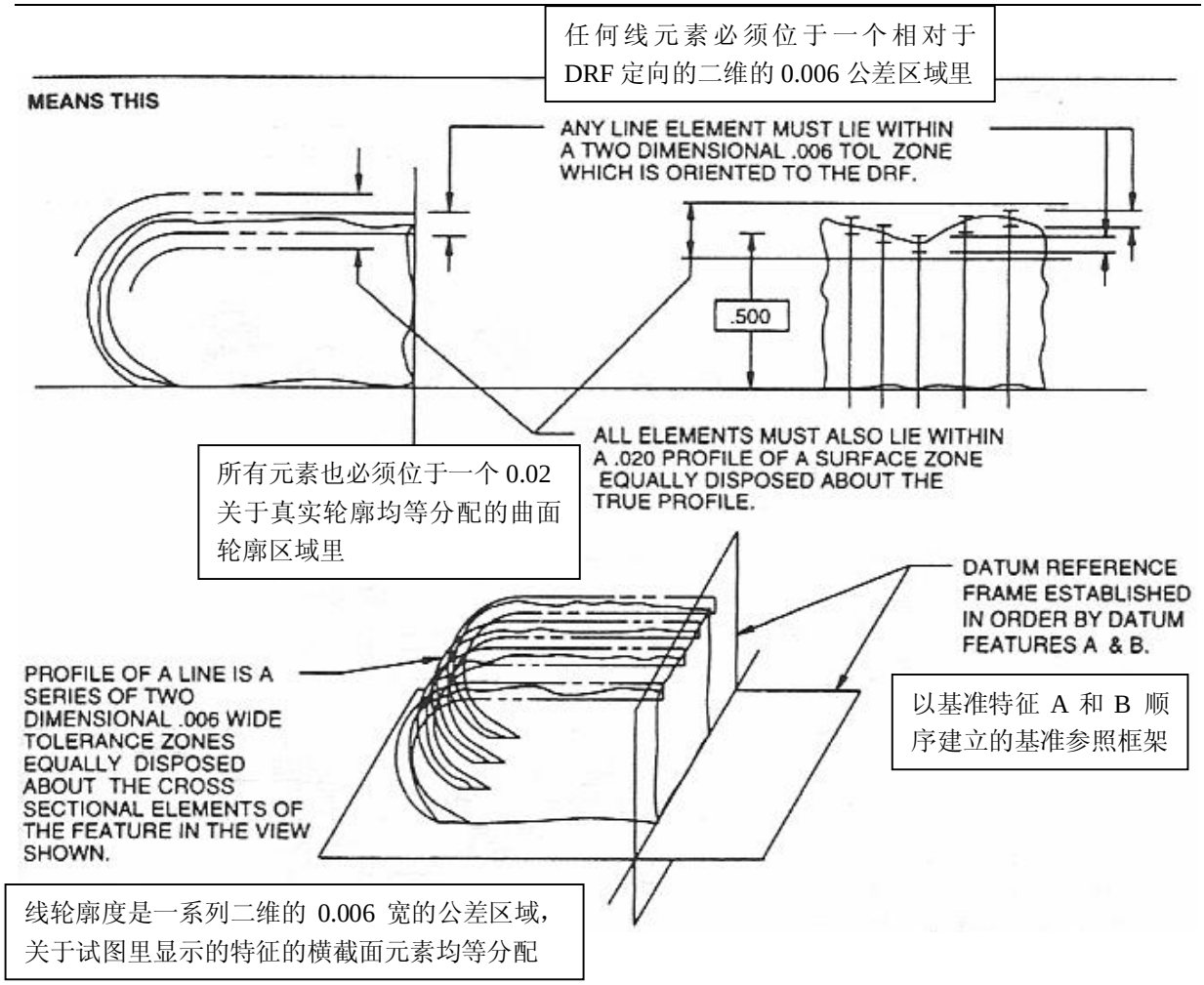
Profile of a surface defines the shape or location of a feature, and profile of a line is used to refine the feature in one direction, as in extruded parts. The line elements apply in the view in which the feature control frame is directed.

/曲面轮廓度定义一个特征的形状或位置，而一条线的轮廓度用于在一个方向上定义特征，像在一个拉伸的零件里。在试图里应用线元素，而且特征控制框被指向。



关于以基准特征 A 和 B 的顺序建立的基准特征框架

ALL DIMENSIONS BASIC



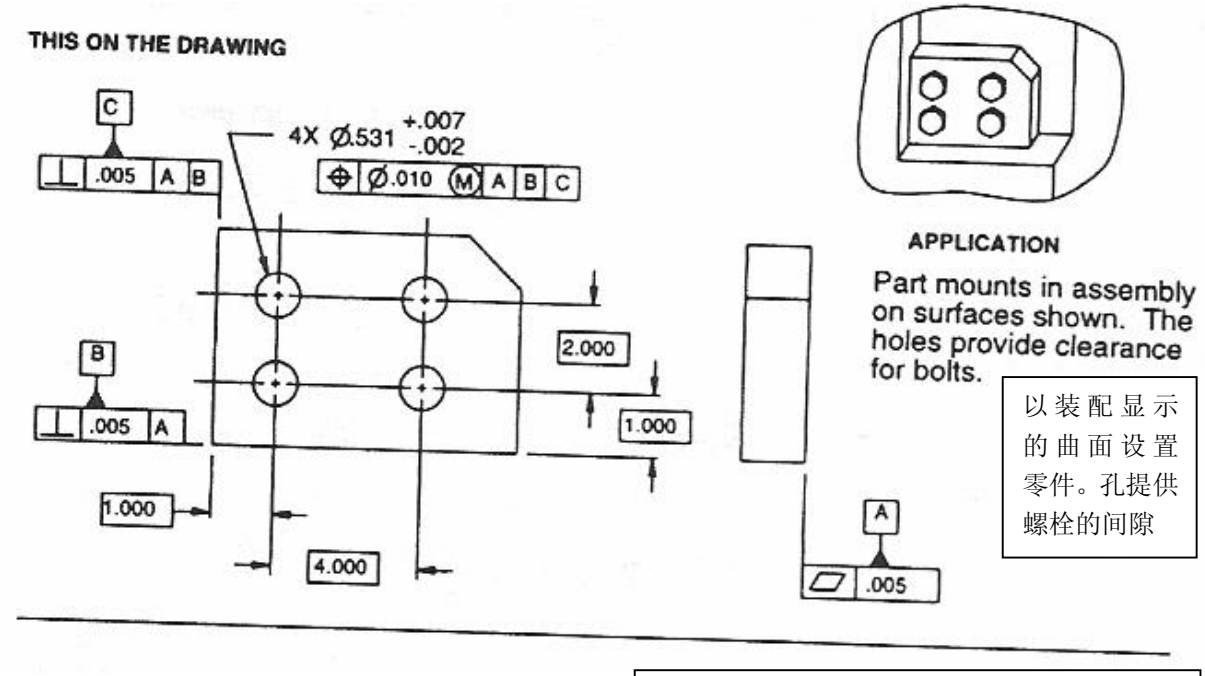
Each line element of the surface must lie between two profile boundaries .006 apart in relation to the datum reference frame. When used in conjunction with a size tolerance or a profile of a surface control, the surface must also fall within these specified limits.

/曲面的每一线元素必须位于两个关于基准参照框架距离为 0.006 的轮廓边界之间。当和一个尺寸公差或曲面轮廓控制一起使用时，曲面必须位于这些指定的极限里。

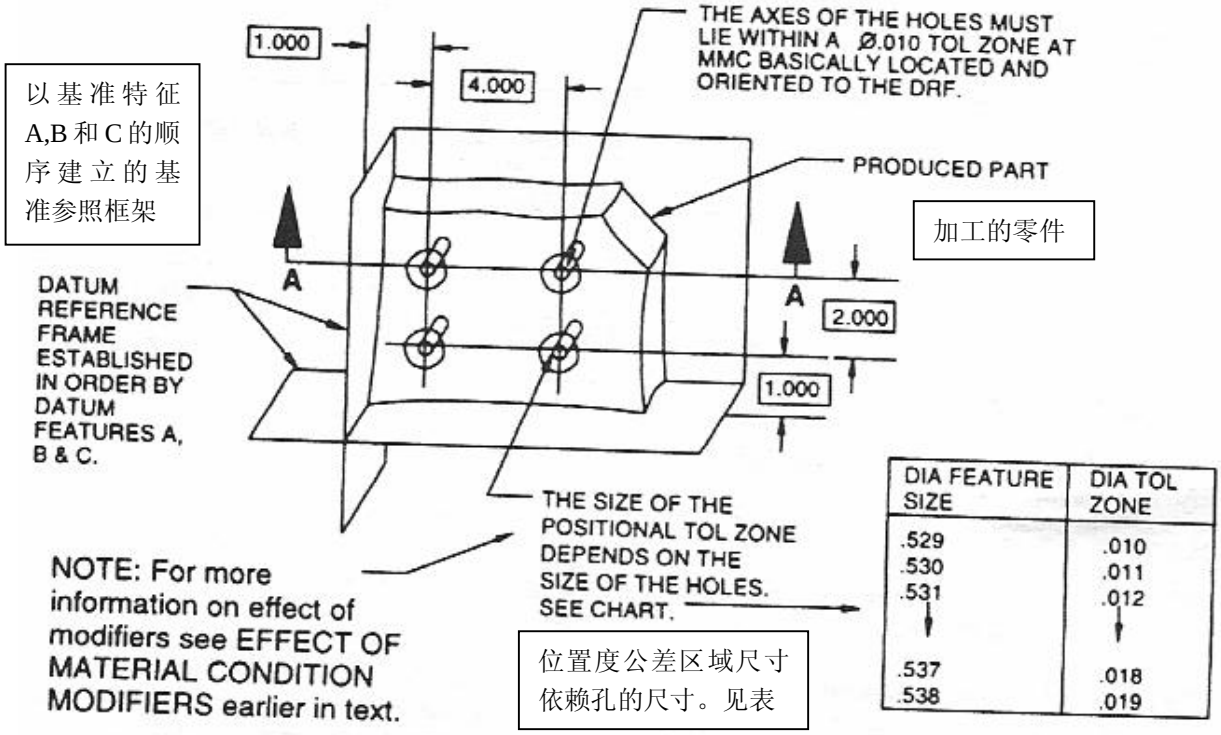
POSITION TOLERANCING/位置度

Position tolerancing is used for locating features of size. It defines a zone within which the axis of a feature is permitted to vary from a true (theoretical exact) position. Basic dimensions establish the true position from the specified datum features as well as the interrelationship between the features.

/位置度用于定位尺寸特征 。它定义一个区域，在这个区域里特征的轴允许由一个真实位置（理论精确）变化。基本尺寸由指定的基准特征建立真实位置，以及特征之间的相互关系。



在基本相对于 DRF 定向的最大实体条件下，孔的轴必须位于一个直径为 0.01 的公差区域里。



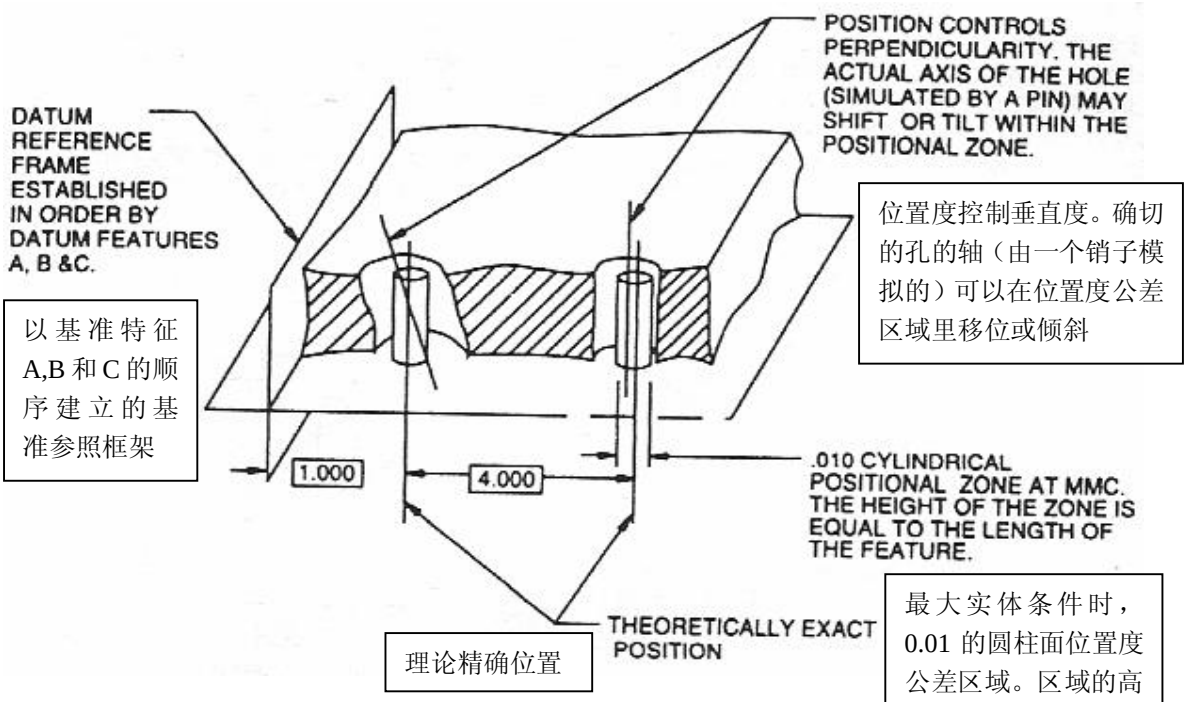
DIA FEATURE SIZE	DIA TOL ZONE
.529	.010
.530	.011
.531	.012
↓	↓
.537	.018
.538	.019

注：为得到关于修正符更多的信息请参见前文所述的材料条件修正符的影响

POSITION TOLERANCING 位置度公差标注

Position tolerancing related to datums controls location and orientation to the datums. In this case, the position tolerancing controls the perpendicularity of the holes. The axes of the holes may shift or tilt as long as they lie within the positional tolerance zones.

/和基准相关的位置度公差标注控制对于基准的位置和方向。在这个情况下，位置度控制孔的垂直度。孔的轴可以在位置度区域长度上移位或倾斜。

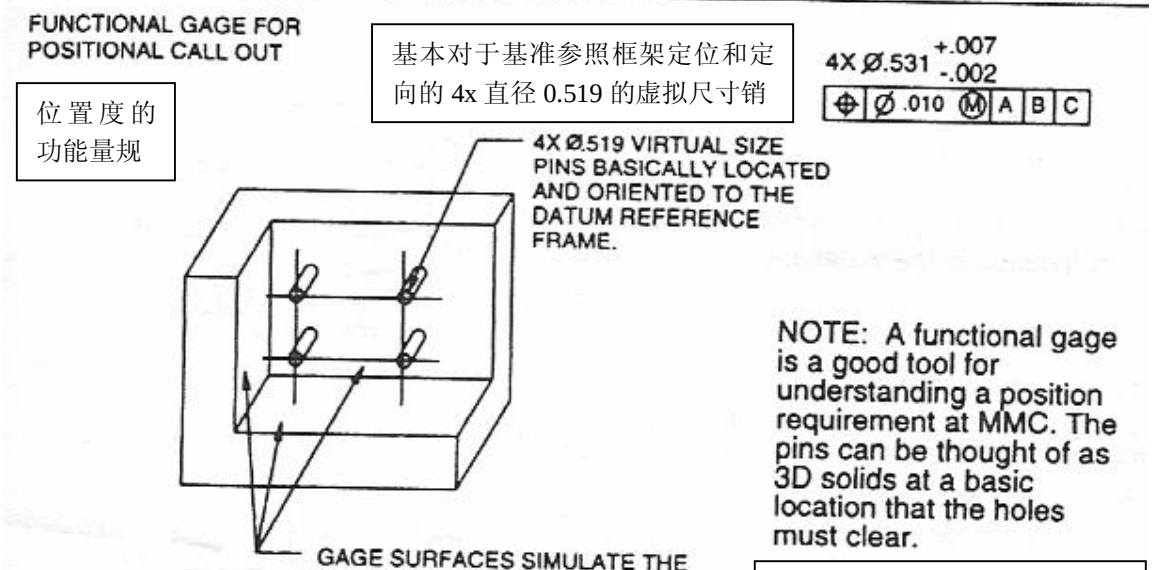


位置度控制垂直度。确切的孔的轴（由一个销子模拟的）可以在位置度公差区域里移位或倾斜

以基准特征 A,B 和 C 的顺序建立的基准参照框架

理论精确位置

最大实体条件时，0.01 的圆柱面位置度公差区域。区域的高度等于特征的长度



位置度的功能量规

基本对于基准参照框架定位和定向的 4x 直径 0.519 的虚拟尺寸销

量规曲面模拟由基准特征 A,B 和 C 建立的基准参照框架

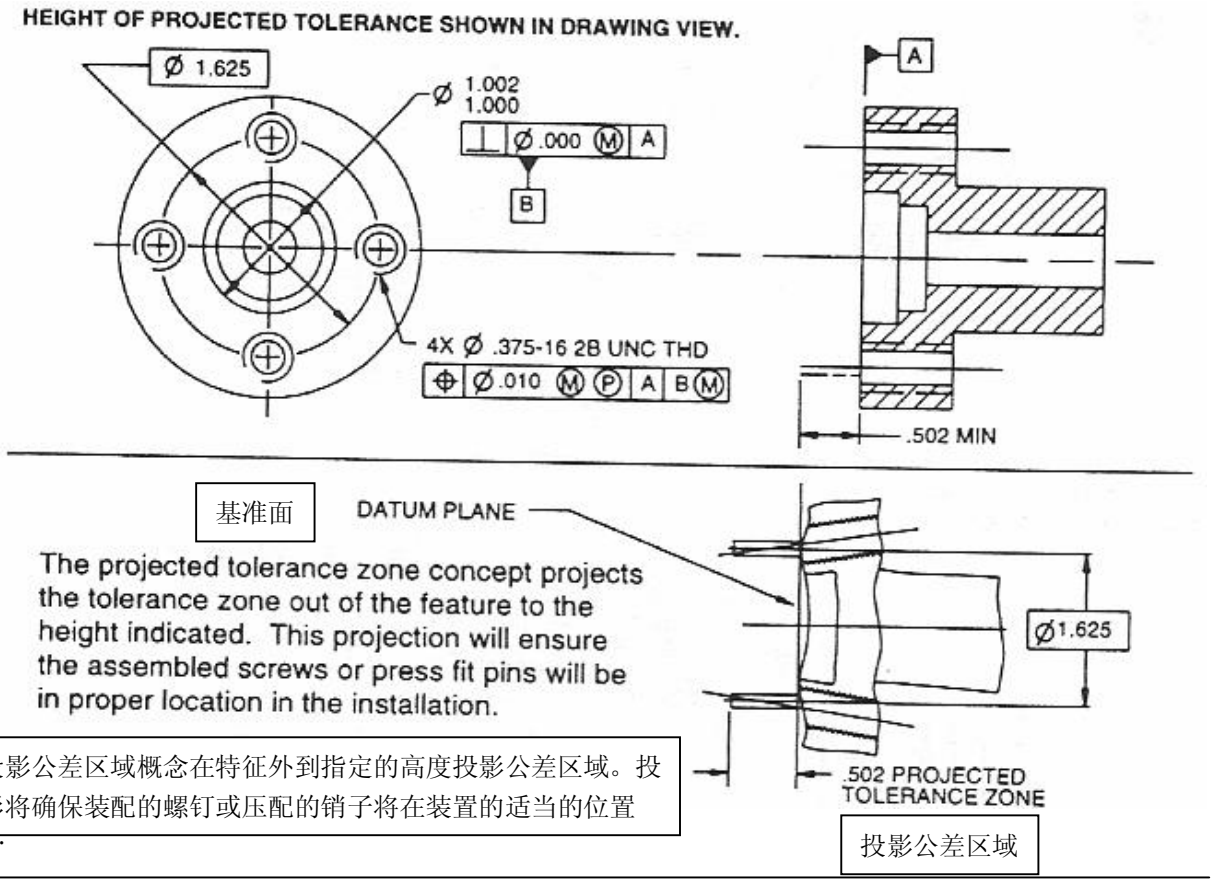
注：为了理解最大实体条件下的位置度要求，功能量规是一个好工具。 销子能被认为是孔位置必须基本清楚的 3D 实体。

Shown above is a sample functional gage that will verify the position call out. In addition, the size of the holes must also be verified. This can be completed with a simple go-no go type gage. The functional gage is shown for information only. Position can be verified with many other methods as well.
/以上显示的是一个样件功能量规，将验证位置度。另外，孔的尺寸也必须被验证。这可以用一个简单的止通规来验证。功能量规只作为信息显示。位置度也能用许多其它方法验证。

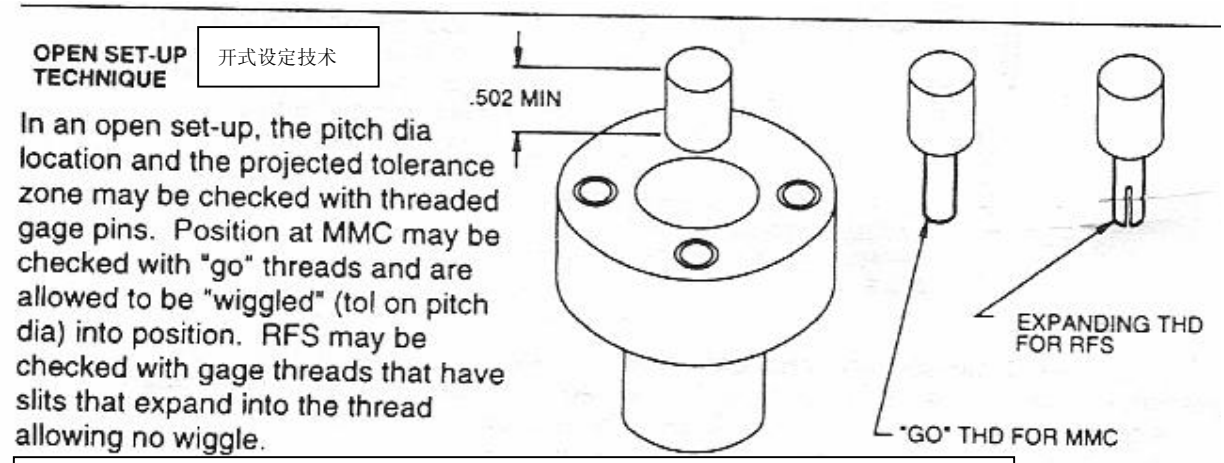
PROJECTED TOLERANCE ZONE APPLICATION/投影公差区域应用

All geometric tolerances applied to features extend for the full length and depth of the feature. In some cases it may be necessary to “project” a tolerance zone. The projected tolerance zone concept will “project “a tolerance zone out of the feature by a specified amount.
/所有应用到特征的几何公差延伸到特征的全部长度和深度。一些情况下，投影一个公差区域是必需的。投影公差区域概念将通过指定的数量在特征之外投影一个公差区域。
Where the direction of the projected tolerance zone is clear, only a symbolic call-out and height dimension is specified in the feature control frame. Where the direction of the projected tolerance zone is not clear, the symbolic call-out for the projected tolerance zone is still placed in the feature control frame following the feature tolerance and any modifier. The dimensional height of the projected tolerance zone, however, is shown in the drawing view with a heavy chain line that is drawn closely adjacent to the centerline of the hole.
/投影公差区域方向清晰的地方，仅仅有一符号，并且高度尺寸在特征控制框里被指定。投影公差区域方向不清晰的地方，投影公差区域的符号还放在特征控制框里，并紧随着特征公差和修正符。然而，投影公差区域的空间高度在工程图试图里显示一个粗虚线，紧挨着孔的中心线。

试图中显示的投影公差的高度



投影公差区域概念在特征外到指定的高度投影公差区域。投影将确保装配的螺钉或压配的销子将在装置的适当的位置



在一个开式设置上，节圆直径位置和投影的公差区域可以用螺纹规检查。最大实体条件时位置度可以用“通“螺纹检查，并且允许在位置度里（节圆直径的公差）“摆动“。RFS 可以用有狭长切口的螺纹规检查，这些切口延伸到不允许摆动的螺纹。

COMPOSITE TOLERANCING IS SPECIAL/复合公差是特别的

Composite tolerancing can be applied to both position and profile tolerancing. The position or profile symbol is entered once and is applicable to both horizontal entries. Composite tolerancing has a different interpretation than two single segmented feature control frames.

/复合公差标注能应用到位置度和轮廓度公差标注。位置度和轮廓度符号输入一次，并且应用到两个水平条目。复合公差标注和两个单独的特征控制框相比，有不同的解释。

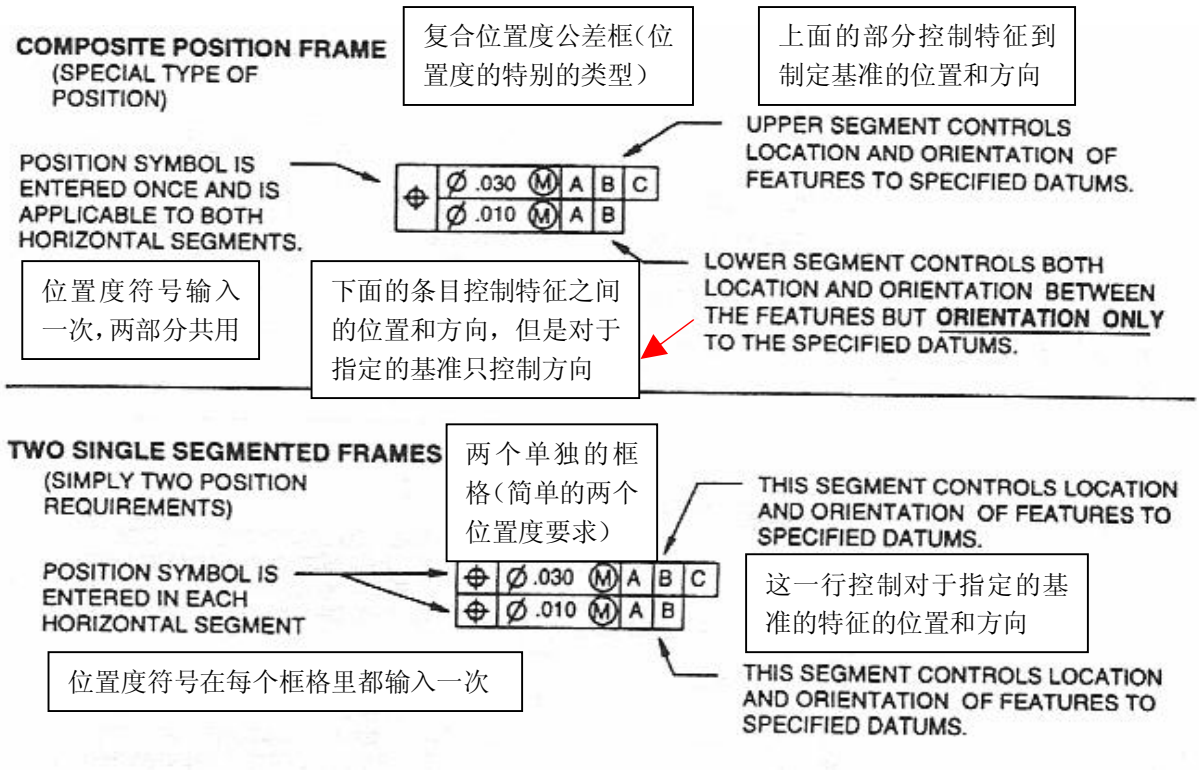
The upper segment on a position composite feature control frame controls location and orientation to the datums. The lower segment on a position composite controls location between the features but only orients (not location) the features to the specified datums.

/在位置度复合特征控制框的上面部分控制到基准的位置和方向。下面的部分除了控制对于指定基准的特征的方向（不是位置）以外，控制特征之间的位置。

In contrast, all single segmented position frames control both the location and orientation between the features as well as the specified datums. Two single position segmented controls are interpreted simply as two position requirements.

/与此相比，所有单独分割的位置度框格控制特征以及指定的基准之间的位置和方向。两个单独的位置度分割控制简单的解释为两个位置要求。

DIFFERENCE BETWEEN A COMPOSITE TOLERANCE AND 2 SINGLE SEGMENTED FEATURE CONTROL FRAMES/ 一个复合公差和两个单独分割的特征控制框的差



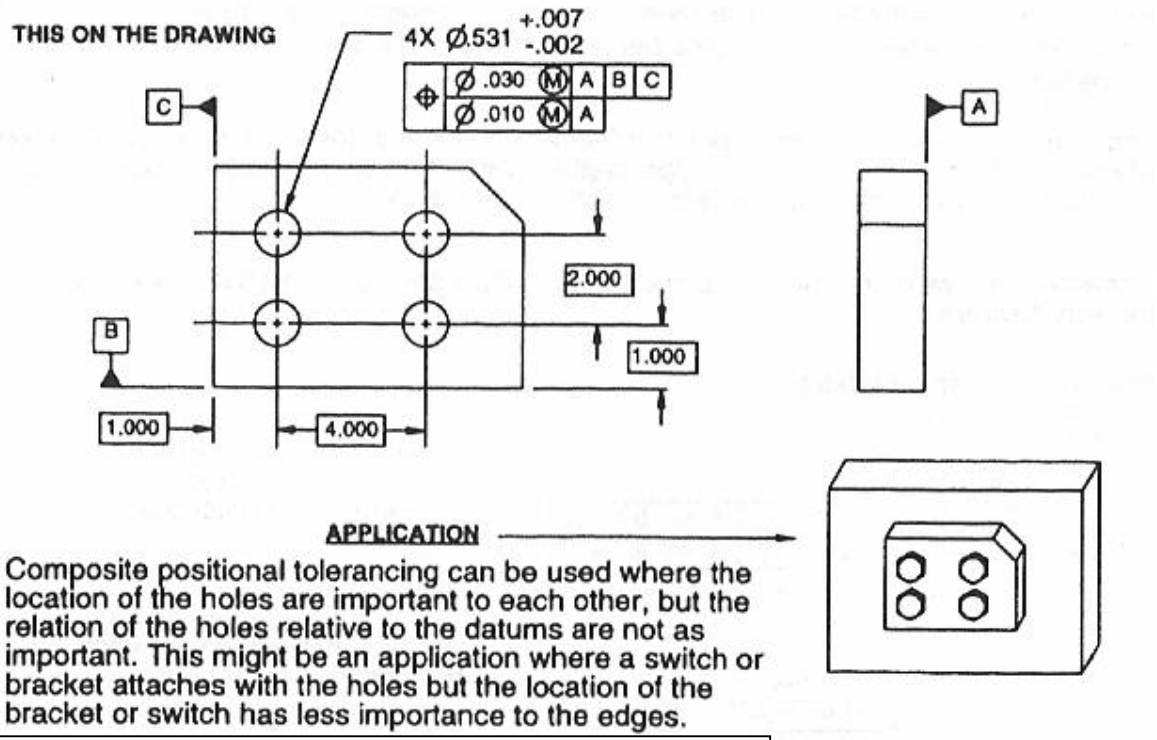
The difference between the terms location and orientation should be clear. Location locates features and is associated with basic linear dimensions. It can also include orientation. Orientation, on the other hand, is not associated with location or with basic linear dimensions, only basic angles. Orientation is usually thought of as parallelism, perpendicularity or angularity. The composite concepts explained above apply to both position and profile tolerances.

/术语位置和方向之间的不同应该是清楚的。位置用来定位特征，并且和基本的线性尺寸相关联。它也包括方向。另一方面，方向不和位置或基本线性尺寸相关联，只和基本角度相关联。方向通常认为是平行度，垂直度或倾斜度。以上的复合概念解释都应用到位置和轮廓公差。

COMPOSITE POSITION TOLERANCING – 1 DATUM FEATURE/复合位置度公差-一个基准特征

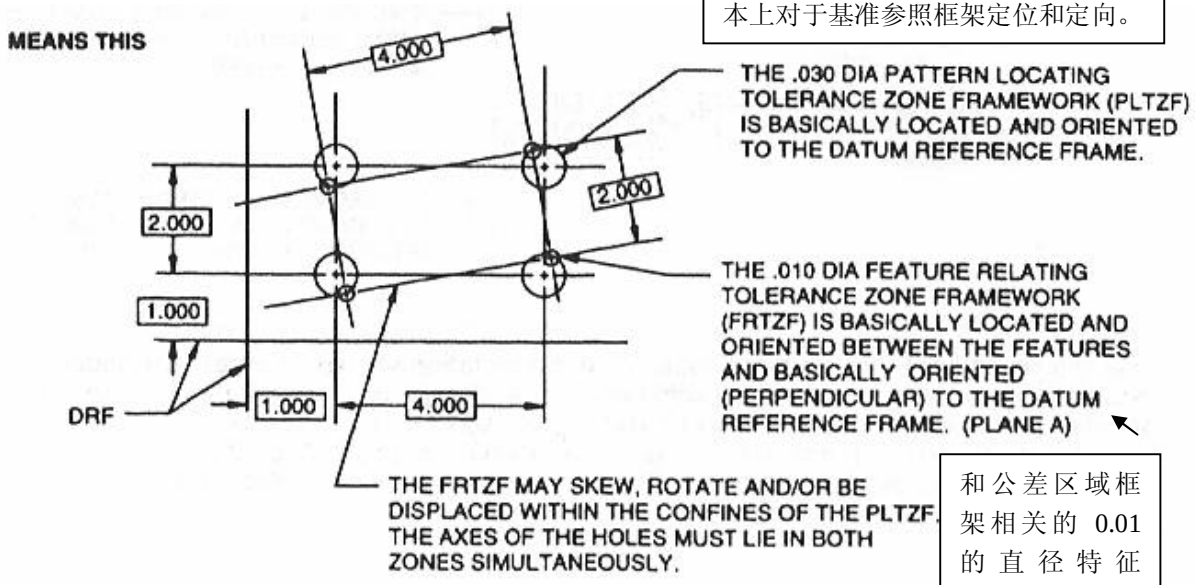
Composite positional tolerancing is a special method of locating holes. The position symbol is entered once and is applicable to both horizontal segments. In the example below, the upper segment locates and orients the holes to each other and the specified datums. The lower entry locates the holes to each other and orients (in this case, controls perpendicularity) the holes to the specified datums.

/复合位置度公差标注是一个定位孔的特别的方法。位置度符号输入一次，并且应用到两个水平的分割部分。在下面的例子里，对于其它的孔和指定的基准，上面的部分对孔定位和定向。 对于指定的基准，相对于其它孔和方向（在这种情况下，控制垂直度），下面的条目对孔定位。



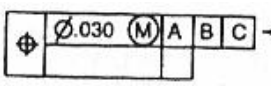
复合位置度的公差标注能应用在孔的相互之间位置重要的地方，但是孔相对于基准的关系不那么重要。这个应用可能是一个开关或和孔固定在一起的支架，但是支架或开关对边的位置不太重要。

0.03 直径公差区域框架（PLTZF）基本上对于基准参照框架定位和定向。



FRTZF 可能歪斜，旋转和/或取代 PLTZF 的限制。孔的轴必须同时位于两个区域里。

和公差区域框架相关的 0.01 的直径特征（FRTZF）基本上在特征之间定位和定向，并且基本上对于基准参照框架（平面 A）定向。



THE UPPER SEGMENT SPECIFIES THE PATTERN LOCATION TO THE SPECIFIED DATUMS.

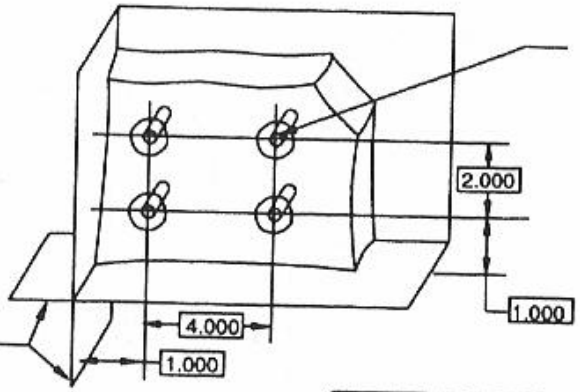
上面一行指定对于指定基准的布置位置

THE HOLES MUST MEET BOTH REQUIREMENTS

孔必须满足两个要求

1. PATTERN REQUIREMENT

布置要求



FIRST:
THE AXES OF THE HOLES MUST LIE WITHIN A .030 DIA AT MMC PATTERN LOCATION ZONE THAT IS BASICALLY LOCATED TO THE DATUM REFERENCE FRAME

第一：
孔的轴必须在 MMC 时位于一个直径 0.03 的位置公差区域里，这个公差区域基本对于基准参照框定位。

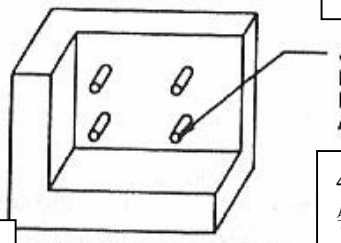
DRF ESTABLISHED BY DATUM FEATURES A, B & C.

通过基准特征 A,B 和 C 建立 DRF

样件功能量规

SAMPLE FUNCTIONAL GAGE.

The composite control requires two checks. Shown is a sample functional gage to verify the upper segment. The holes must also be verified for size.

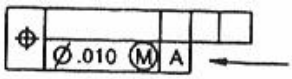


4X .499 DIA VIRTUAL SIZE PINS AND BASICALLY LOCATED TO EACH OTHER AND THE DRF.

4 个直径 0.499 的虚拟尺寸销子，并且基本对于互相之间和 DRF 定向.

复合的控制要求两个检查。为了验证上面的公差，图示是一个样件功能量规。孔也必须验证尺寸。

下面一行指定孔对孔的要求和这个要求对于所声明的基准的定向

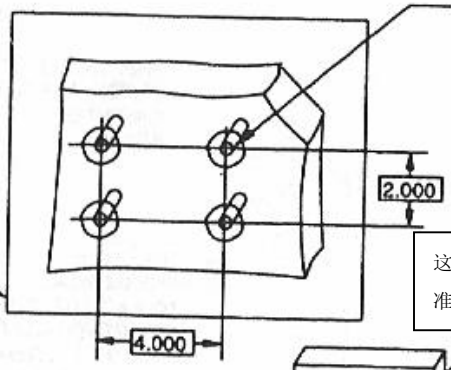


THE LOWER SEGMENT SPECIFIES THE HOLE TO HOLE REQUIREMENT AND THE ORIENTATION OF THIS REQUIREMENT TO THE STATED DATUMS.

第二： MMC 时孔的轴必须位于 0.01 直径特征相关区域（孔对孔）里。

2. FEATURE RELATING REQUIREMENT

特征相关要求



SECOND:
THE AXES OF THE HOLES MUST LIE WITHIN A .010 DIA AT MMC FEATURE RELATING ZONE (HOLE TO HOLE). THIS FEATURE RELATING ZONE IS BASICALLY ORIENTED TO THE DRF. SINCE DATUM A IS REFERENCED IN THE LOWER ENTRY, PERPENDICULARITY IS ALSO CONTROLLED.

这个特征相关区域基本上对于 DRF 定向。既然基准 A 参照下面的条目，垂直度也被控制。

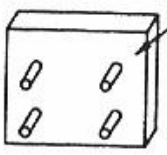
DRF ESTABLISHED BY DATUM FEATURE A.

通过基准 A 建立的 DRF

样件功能量规，显示的是下面部分的样件功能量规

SAMPLE FUNCTIONAL GAGE

Shown is a sample functional gage for the lower segment



4X .519 DIA VIRTUAL SIZE PINS BASICALLY LOCATED TO EACH OTHER

4 个直径 0.519 的虚拟尺寸销子基本上互相定位

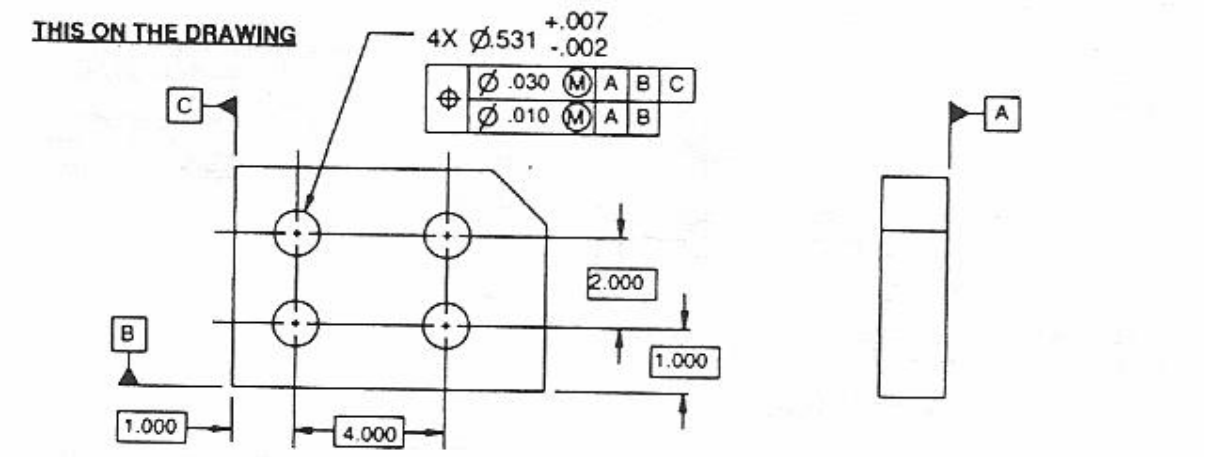
Since the MMC feature modifier is specified, additional positional tolerance is available for both the pattern and hole to hole specification as the holes depart from MMC.

既然指定 MMC 特征修正符，当孔偏离 MMC 时，对于布置和孔对孔的规范可得到额外的位置度公差

COMPOSITE POSITION TOLERANCING -2 DATUM FEATURES/复合位置度公差-两个基准特征

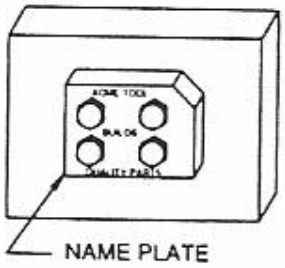
Composite position tolerancing can be applied with two datum features in the lower segment. In the example below, the upper segment locates and orients the pattern of holes to the datums. The lower segment locates the holes to each other and orients (in this case, perpendicular to A and parallel to B) the holes to the specified datums.

/复合位置度公差标注能应用到下面一行，带有两个基准特征。在下面的例子里，上面一行相对于基准对孔阵列定位和定向。下面一行相对于其它的孔和对指定基准的孔的方向，对孔定位（在这种情况下，对于 A 的垂直度和对于 B 的平行度）。



APPLICATION

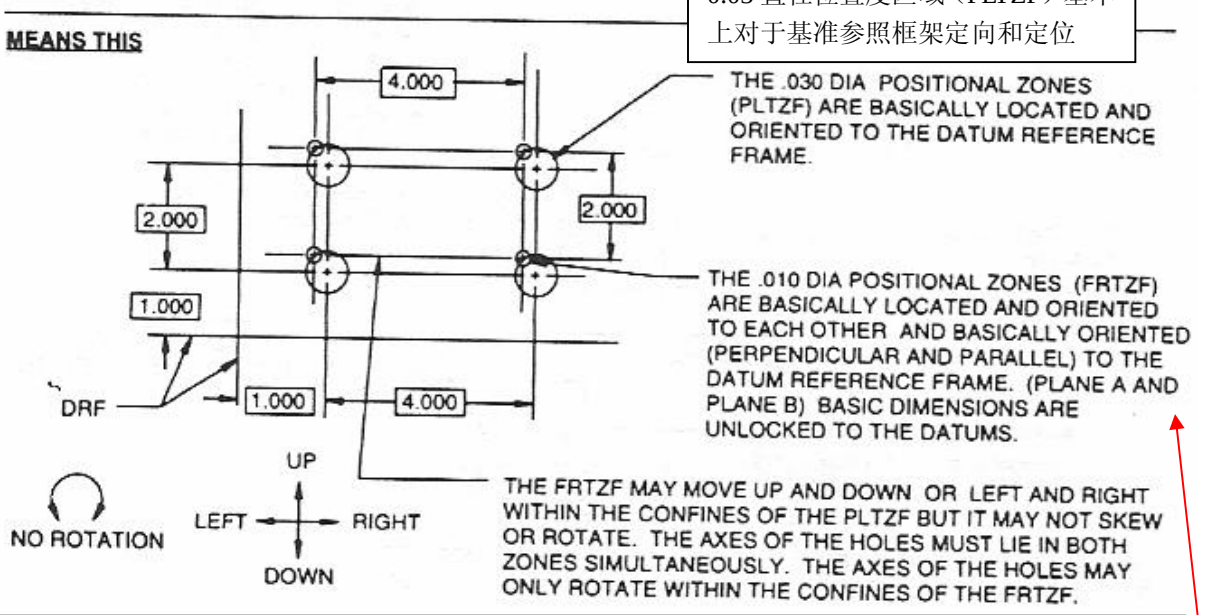
Composite positional tolerancing with 2 datums in the lower segment can be used where the location of the holes are important to each other, but the relation of the holes, relative to the datums, are not as important. The group of holes, however, may shift back and forth and up and down but may not rotate relative to the DRF as much as specified in the upper segment. This might be an application where a name plate attaches with the holes. The location of the name plate or gage is unimportant, but it can not mount crooked or skewed.



名牌

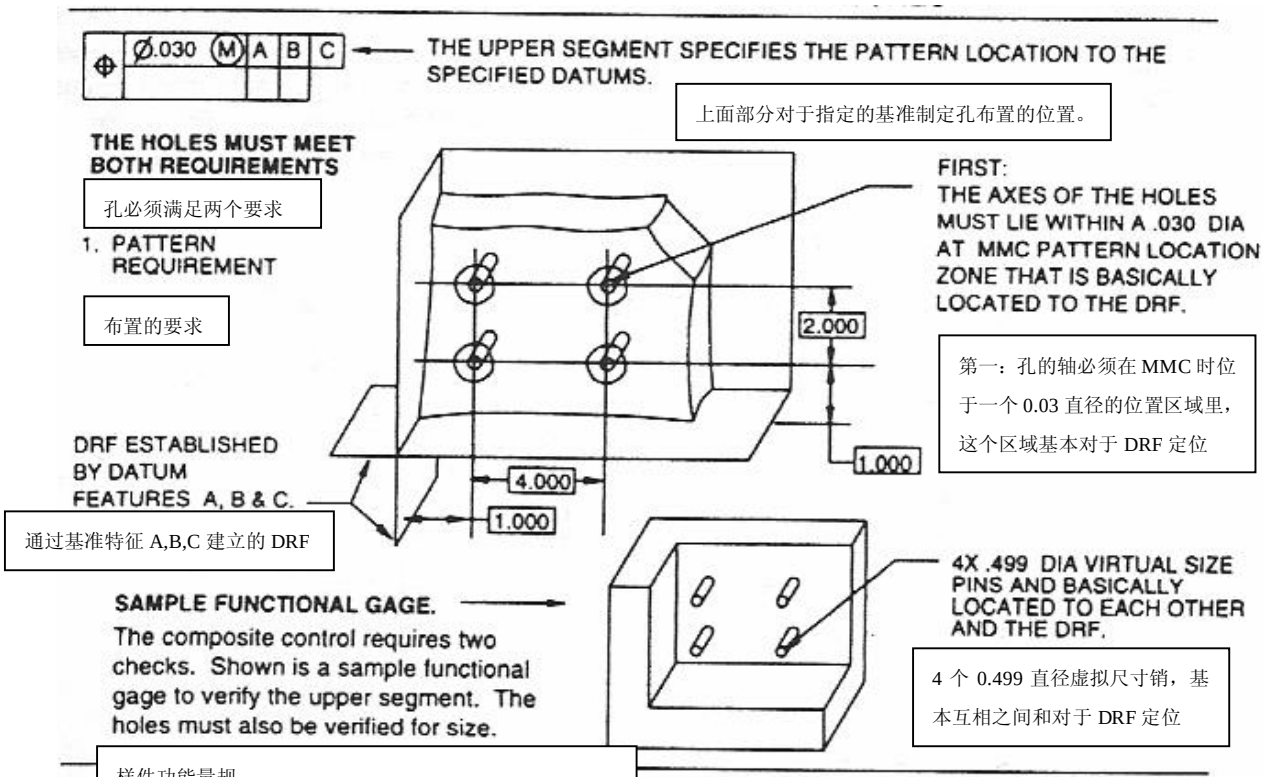
在下面一行有两个基准的复合位置度公差可能用在孔的位置互相之间重要的地方，但是相对于基准的孔的关系不是那么重要。然而，这组孔可以前后，上下移动，但是不可以相对于 DRF 旋转上面一行指定的那么多。这可能是一个名牌上有孔的地方的应用。名牌或量规的位置不重要，但是不可装成弯的或偏斜的。

0.03 直径位置度区域（PLTZF）基本上对于基准参照框架定向和定位



FRTZF 可以在 PLTZF 的限制里上下或左右移动，但是不可以偏斜或旋转。孔的轴必须同时位于两个区域里。孔的轴只可以在 FRTZF 的限制里旋转。

0.01 直径位置度区域（FRTZF）基本上互相定位和定向，并且基本对于基准参照框定向（垂直度和平行度）。（平面 A 和平面 B）基本尺寸对于基准是开启的。



上面部分对于指定的基准制定孔布置的位置。

孔必须满足两个要求

布置的要求

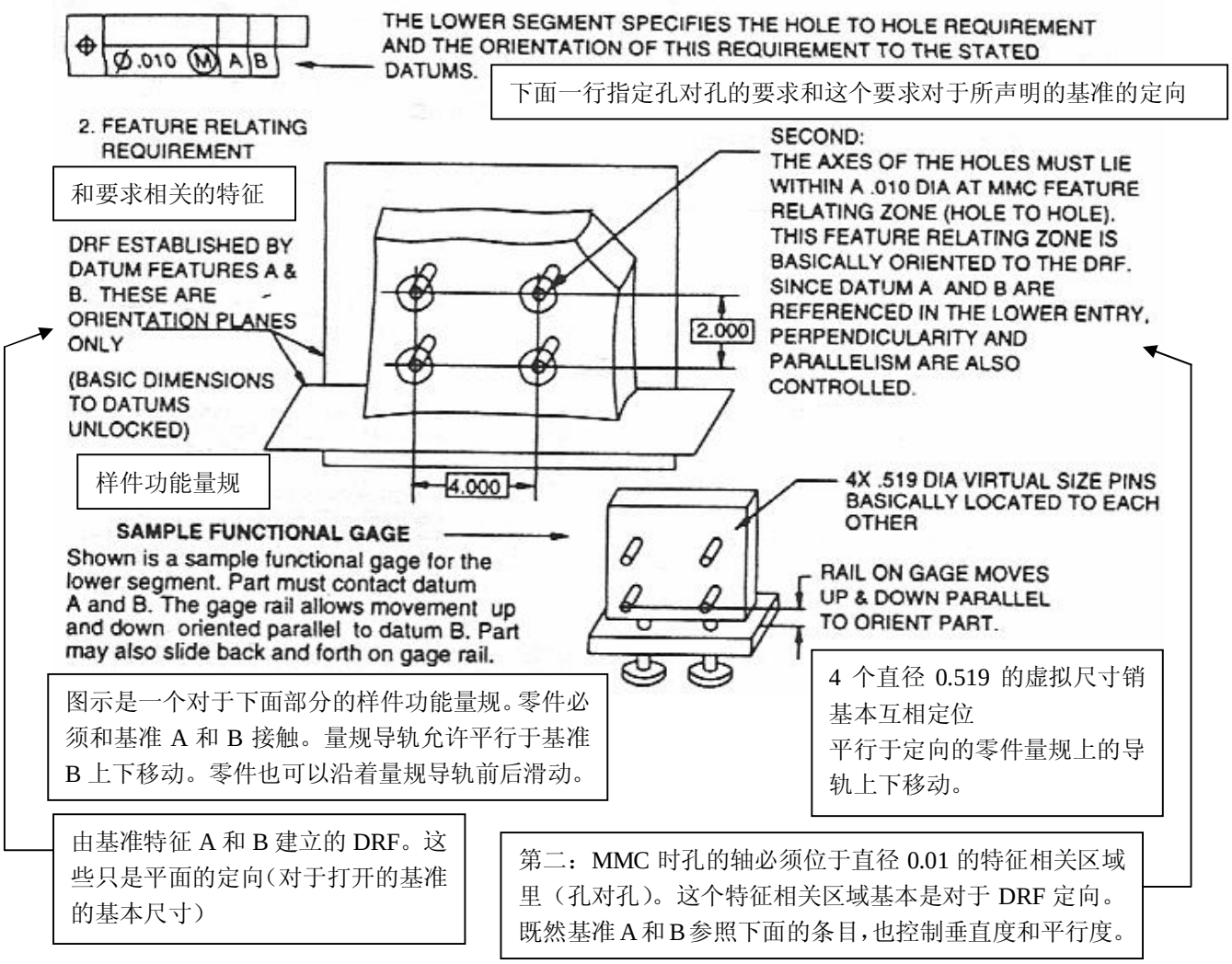
第一：孔的轴必须在 MMC 时位于一个 0.03 直径的位置区域里，这个区域基本对于 DRF 定位

通过基准特征 A,B,C 建立的 DRF

4 个 0.499 直径虚拟尺寸销，基本互相之间和对于 DRF 定位

样件功能量规。

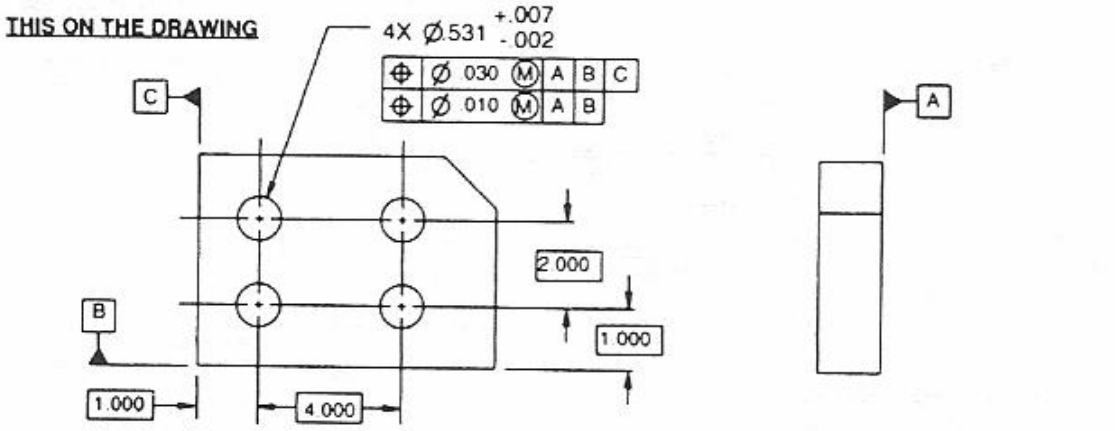
复合控制要求两个检查。显示的是一个为了验证上面一行的样件功能量规。孔的尺寸也必须被验证。



Note: if datum C were also entered in the lower segment, the interpretation would be the same. Since the lower entry is orientation only to the DRF, datum A and B are enough to establish complete orientation. The addition of datum C would be redundant. Since the MMC modifier was specified, additional positional tolerance is allowed for both the pattern and hole to hole specification as the features depart from MMC.
/注：如果在下面的部分也输入基准 C，解释是同样的。因为下面的条目只对 DRF 定向，基准 A 和基准 B 足以建立完整的方向。附加的基准 C 会是多余的。因为指定 MMC 修正符，对于孔和孔阵列偏离 MMC 的特征的规范允许附加的位置公差。

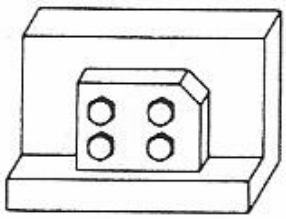
POSITION TOERANCING – 2 SINGLE SEGMENTED FRAMES/位置度公差-两个单独的分割框

In some cases, there may be a need to apply two single segmented position feature control frames to a feature. The part below illustrates this application. Single segmented position frames simply control position and orientation to the specified datums. Multiple single segmented frames may be used and datums are added or deleted as required.
/一些情况下，可能有对于一个特征用两个单独的分割的位置特征控制框的需求。下面的零件图示这种应用。单独的分割的位置框简单的控制对于指定基准的方向。可以用多样的单独分割框，并且根据 要求基准可以加上或删掉。



应用

APPLICATION

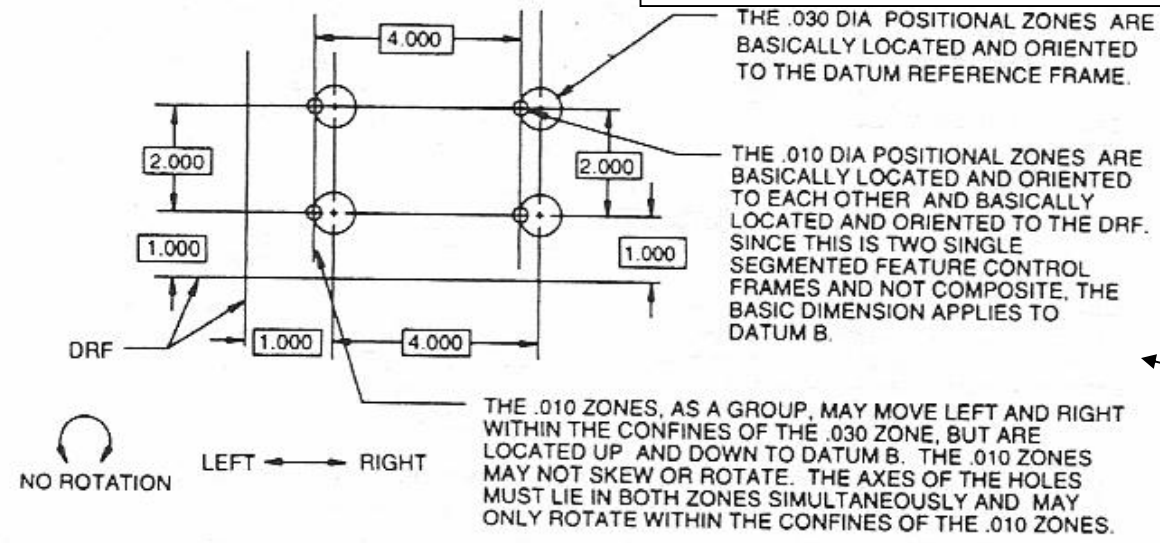


Two single segmented feature control frames can be used where the holes are important to each other. The holes are also more important to the datums in one direction than the other. The holes can move back an forth in a group but up and down the relationship is tighter.

两个单独分开的特征控制框能用于孔之间位置重要的地方。对于一个方向的基准孔也更重要。孔能整组前后移动，而不能上下移动

MEANS THIS

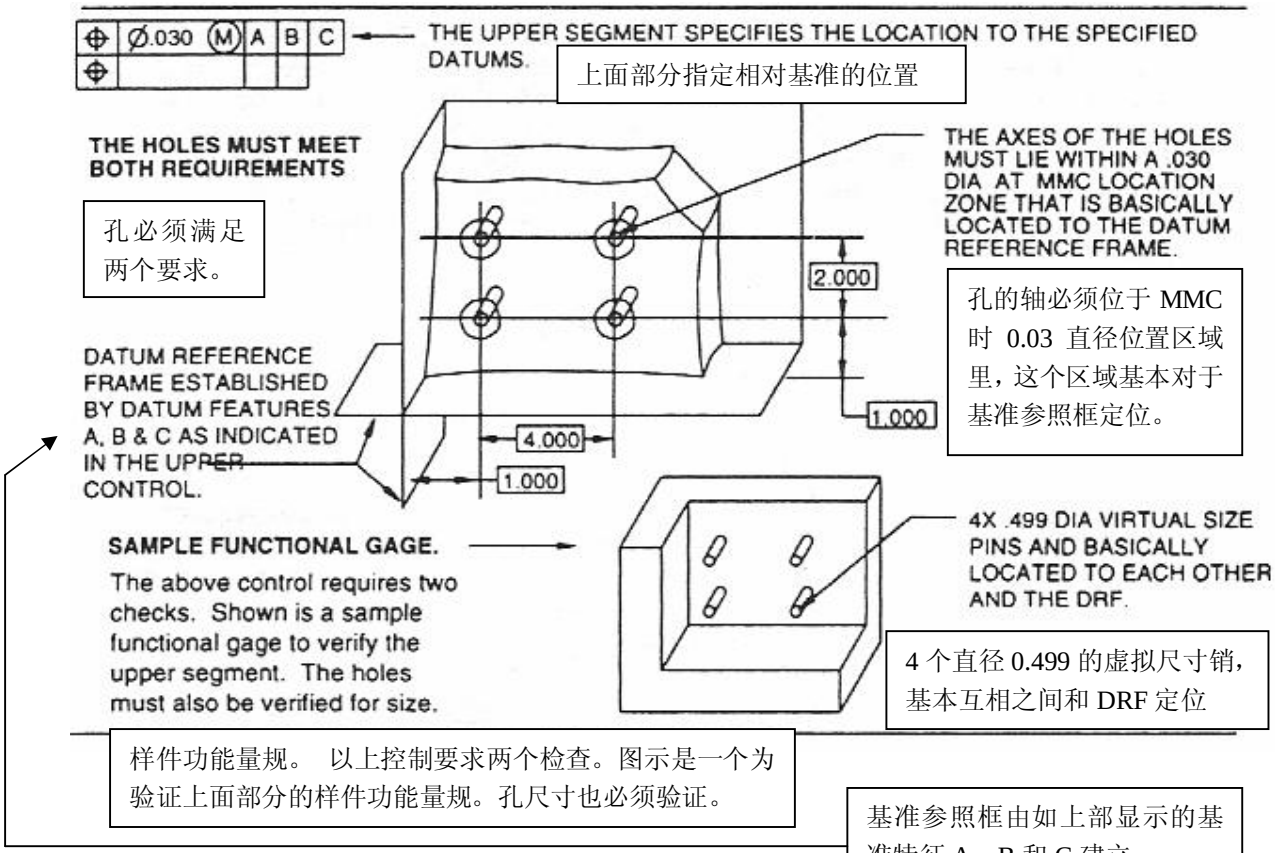
直径 0.03 位置度公差区域基本对于基准参照框定位和定向



0.01 的区域，作为一组，可以在 0.03 区域的限制里左右移动，但是对于基准 B 上下定向。0.01 区域不可以倾斜和旋转。孔的轴必须同时位于两个区域，并且只在 0.01 限制里旋转。

0.01 直径位置度区域基本互相定位和定向，并且也基本对于 DRF 定位和定向。既然这是两个单独的分开的特征控制框，不是复合的，基本尺寸应用到基准 B。

POSITION TOLERANCING – TWO SINGLE SEGMENTED FRAMES /位置度公差标注-两个单独的分割框



上面部分指定相对基准的位置

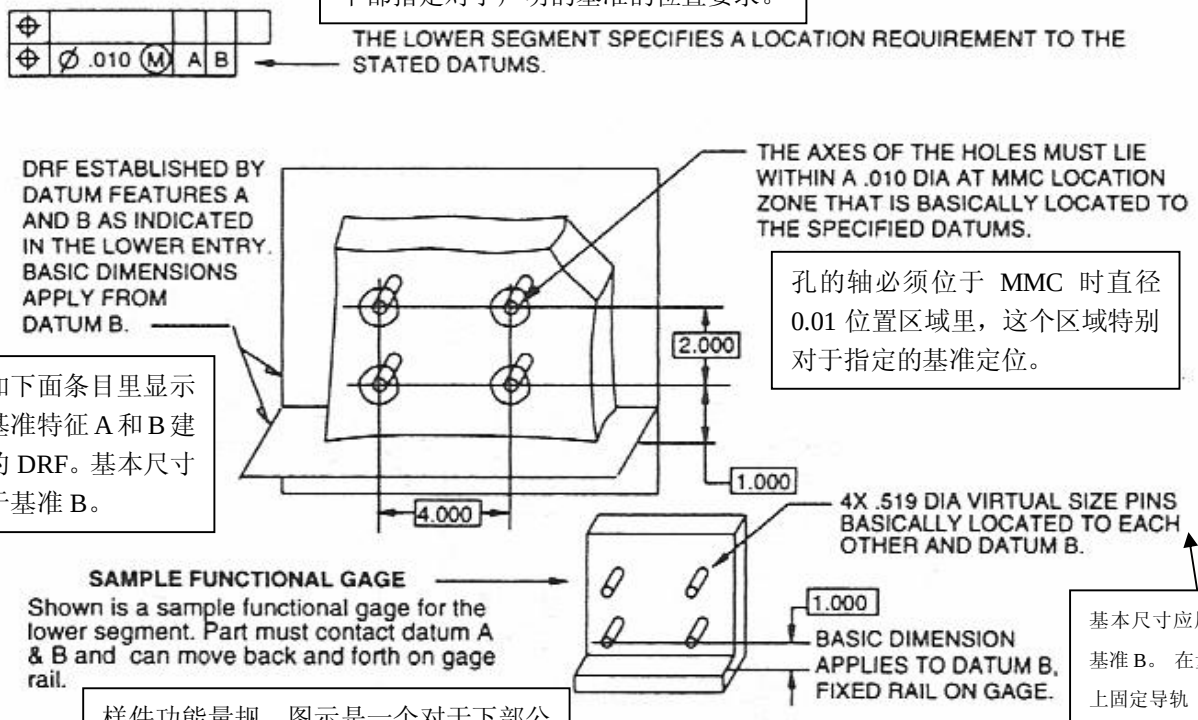
孔必须满足两个要求。

孔的轴必须位于 MMC 时 0.03 直径位置区域里，这个区域基本对于基准参照框定位。

4 个直径 0.499 的虚拟尺寸销，基本互相之间和 DRF 定位

样件功能量规。 以上控制要求两个检查。图示是一个为验证上面部分的样件功能量规。孔尺寸也必须验证。

基准参照框由如上部显示的基准特征 A，B 和 C 建立



下部指定对于声明的基准的位置要求。

由如下面条目里显示的基准特征 A 和 B 建立的 DRF。基本尺寸基于基准 B。

孔的轴必须位于 MMC 时直径 0.01 位置区域里，这个区域特别对于指定的基准定位。

基本尺寸应用到基准 B。 在量规上固定导轨

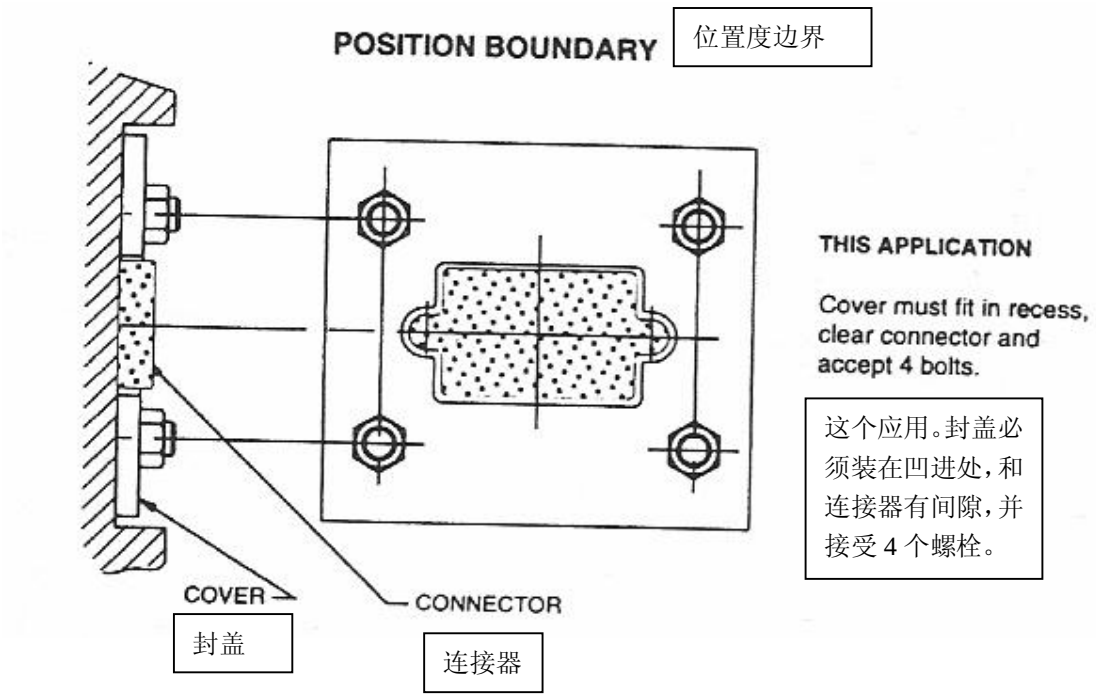
样件功能量规。图示是一个对于下部公差的样件功能量规。零件必须接触基准 A 和 B，并且能在量规导轨上前后移动

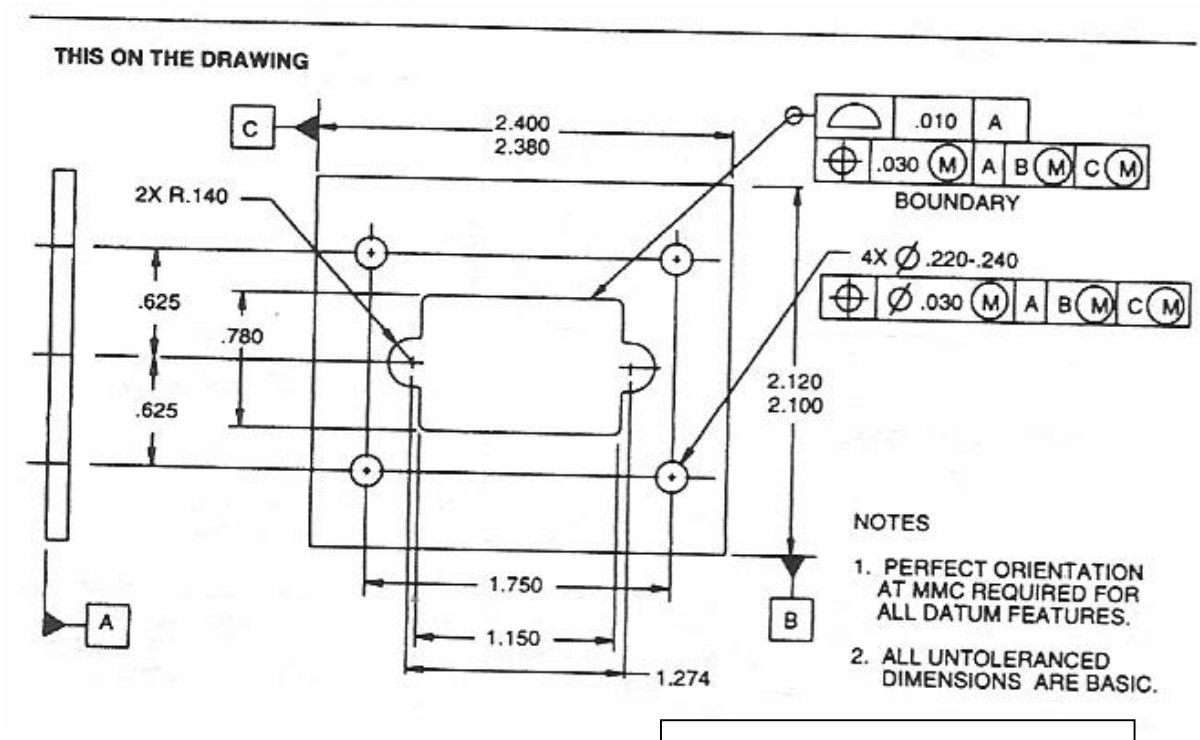
4 个直径 0.519 的虚拟尺寸销基本互相定位和对于基准 B 定位

NOTE: If datum C were also entered in the lower segment, there would be a conflict. Since both the upper and lower segment are location to the datums, there would be no need for the larger tolerance in the upper segment of the feature control frame. Since the MMC modifier is specified, additional positional tolerance is allowed for both the upper and lower segment as the features depart from MMC.
/注：如果在下面的部分也输入基准 C，会有一个冲突。因为上面和下面的部分都是对于基准的位置，在特征控制框里上面的部分里，对于更大的公差会没有需求。既然指定 MMC 修正符，对于上面和下面 的偏离 MMC 的特征分割框，允许附加的位置公差。

POSITION – BOUNDARY/位置度-边界

Position may be used to locate irregular features. The term BOUNDARY is placed under the feature frame. Rather than locating the axis or median plane of the feature, a virtual condition boundary is established. In the illustration below a datum reference frame is established by the flat surface and the height and width. The profile tolerance on the irregular opening defines the size, shape and orientation of the feature. The position boundary tolerance defines a boundary in which no element of the feature may lie. The feature modifiers MMC or LMC may be applied.
/位置度用于定位不规则的特征。术语边界放在特征控制框下面。不是定位特征的轴或中间的面，而是建立一个虚拟的条件边界。在下面的图示里，通过平曲面，高度和宽度建立一个基准参照框。不规则的开口的轮廓公差定义特征的尺寸，形状和方向。位置边界公差定义一个边界，在这个边界里没有特征元素。可以应用特征修正符 MMC 或 LMC。

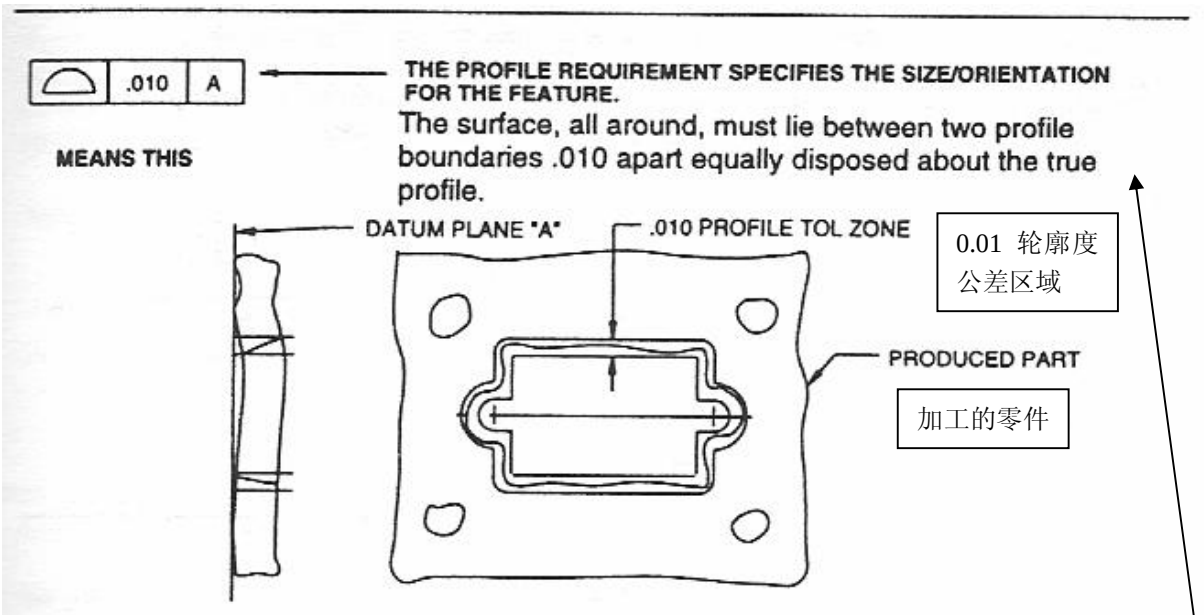




注：

1. 对于所有基准特征在 MMC 要求时精确定向。
2. 所有没有标公差的尺寸是基本的。

POSITION – BOUNDARY/位置度-边界



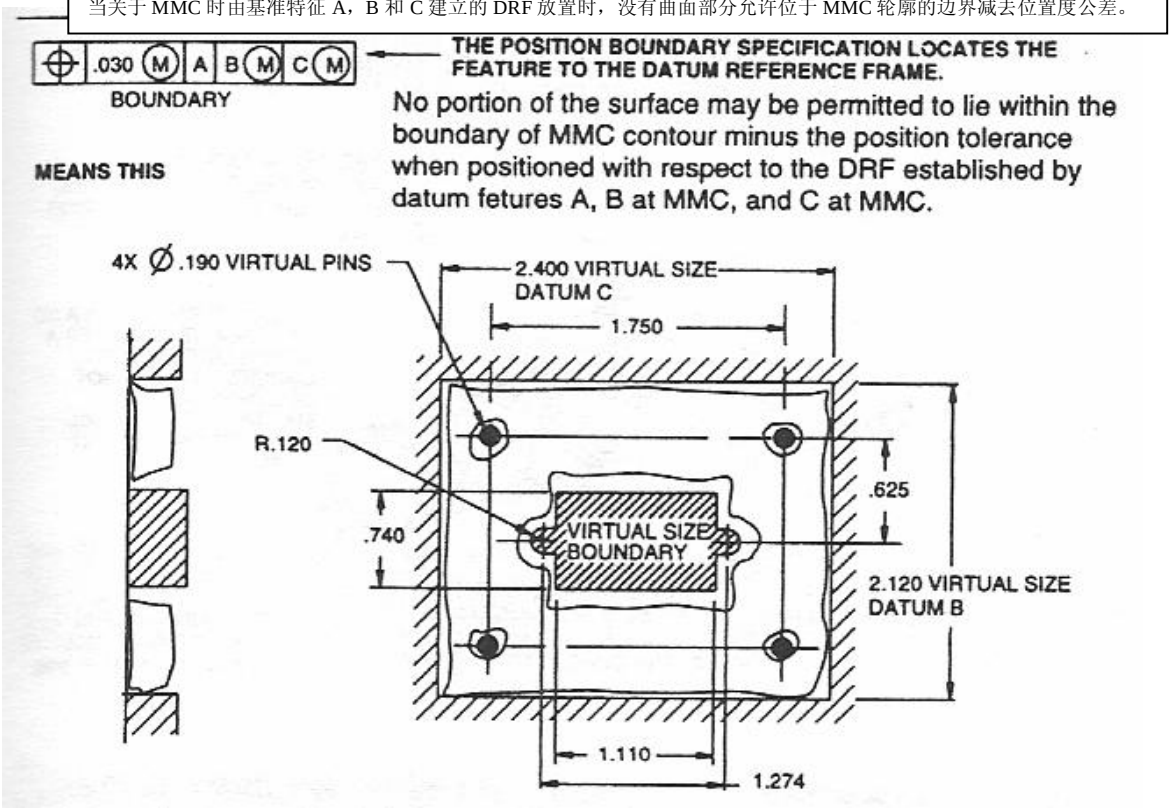
0.01 轮廓度公差区域

加工的零件

轮廓要求指定特征的尺寸和定向。
曲面，所有圆角，必须位于两个距离为 0.01，并关于真实轮廓均等分配的轮廓边界之间。

位置度边界规范相对于基准参照框定位特征。

当关于 MMC 时由基准特征 A，B 和 C 建立的 DRF 放置时，没有曲面部分允许位于 MMC 轮廓的边界减去位置度公差。



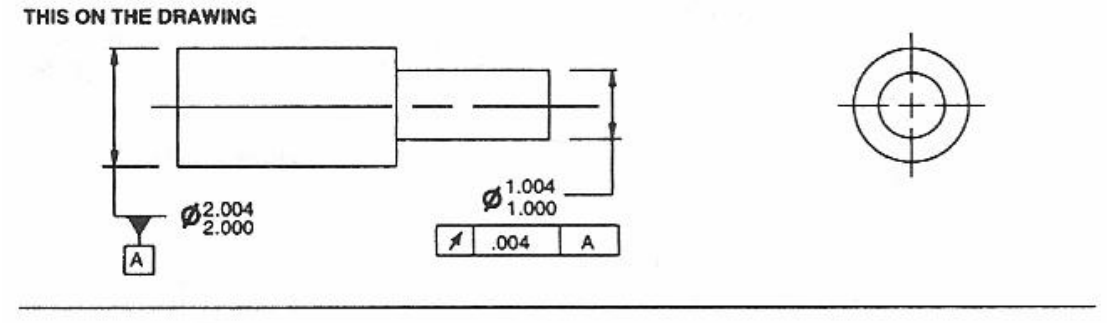
The position boundary concept is a similar, but somewhat different, concept than the composite profile approach to locating irregular features. Position boundary, when applied with MMC or LMC modifiers, defines only an inner or outer boundary in which no element of the feature must lie. Composite profile establishes both an inner and outer boundary in which the feature must lie.

/和定位不规则特征的复合轮廓度方法相比，位置度边界概念是一个相似的但又有点不同。应用 MMC 或 LMC 修正符时位置度边界只定义一个内部或外部边界，在这里没有特征元素。复合轮廓度建立一个有特征元素的内部和外部边界。

CIRCULAR RUNOUT/圆跳动

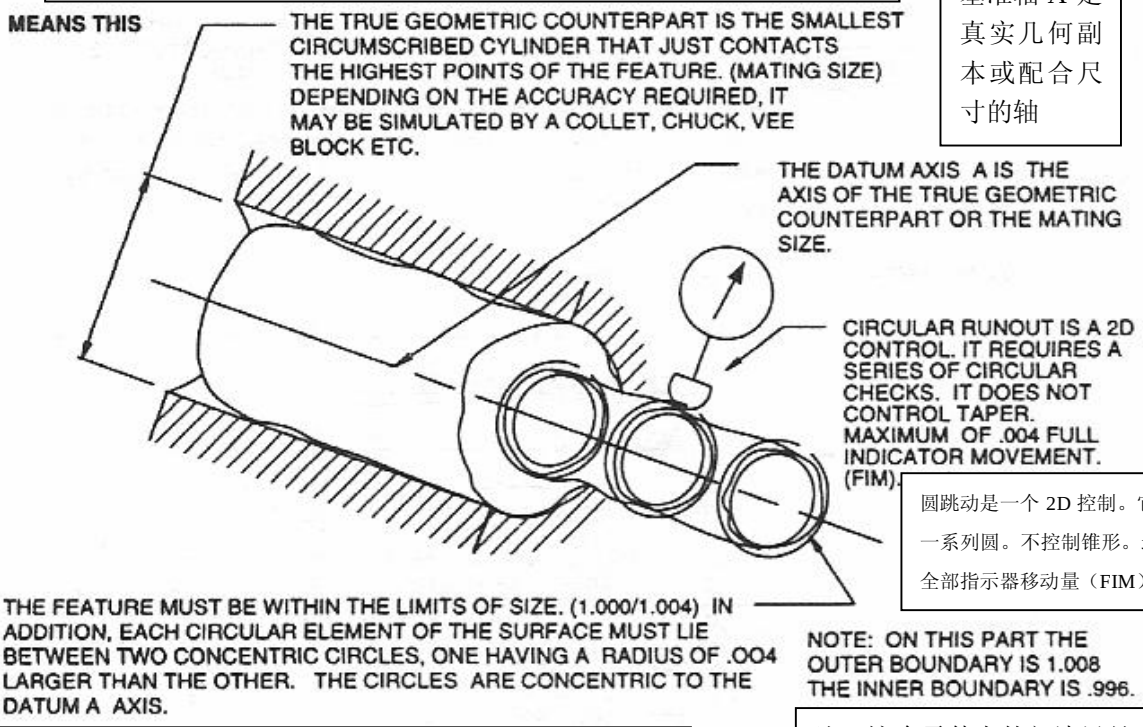
Circular runout is a two dimensional, surface to an axis control. The tolerance is applied independently at each circular cross section. When applied to a surface constructed around a datum axis, circular runout will control cumulative variations of circularity and coaxiality. Unlike total runout, it does not control taper.

/圆跳动是一个两维的对于控制轴的曲面。公差独立应用在每一个圆横截面。当应用 到围绕一个基准轴建构的曲面时，圆跳动将 控制同轴性的累积误差。和全跳动不同，它不控制锥形。



真实的几何副本是最小的外接圆柱，只和特征的最高点（配合尺寸）接触。依据要求的精度，可能用一个夹头，卡盘，V 型块等来模拟。

基准轴 A 是真实几何副本或配合尺寸的轴



圆跳动是一个 2D 控制。它要求检查一系列圆。不控制锥形。最大 0.004 全部指示器移动量（FIM）

特征必须在尺寸极限里（1.000/1.004）。另外，每个曲面的圆周元素必须位于两个同心圆之间。 一个比另一个半径大 0.004。圆对于基准 A 轴是同心的。

注：这个零件上外部边界是 1.008，内部边界是 0.996

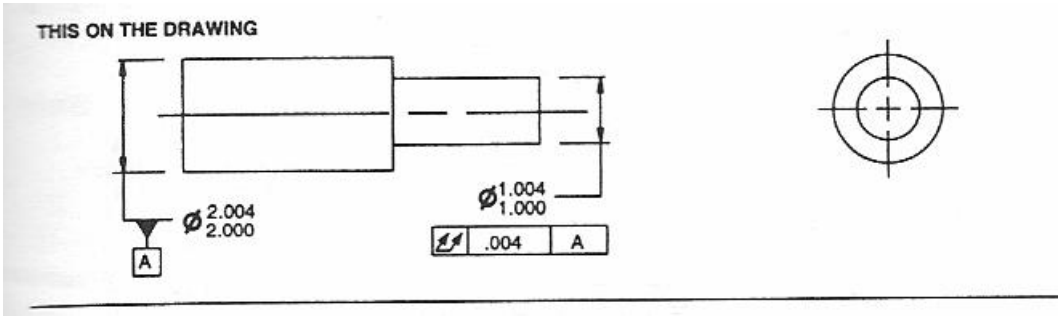
The circular runout specification may be verified with a dial indicator, CMM or by other methods. If a dial indicator is used, each circular cross section of the surface must lie within the specified runout tolerance (.004 full indicator movement) when the part is rotated 360 degrees about the datum axis. The indicator is reset at every location along the surface in a position normal to the true geometric shape. The feature must also be within the limits of size.

/圆跳动规范可以用千分表，坐标测量机或通过其它方法验证。如果用千分表，当零件围绕基准轴旋转 360 度时，每一曲面的圆横截面必须在指定的圆跳动公差里（指示器移动 0.004）。在每一个沿着垂直于真实几何形状方向的曲面位置，指示器要重新设定。

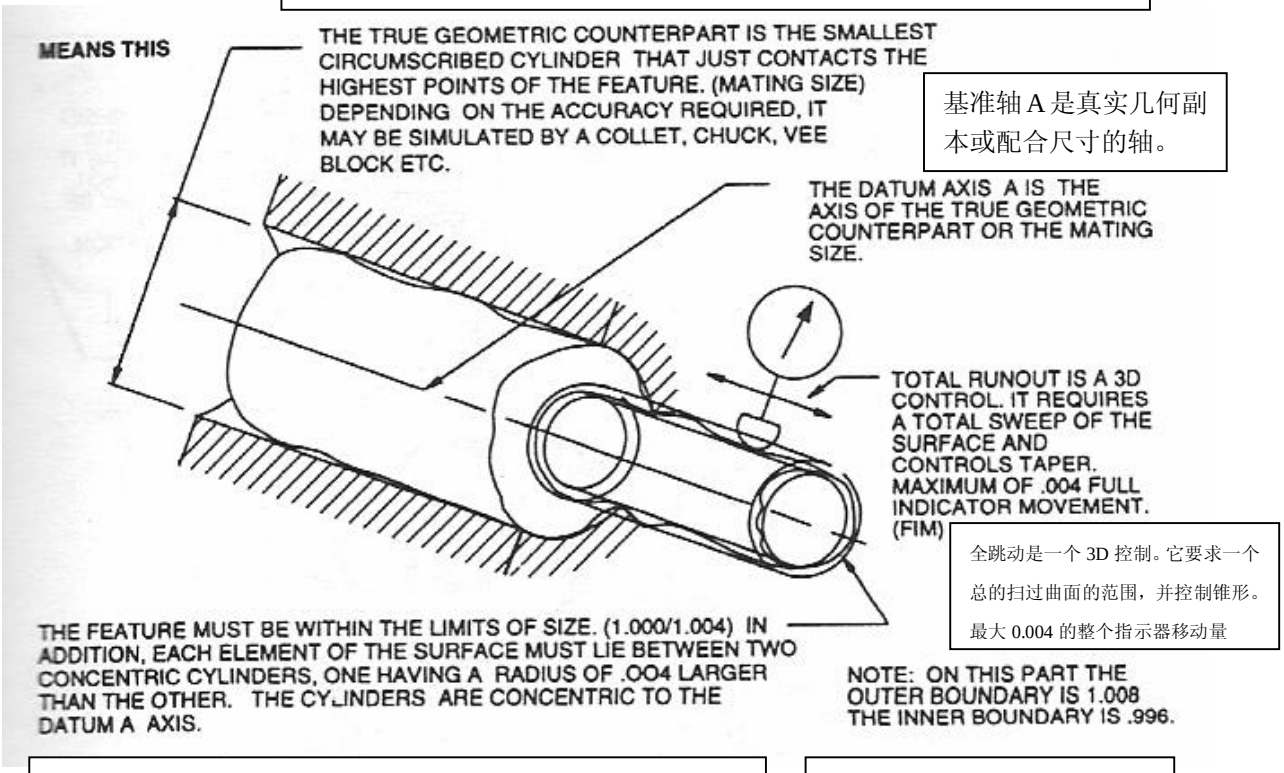
TOTAL RUNOUT/全跳动

Total runout is a three dimensional, surface to an axis control. Total runout provides a composite control of all surface elements. When applied to a surface constructed around a datum axis, total tunout will control the cumulative variations of circularity, straightness, coaxiality, angularity, taper and variations in the surface.

/全跳动是一个三维的，相对于控制轴的曲面。全跳动提供一个所以曲面元素的复合控制。当应用到一个围绕基准轴建构的曲面时，全跳动将控制同轴度，直线度，倾斜度，锥度的累积误差和曲面的变化。



真实几何副本是最小的外接圆柱，只和特征的最高点接触（配合尺寸）。依靠所要求的精度，可以用一个夹头，卡盘，V 型块等模拟。



基准轴 A 是真实几何副本或配合尺寸的轴。

全跳动是一个 3D 控制。它要求一个总的扫过曲面的范围，并控制锥形。最大 0.004 的整个指示器移动量

特征必须在尺寸极限里（1.000/1.004）。另外，每一个曲面的元素必须位于两个同心圆柱面之间。一个比另一个半径大 0.004。圆柱面对于基准 A 轴是同心的。

注：这个零件上外部边界是 1.008，内部边界是 0.996。

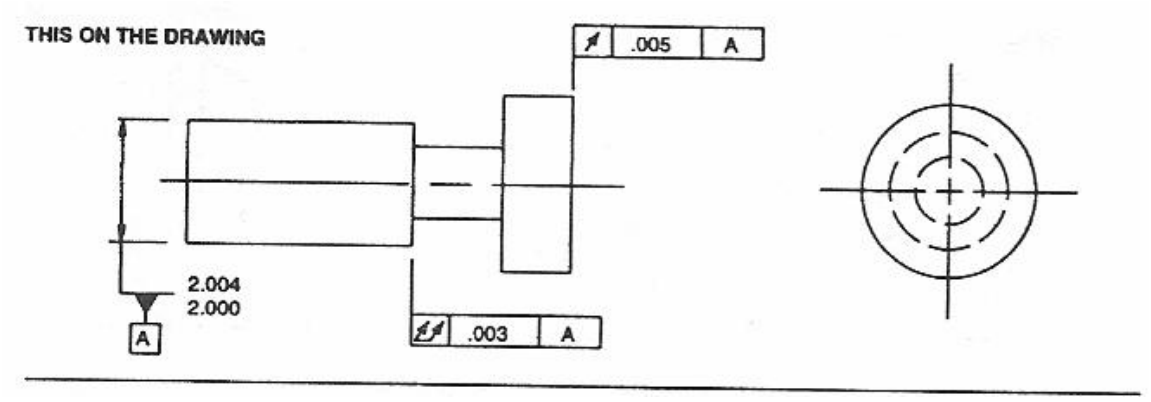
The total runout specification may be verified with a dial indicator, CMM or by other methods. If a dial indicator is used, the entire surface must lie within the specified runout tolerance (.004 full indicator movement) when the part is rotated 360 degrees about the datum axis. The indicator is placed at every location along the surface in a position normal to the true geometric shape without a reset of the indicator. The feature must also be within the limits of size.

/全跳动规范可以用千分表，坐标测量机或其它方法验证。如果用千分表，当零件围绕基准轴旋转 360 度时，全部曲面必须位于指定的跳动公差里（指示器移动时 0.004）。没有重新设定指示器时，指示器位于沿着垂直于真实几何形状的一个位置的曲面位置。特征也必须在尺寸极限里。

RUNOUT – APPLICATION TO SURFACES AT RIGHT ANGLES 跳动-对直角处曲面的应用

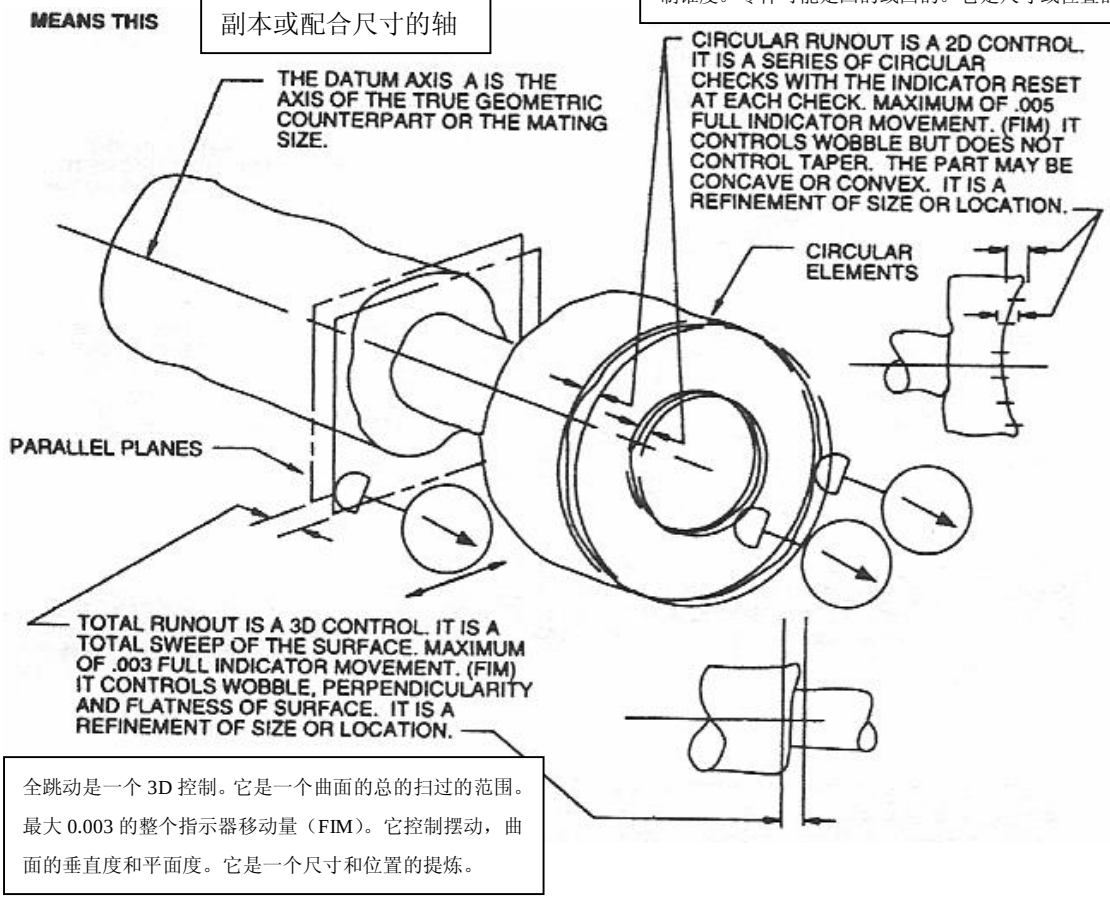
Both total runout and circular runout may be applied to surfaces constructed at right angles to a datum axis as shown below. Total and circular runout are a refinement of size and location tolerances. Total runout and perpendicularity will provide identical results and can be used interchangeably. Circular runout is a 2D specification and controls circular elements. The surface may be convex or concave within the size or location tolerance.

/全跳动和圆跳动都可以应用到对于下面所示的基准轴在直角处建构的曲面上。全跳动和圆跳动是一个尺寸和位置公差的精细化。全跳动和垂直度将提供同样的结果，并且可交替使用。圆跳动是一个 2 维规范，并控制圆形的元素。曲面在尺寸或位置公差里可以是凸起的或凹下的。



圆跳动是一个 2D 控制。它是用指示器设定对一系列圆的检查。最大 0.005 的整个指示器移动量（FIM）。它控制摆动，但不控制锥度。零件可能是凹的或凸的。它是尺寸或位置的提炼。

基准轴 A 是真实几何副本或配合尺寸的轴



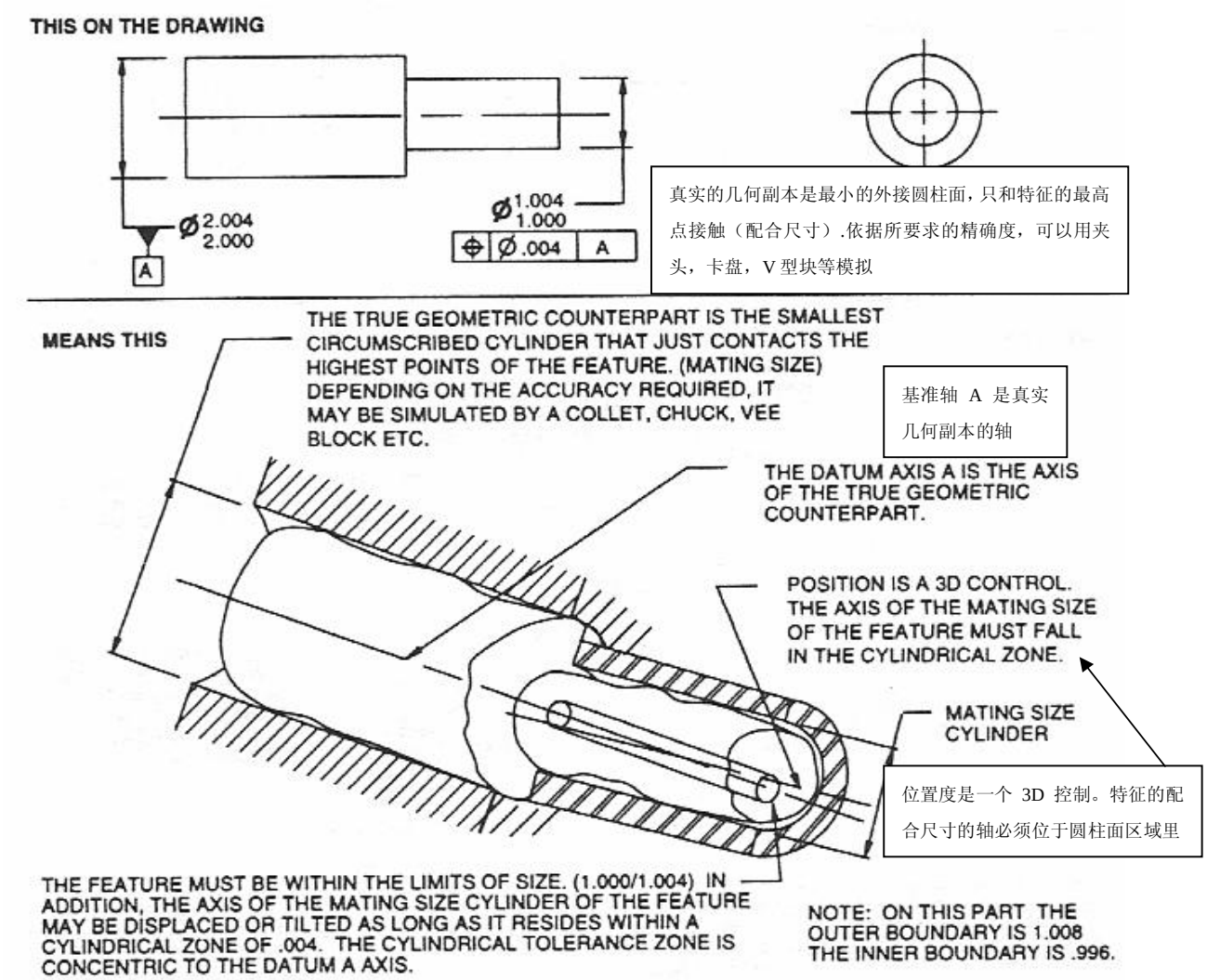
全跳动是一个 3D 控制。它是一个曲面的总的扫过的范围。最大 0.003 的整个指示器移动量（FIM）。它控制摆动，曲面的垂直度和平面度。它是一个尺寸和位置的提炼。

The total and circular runout specifications may be verified with a dial indicator, CMM or by other methods. If a dial indicator is used the specified surface and circular elements must lie within the specified runout tolerance when the part is rotated 360 degrees around the datum axis. In addition, the surfaces must be within the limits of size or location.

/全跳动和圆跳动规范可以用千分表，坐标测量机或通过其它方法验证。如果用一个千分表，当零件围绕基准轴旋转 360 度时，指定的曲面和圆元素必须位于指定的跳动公差里。另外，曲面必须位于尺寸或位置的极限里。

POSITON COAXIAL FEATURES/位置度同轴特征

Position for coaxial features is a three dimensional, axis to axis control. It defines a cylindrical zone within which the axis of the mating size of the feature must lie. When applied to a feature constructed around a datum axis, position controls orientation and location. It has no effect on size, form or variations in the surface. If required, the MMC, LMC or RFS modifier may be applied to the features as well as the datum feature.
/同轴特征的位置是一个三维的，轴到轴的控制。它定义一个圆柱区域，特征的配合尺寸轴线必须位于这个区域里。当应用到围绕一个基准轴建构的特征时，位置控制方向和位置。它对尺寸形状或曲面变异没有影响。如果需要，MMC，LMC 或 RFS 修正符可以应用到和基准特征一起的特征上。



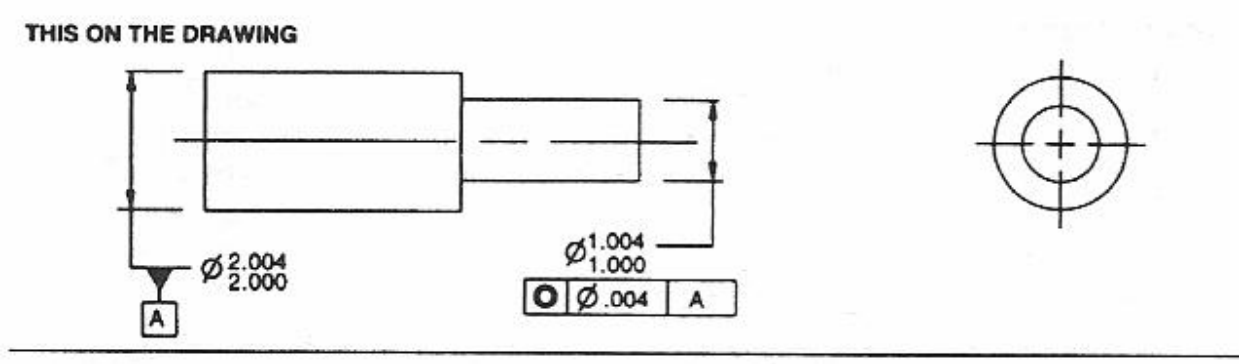
The coaxial position specification may be verified with a dial indicator, CMM or by other methods. If a dial indicator is used, and the full indicator reading does not exceed .004, the part is good. If the full indicator reading exceeds .004, there is a possibility that the part is still good, the reason for this is, as an indicator rides on the surface, it inadvertently reports a composite error of position, surface variation and form errors (ovality). Since

position is only an axis to axis control, surface variations are not included in the requirement. A ‘mapping’ and further evaluation of the surface variations are not included in the requirement. A ‘mapping’ and further evaluation of the surface might be required to accept features beyond be .004 full indicator movement.
/同轴的位置度规范可以用千分表，坐标测量机或其它方法验证。如果用千分表，全部指示器读数不超过 0.004，零件是好的。如果全部指示器读数超过 0.004，有一个可能零件还是好的，原因是，当指示器靠在曲面上时，它不注意的报告一个位置的复合错误，曲面变异和形状错误（椭圆度）.因为位置只是一个轴对轴的控制，曲面变异不包括在要求里。一个‘映射’和进一步曲面变异的评估不包括在要求里。一个‘映射’和曲面的进一步评估可以被要求在 0.004 指示器移动范围内接受特征。

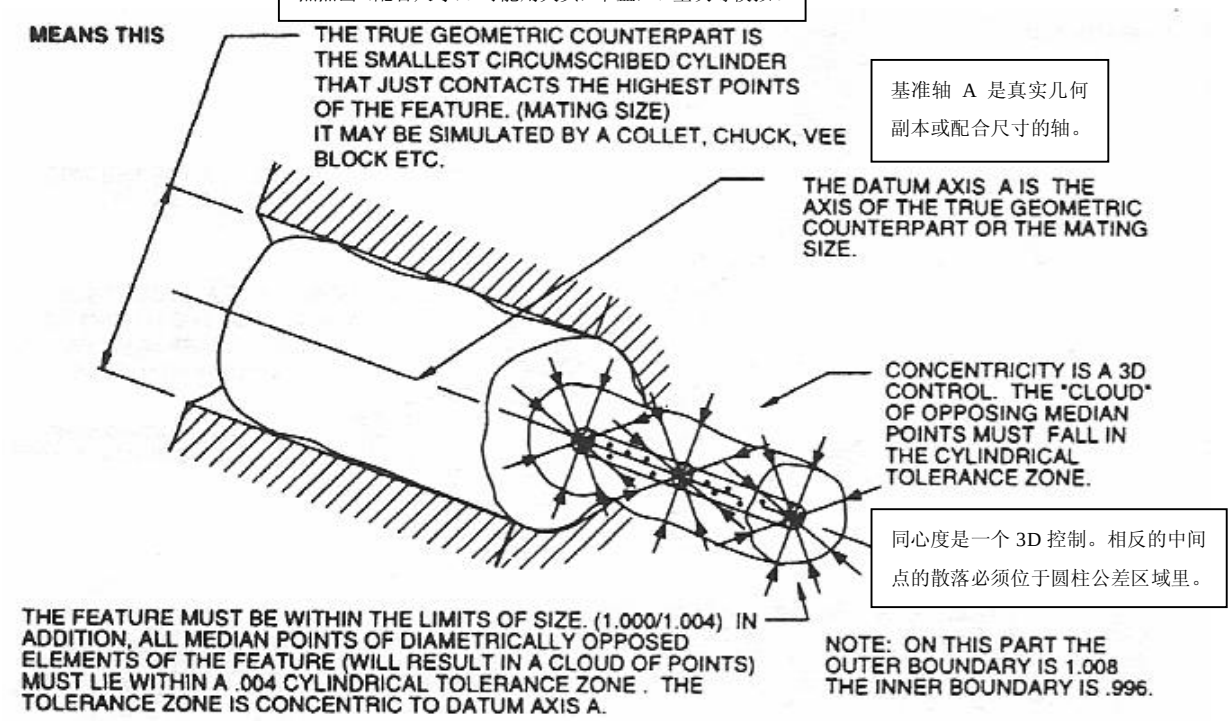
CONCENTRICITY/同心度

Concentricity is a three dimensional, opposed median points to an axis control. Concentricity will control location and can have some effect on the form and orientation of a feature. It will not control the form of perfectly oval parts but may have an impact on irregular or ‘D’ shapped features.
/同心度是一个三维的，对于一个轴控制的反对的中央的点。同心度将控制位置和可能对特征的形状和方向有一些影响。它不控制理想的椭圆零件的形状，但可以对不规则的或‘ D’形特征有影响。

The application of concentricity is complex and rare. If a coaxial relationship is required between features the user should consider position, runout or profile tolerances.
/同心度的应用是复杂的，并且不常用。如果在特征之间要求一个同轴关系，用户应该考虑位置度，跳动或轮廓度。



真实的几何副本是最小的外接圆柱面，只和特征的最高点杰出（配合尺寸）。可能用夹头，卡盘，V 型块等模拟。



基准轴 A 是真实几何副本或配合尺寸的轴。

同心度是一个 3D 控制。相反的中间点的散落必须位于圆柱公差区域里。

特征必须位于尺寸极限里（1.000/1.004）。另外，所有特征的作为直径的相反的元素的中间点（将导致一个点云）必须位于 0.004 的圆柱公差区域里。公差区域对于基准轴 A 是同心的。

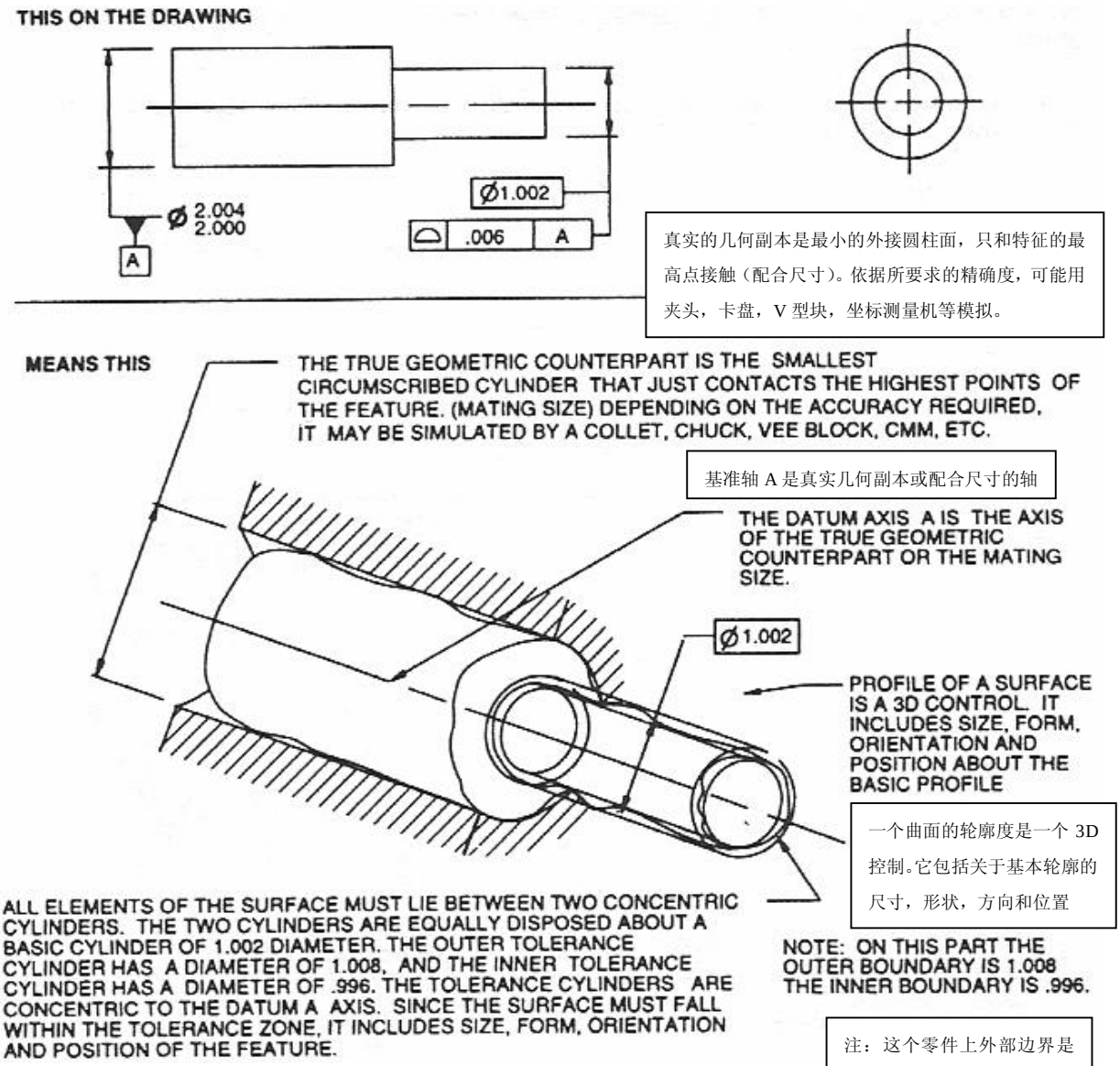
注：这个零件上外部边界是 1.008，内部边界是 0.996。

The concentricity specification may be verified with dial indicators, a CMM or by other methods. If dial indicators are used, two diametrically opposed, mastered indicators are placed on either side of the feature and positioned and rotated about the datum axis.
/同心度规范可以用千分表，坐标测量机或通过其它方法验证。如果用千分表，两个直径相反的，主要的指示器不是放在特征的一边，就是关于基准轴定位和旋转。

CAUTION: at present the ISO standards do not recognize the unique interpretation of concentricity and symmetry as defined in the ASME Y14.5M, 1994 standard. The concentricity and symmetry characteristics in ISO have the same interpretations as the position characteristic in the ASME Y 14.5M, 1994 standard.
/小心：目前，ISO 标准不认识 ASME Y14.5M，1994 定义的同心度和对称度的独特的解释。作为 ASMEY14.5，1994 标准里的位置特征，ISO 里同心度和对称度特征有同样的解释。

PROFILE OF A SURFACE – SURFACE TO AXIS CONTROL/曲面的轮廓度-对于轴控制的曲面

Profile of a surface for coaxial features is a three dimensional surface to an axis control. It defines a zone of tolerance (.006 wide on each side) that lies between two concentric cylinders that are equally disposed about a basic diameter. When applied to a surface constructed around a datum axis, profile of a surface will control cumulative variations of size, circularity, straightness, coaxiality, angularity, taper and variations in the surface.
/对于一个同轴的特征曲面的轮廓是一个对于一个轴控制的三维曲面。它定义一个公差区域（每边 0.006 宽），位于两个同轴圆柱面之间，这两个圆柱面倾斜于一个基本直径。当应用到一个围绕基准轴构建的曲面时，一个曲面的轮廓将控制尺寸的累积变化，同轴度，直线度，同心度，倾斜度，锥度和曲面里的变异。



真实的几何副本是最小的外接圆柱面，只和特征的最高点接触（配合尺寸）。依据所要求的精确度，可能用夹头，卡盘，V 型块，坐标测量机等模拟。

基准轴 A 是真实几何副本或配合尺寸的轴

一个曲面的轮廓度是一个 3D 控制。它包括关于基本轮廓的尺寸，形状，方向和位置

注：这个零件上外部边界是 1.008，内部边界是 0.996。

所有曲面的元素必须位于两个同心圆柱面之间。这两个圆柱面是关于一个基本的 1.002 直径的圆柱面均等分配的。外部公差圆柱面有一个 1.008 直径，内部公差圆柱面有一个直径 0.996。公差圆柱面对于基准轴 A 是同心的。既然曲面必须位于公差区域里，它包括特征的尺寸，形状，方向和位置。

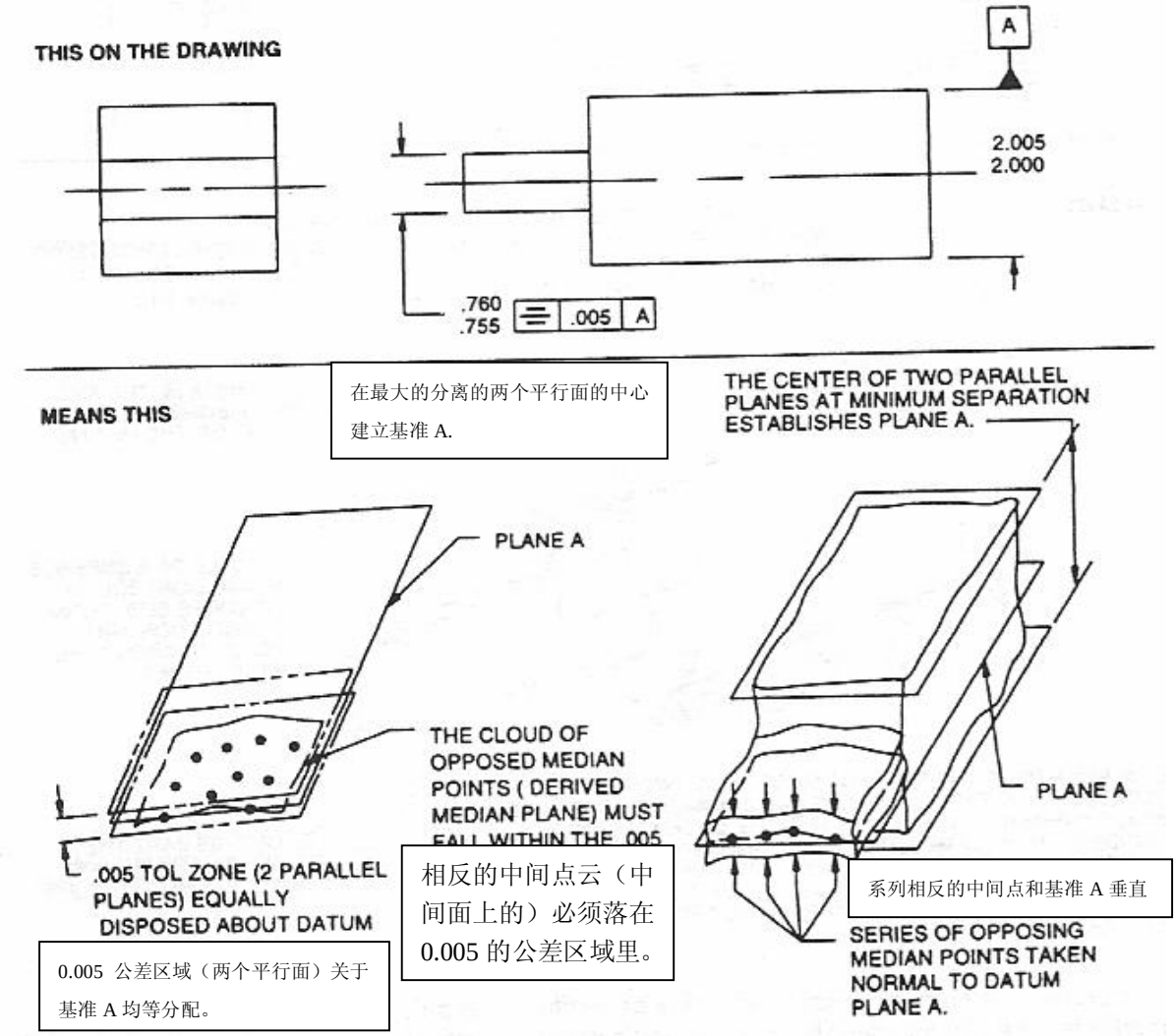
The profile of a surface specification may be verified with a dial indicator, CMM or other methods. If a dial indicator is used, the indicator is ‘mastered’ or set at the basic diameter of 1.002. The entire surface, including the size of the feature, must lie within the specified profile tolerance (.006 full indicator movement) when the part rotated 360degrees about the datum axis. The indicator is placed at every location along the surface in a position normal to the true geometric shape without a reset of the indicator. The size of the feature is contained within the profile tolerance.

/一个曲面规范的轮廓可以用一个千分表，坐标测量机或其它方法验证。如果用一个千分表，指示器是‘主要的‘或设定在基本直径 1.002 处。当零件围绕基准轴旋转 360 度时，全部曲面，包括特征尺寸，必须位于指定的轮廓度公差（0.006 指示器全移动）里。指示器在不重新设定情况下放在沿着每一垂直于真实几何形状位置的曲面位置。特征尺寸包含在轮廓度公差里。

SYMMETRY/对称度

Symmetry is a condition where the median points of all opposed elements of a feature are congruent with the axis or center plane of a datum feature. Symmetry is the same concept as concentricity except that it is applied to non-cylindrical feature. Symmetry differs from position in that it controls opposing points (derived median plane) where as position controls the center plane of the actual mating envelope. The application of symmetry is rare and is commonly misused. Irregularities in the form of an actual feature may make it difficult to establish the location of a feature's median points. Therefore, unless there is a definite need for the control of features median points, it is recommended that a control of position or profile be used.
/对称度是一个条件，一个特征所有的相反的元素的中间的点是适合轴或基准特征中间面的。除了应用到非圆柱特征以外，和同心度相比对称度是同样的概念。对称度不同于位置度，位置度控制相反的点（偏离中间平面），作为位置度控制真实的配合包络轨迹的中间平面。对称度的应用是很少的，并且一般会被误用。真实特征的形状的不规则可以使建立特征的中间点的位置困难。因此，除非有明确的对特征中间点的控制的需求，建议使用位置度或轮廓度的控制。

NOTE: symmetry in ISO standards is not interpreted as shown below. Symmetry in ISO standards is interpreted the same as position.
/注：在 ISO 标准里的对称度不是下面所示的解释。ISO 标准里的对称度和位置度解释是一样的。

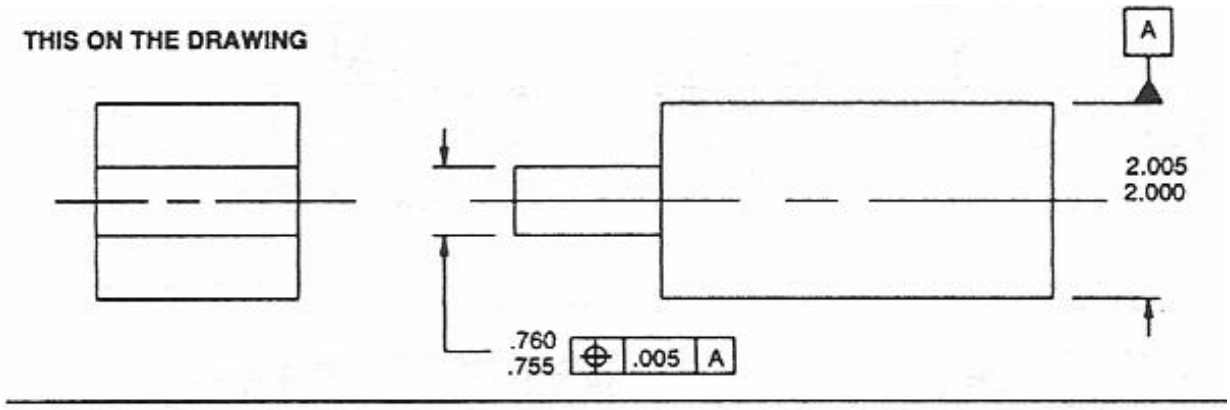


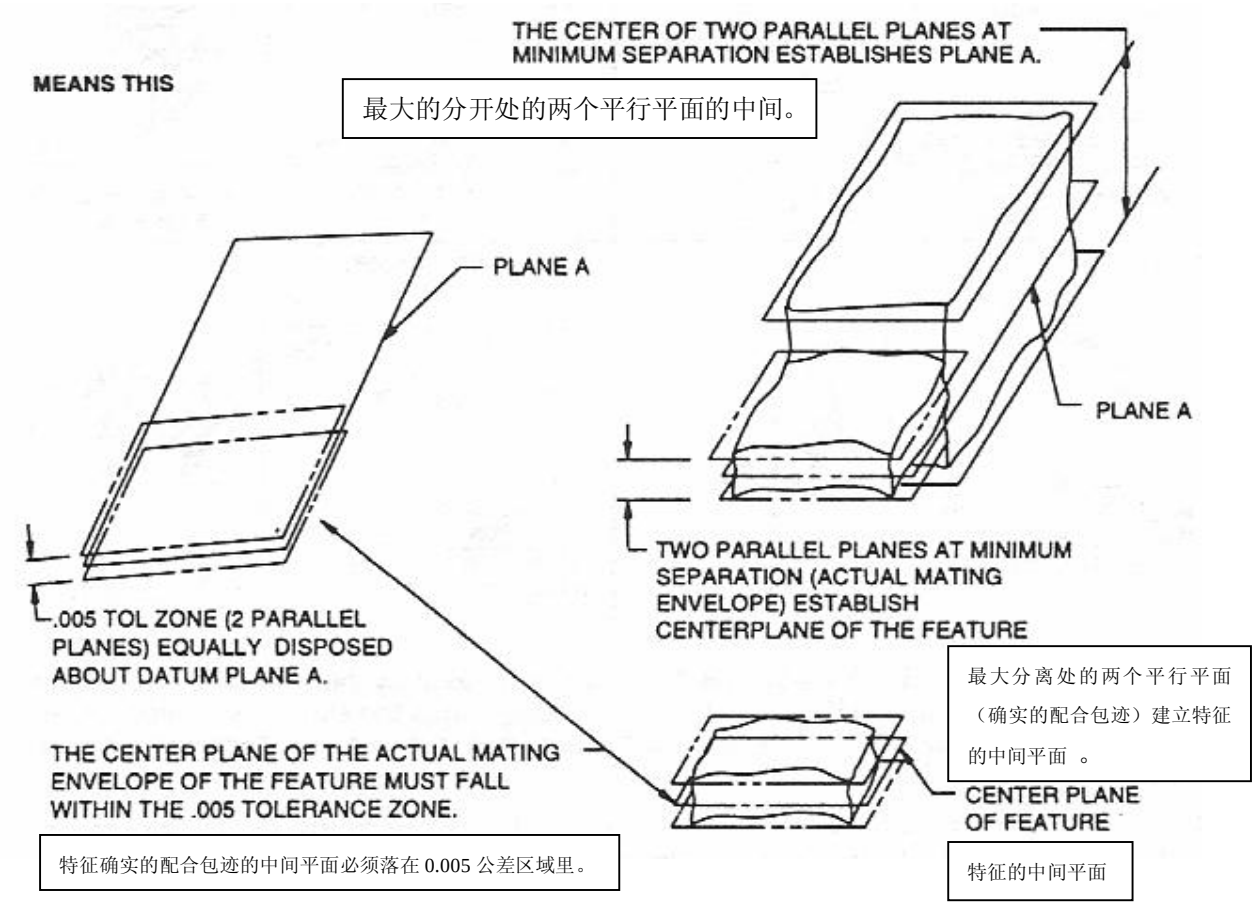
Within the limits of size and regardless of feature size, all median points of opposed elements of the feature must lie within two parallel planes .005 apart. The two parallel planes are equally disposed about datum plane A. the specified tolerance and the datum reference can only apply on an RFS basis.
/在尺寸极限里和忽略特征尺寸，所有的特征的相反的元素的中间点必须在距离为 0.005 的两个平行平面之间。这两个平行平面关于基准面 A 有相等的倾向。指定的公差和基准参照只能应用在 RFS.

POSITION – NON – CYLINDRICAL FEATURE/位置度-非圆柱的特征

The fundamental principles of position tolerancing can be applied to noncylindrical features such as slots and tabs. A position tolerance is shown below locating the center plane of a tab. The tolerance value represents a distance between two parallel planes. This tolerance zone also defines the limits within which variation in attitude or orientation must be confined.
/位置度公差标注的基本原理可能应用到非圆柱特征，例如槽和短小突出部。位置度公差如下面所示，定位短小突出部的中间平面。公差值代表两个平行平面之间的距离。这个公差区域也定义极限，在这个极限里姿势的变异或方向必须被限制。

The tolerance on the feature and datum reference below is applied on an RFS basis. If desired the tolerance could also have been applied on a MMC and/or LMC basis as well. In this case, additional tolerance is available as the feature and datum feature depart from the specified material condition.
/特征上的公差和下面的基准参照以 RFS 为基础应用. 如果想得到的公差也能以 MMC 和/或 LMC 为基础应用. 在这个情况下，因为特征和基准参照偏离指定的材料条件，可得到额外的公差。





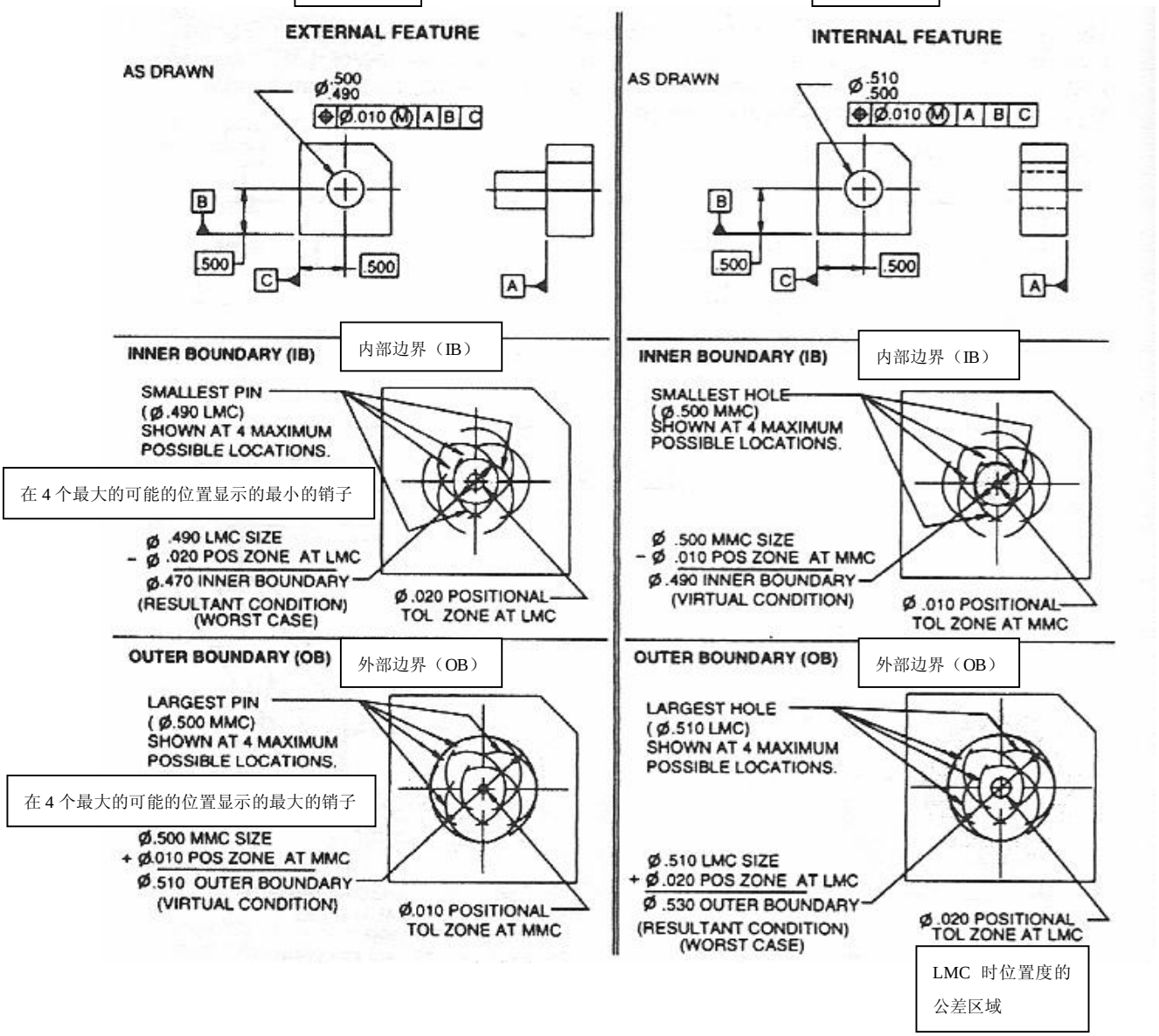
The center plane of the actual mating envelope of the feature must lie between two parallel planes .005 apart which are equally disposed about the center plane of datum feature A.
/特征的真实配合包络轨迹的中间平面必须位于距离为 0.005 的两个平行平面之间，它们倾向于等于基准特征 A 的中间平面。

INNER AND OUTER BOUNDARIES MMC CONCEPT – INTERNAL/EXTERNAL /内部和外部边界 MMC 概念-内部的/外部的

Graphical representations of the inner and outer boundary and the virtual and resultant condition for the MMC concept are shown below. These are important tools to understand and evaluate the ‘worst case ‘boundaries of a feature after geometric tolerancing has been applied. The technical definitions of the terms are shown below.
/MMC 概念的内部，外部边界和虚拟的和合成条件的图形表示如下。应用几何公差标注以后，为理解和评估‘最差的情况‘特征的边界，这些是重要的工具。术语的技术上的定义如下。

外部特征

内部特征



内部边界（IB）

内部边界（IB）

在 4 个最大的可能的位置显示的最小的销子

外部边界（OB）

外部边界（OB）

在 4 个最大的可能的位置显示的最大的销子

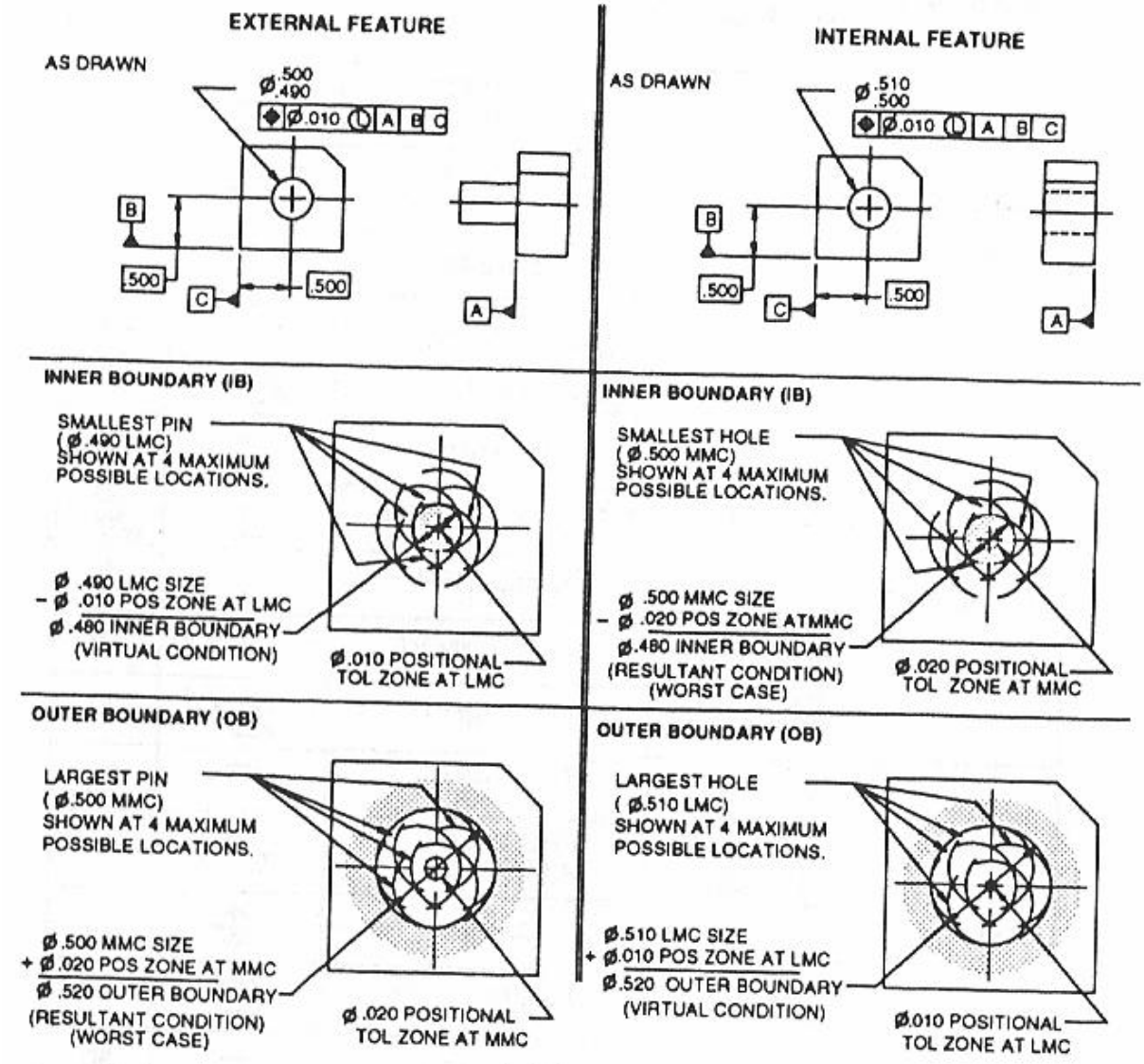
LMC 时位置度的公差区域

INNER BOUNDARY – (IB) A worst case boundary generated by the smallest feature (MMC for an internal feature and LMC for an external feature) minus the stated geometric tolerance and any additional geometric tolerance (if applicable) from the feature's departure from its specified material condition.
/内部边界-（IB）通过最小的特征(对一个内部特征的 MMC 和对于一个外部特征的 LMC)减去所述的几何公差，并且任意额外的从特征偏离它的指定的材料条件的几何公差（如果应用），产生一个最差情况边界。

OUTER BOUNDARY – (OB) A worst case boundary generated by the largest feature (LMC for an internal feature and MMC for an external feature) plus the stated geometric tolerance and any additional geometric tolerance (if applicable) from the feature's departure from its specified material condition.
/外部边界-（OB）通过最大的特征（对于一个内部特征 LMC 和对于一个外部特征 MMC）加上所述的几何公差，并且任意额外的从特征偏离它的指定的材料条件的几何公差（如果应用），产生一个最差情况边界。

INNER AND OUTER BOUNDARIES LMC CONCEPT – INTERNAL/EXTERNAL
/内部和外部边界的 LMC 概念-内部的/外部的

Graphical representations of the inner and outer boundary and the virtual and resultant condition for the LMC concept are shown below. These are important tools to understand and evaluate the 'worst case' boundaries of a feature after geometric tolerancing has been applied. The technical definitions of the terms are shown below.
/内部的，外部的和虚拟的 LMC 的合成条件的边界的图形表示如下。应用几何公差标注以后，为理解和评估特征的‘最差情况‘边界，这些是重要的工具。



The terms, virtual and resultant condition apply only to the MMC and LMC concept.
The IB and OB concept applies to all geometric tolerances.
/术语，虚拟的和合成的条件只应用到 MMC 和 LMC 概念。
IB 和 OB 概念应用到所有几何公差。

VIRTUAL CONDITION – (VC) A constant boundary generated by the collective effect of a size feature's specified MMC or LMC material condition and the geometric tolerance for that material condition.
/虚拟条件-（VC）-通过特征的指定的 MMC 或 LMC 材料条件和材料条件下的几何公差的集体的影响，产生一个不变的边界。

RESULTANT CONDITION (RC) - A variable boundary generated by the effects of a size feature's specified

MMC or LMC material condition, the geometric tolerance for that material condition, the size tolerance, and the additional geometric tolerance derived from the feature's departure from its specified material condition.
/合成条件（RC）-通过特征的指定的 MMC 或 LMC 材料条件，材料条件下的几何公差，尺寸公差和偏离特征指定材料条件的额外的几何公差的影响，产生一个可变的边界。

GEOMETRIC CHARACTERISTIC OVERVIEW/几何特征概述

DATUMS 基准	TYPE OF TOLERANCE /公差类型	CHARACTERISTIC 特征	SYMBOL 符号	2D OR 3D
DATUMS NOT ALLOWED 不允许的基准	FORM 形状	STRAIGHTNESS(LINE ELEMENT) 直线度（线元素）		2D
		STRAIGHNESS (AXIS OR MEDIAN PLANE) 直线度（轴或中间平面）		3D
		FLATNESS 平面度		3D
		CIRCULARITY(ROUNDNESS) 圆度		2D
		CYLINDRICITY 圆柱度		3D
DATUMS REQUIRED 要求基准	ORIENTATION 方向	ANGULARITY 倾斜度		3D(SEE NOTE 3)
		PERPENDICULARITY 垂直度		3D(SEE NOTE 3)
		PARALLELISM 平行度		3D(SEE NOTE 3)
	RUNOUT(SEE NOTE 1) 跳动（参见注解 1）	CIRCULAR RUNOUT 圆跳动		2D
		TOTAL RUNOUT 全跳动		3D
DATUMS REQUIRED (SEE NOTE 2) 要求基准(参见注解 2)	PROFILE (LOCATION OF SURFACES) 轮廓度（曲面的位置）	PROFILE OF A LINE 线轮廓度		2D
		PROFILE OF A SURFACE 面轮廓度		3D
	LOCATION OF FEATURES OF SIZE 尺寸特征的位置	POSITION 位置度		3D
		CONCENTRICTITY 同轴度		3D
		SYMMETRY 对称度		3D

NOTE:

1. CAN CONTROL FORM, ORIENTATION AND LOCATION.
2. THERE ARE SPECIAL CASES WHERE POSITION AND PROFILE MAY NOT REQUIRE DATUMS
3. THESE CHARACTERISTICS CAN BE MADE 2D BY WRITING 'LINE ELEMENTS' UNDER THE

FEATURE CONTROL FRAME.

4. THESE CHARACTERISTICS CONTROL OPPOSING MEDIAN POINTS.

注：

1. 可以控制形状，方向和位置。
2. 有特别的情况，位置度和轮廓度不要求基准。
3. 通过在特征控制框下面写‘线元素‘，这些特征能构成 2D
4. 这些特征控制相反的中间点。

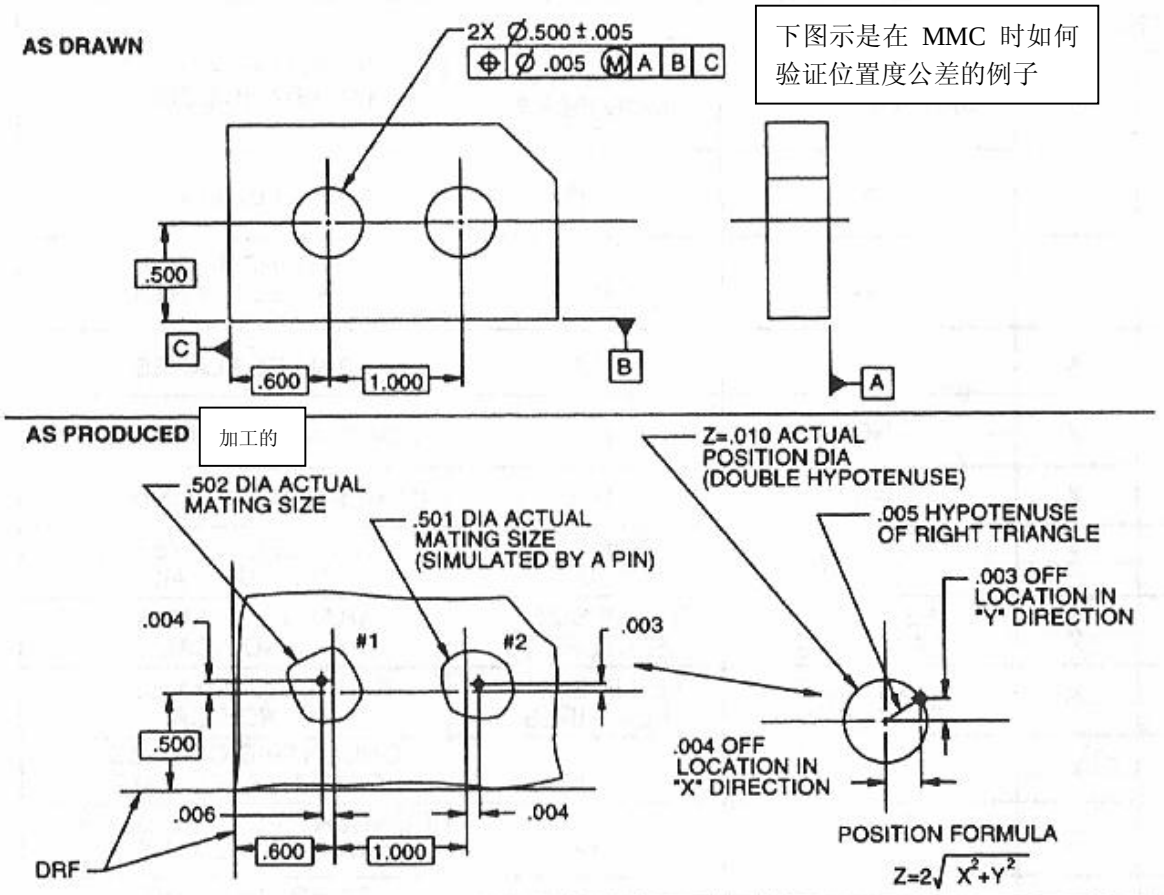
GEOMETRIC CHARACTERISTIC OVERVIEW CONTINUED/几何特征概述续

CONTROLS/控制		APPLICABLITY OF FEATURE MODIFIERS /特征修正符的应用	APPLICABILITY OF DATUM MODIFIERS /基准修正符的应用	COMMON SHAPES OF TOLERANCE ZONE /公差区域的公共形状
AXIS OR MEDIAN PLANE 轴或中间平面	SURFACE /曲面			
	X	NO	N/A	PARALLEL LINES/平行线
X		YES	N/A	CYLINDRICAL PARALLEL PLANES 圆柱平行面
	X	NO	N/A	PARALLEL 平行
	X	NO	N/A	CONCENTRIC CIRCLES /同心圆
	X	NO	N/A	CONCENTRIC CYLINDERS 同心柱面
X	X	YES IF SIZE FEATURES /如果是尺寸特征应用	YES IF SIZE FEATURES	PARALLEL PLANES CYLINDRICAL /平行面，圆柱的
X	X	YES IF SIZE FEATURES	YES IF SIZE FEATURES	PARALLEL PLANES CYLINDRICAL
X	X	YES IF SIZE FEATURES	YES IF SIZE FEATURES	PARALLEL PLANES CYLINDRICAL
	X	NO	NO	CONCENTRIC CIRCLES CIRCULAR ELEMENTS
	X	NO	NO	CONCENTRIC CYLINDERS PARALLEL PLANES
	X	NO	YES IF SIZE FEATURES	2D PROFILE LINE BOUNDARIES
	X	NO	YES IF SIZE FEATURES	3D PROFILE SURFACE BOUNDARIES
X		YES	YES IF SIZE FEATURES	CYLINDRICAL, BOUNDARY PARALLEL PLANES
SEE NOTE 4		NO	NO	CYLINDRICAL
SEE NOTE 4		NO	NO	PARALLEL PLANES

POSITION – HOLE VERIFICATION AT MMC/位置度-MMC 时孔的验证

The example below illustrates how to verify a position tolerance at MMC.

下图示是在 MMC 时如何验证位置度公差的例子



Position tolerance actual is calculated by using "X" and "Y" deviations in the position formula or position chart. In order to pass acceptance, position tolerance actual must be less than position tolerance allowed.

Allowed position tolerance is calculated by taking the difference between the hole MMC and hole actual size and adding it to the position tolerance stated in the feature control frame.

Enter MMC of hole

Enter produced hole size

Enter "X" deviation from basic location

Enter "Y" deviation from basic location

HOLE NO.	HOLE MMC	HOLE ACTUAL SIZE	POSITION TOLERANCE ALLOWED	"X" DIM	"Y" DIM	POSITION TOLERANCE ACTUAL	ACCEPT	REJECT
1	.495	.502	.012	-.006	+.004	.0144		X
2	.495	.501	.011	+.004	+.003	.010	X	

METRIC - CONVERSION CHART

COORDINATE DIMENSIONS TO DIAMETER POSITIONAL TOLERANCE

COORDINATE MEASUREMENT

	0.02	0.04	0.06	0.08	0.10	0.12	0.14	0.16	0.18	0.20	0.22	0.24	0.26	0.28	0.30	0.32	0.34	0.36	0.38	0.40	0.42	0.44	0.46	0.48	0.50
0.50	1.001	1.003	1.007	1.013	1.020	1.028	1.038	1.050	1.063	1.077	1.093	1.109	1.127	1.146	1.166	1.187	1.209	1.232	1.256	1.281	1.306	1.332	1.359	1.386	1.414
0.48	0.961	0.963	0.967	0.973	0.981	0.990	1.000	1.012	1.025	1.040	1.056	1.073	1.092	1.111	1.132	1.154	1.176	1.200	1.224	1.250	1.276	1.302	1.330	1.358	1.386
0.46	0.921	0.923	0.928	0.934	0.941	0.951	0.962	0.974	0.988	1.003	1.020	1.038	1.057	1.077	1.098	1.121	1.144	1.168	1.193	1.219	1.246	1.273	1.301	1.330	1.359
0.44	0.881	0.884	0.888	0.894	0.902	0.912	0.923	0.936	0.951	0.967	0.984	1.002	1.022	1.043	1.065	1.088	1.112	1.137	1.163	1.189	1.217	1.245	1.273	1.302	1.332
0.42	0.841	0.844	0.849	0.855	0.863	0.874	0.885	0.899	0.914	0.930	0.948	0.967	0.988	1.010	1.032	1.056	1.081	1.106	1.133	1.160	1.188	1.217	1.246	1.276	1.306
0.40	0.801	0.804	0.809	0.816	0.825	0.835	0.848	0.862	0.877	0.894	0.913	0.933	0.954	0.977	1.000	1.024	1.050	1.076	1.103	1.131	1.160	1.189	1.219	1.250	1.281
0.38	0.761	0.764	0.769	0.777	0.786	0.797	0.810	0.825	0.841	0.859	0.878	0.899	0.921	0.944	0.968	0.994	1.020	1.047	1.075	1.103	1.133	1.163	1.193	1.224	1.256
0.36	0.721	0.724	0.730	0.738	0.747	0.759	0.773	0.788	0.805	0.824	0.844	0.865	0.888	0.912	0.937	0.963	0.990	1.018	1.047	1.076	1.106	1.137	1.168	1.200	1.232
0.34	0.681	0.685	0.691	0.699	0.709	0.721	0.735	0.752	0.769	0.789	0.810	0.832	0.856	0.881	0.907	0.934	0.962	0.990	1.020	1.050	1.081	1.112	1.144	1.176	1.209
0.32	0.641	0.645	0.651	0.660	0.671	0.684	0.699	0.716	0.734	0.755	0.777	0.800	0.825	0.850	0.877	0.905	0.934	0.963	0.994	1.024	1.056	1.088	1.121	1.154	1.187
0.30	0.601	0.605	0.612	0.621	0.632	0.646	0.662	0.680	0.700	0.721	0.744	0.768	0.794	0.821	0.849	0.877	0.907	0.937	0.968	1.000	1.032	1.065	1.098	1.132	1.166
0.28	0.561	0.566	0.573	0.582	0.595	0.609	0.626	0.645	0.666	0.688	0.712	0.738	0.764	0.792	0.821	0.850	0.881	0.912	0.944	0.977	1.010	1.043	1.077	1.111	1.146
0.26	0.522	0.526	0.534	0.544	0.557	0.573	0.591	0.611	0.632	0.656	0.681	0.708	0.735	0.764	0.794	0.825	0.856	0.888	0.921	0.954	0.988	1.022	1.057	1.092	1.127
0.24	0.482	0.487	0.495	0.506	0.520	0.537	0.556	0.577	0.600	0.625	0.651	0.679	0.708	0.738	0.768	0.800	0.832	0.865	0.899	0.933	0.967	1.002	1.038	1.073	1.109
0.22	0.442	0.447	0.456	0.468	0.483	0.501	0.522	0.544	0.569	0.595	0.622	0.651	0.681	0.712	0.744	0.777	0.810	0.844	0.878	0.913	0.948	0.984	1.020	1.056	1.093
0.20	0.402	0.408	0.418	0.431	0.447	0.466	0.488	0.512	0.538	0.566	0.595	0.625	0.656	0.688	0.721	0.755	0.789	0.824	0.859	0.894	0.930	0.967	1.003	1.040	1.077
0.18	0.362	0.369	0.379	0.394	0.412	0.433	0.456	0.482	0.509	0.538	0.569	0.600	0.632	0.666	0.700	0.734	0.769	0.805	0.841	0.877	0.914	0.951	0.988	1.025	1.063
0.16	0.322	0.330	0.342	0.358	0.377	0.400	0.425	0.453	0.482	0.512	0.544	0.577	0.611	0.645	0.680	0.716	0.752	0.788	0.825	0.862	0.899	0.936	0.974	1.012	1.050
0.14	0.283	0.291	0.305	0.322	0.344	0.369	0.396	0.425	0.456	0.488	0.522	0.556	0.591	0.626	0.662	0.699	0.735	0.773	0.810	0.848	0.885	0.923	0.962	1.000	1.038
0.12	0.243	0.253	0.268	0.288	0.312	0.339	0.369	0.400	0.433	0.466	0.501	0.537	0.573	0.609	0.646	0.684	0.721	0.759	0.797	0.835	0.874	0.912	0.951	0.990	1.028
0.10	0.204	0.215	0.233	0.256	0.283	0.312	0.344	0.377	0.412	0.447	0.483	0.520	0.557	0.595	0.632	0.671	0.709	0.747	0.786	0.825	0.863	0.902	0.941	0.981	1.020
0.08	0.165	0.179	0.200	0.226	0.256	0.288	0.322	0.358	0.394	0.431	0.468	0.506	0.544	0.582	0.621	0.660	0.699	0.738	0.777	0.816	0.855	0.894	0.934	0.973	1.013
0.06	0.126	0.144	0.170	0.200	0.233	0.268	0.305	0.342	0.379	0.418	0.456	0.495	0.534	0.573	0.612	0.651	0.691	0.730	0.769	0.809	0.849	0.888	0.928	0.967	1.007
0.04	0.089	0.113	0.144	0.179	0.215	0.253	0.291	0.330	0.369	0.408	0.447	0.487	0.526	0.566	0.605	0.645	0.685	0.724	0.764	0.804	0.844	0.884	0.923	0.963	1.003
0.02	0.057	0.089	0.126	0.165	0.204	0.243	0.283	0.322	0.362	0.402	0.442	0.482	0.522	0.561	0.601	0.641	0.681	0.721	0.761	0.801	0.841	0.881	0.921	0.961	1.001

COORDINATE MEASUREMENT

$$\text{POSITIONAL DIAMETER} = 2\sqrt{x^2 + y^2}$$