دانشگاه صنعتی شاهرود
دانشکده مهندسی مکانیک

Geometric Dimensioning and Tolerancing

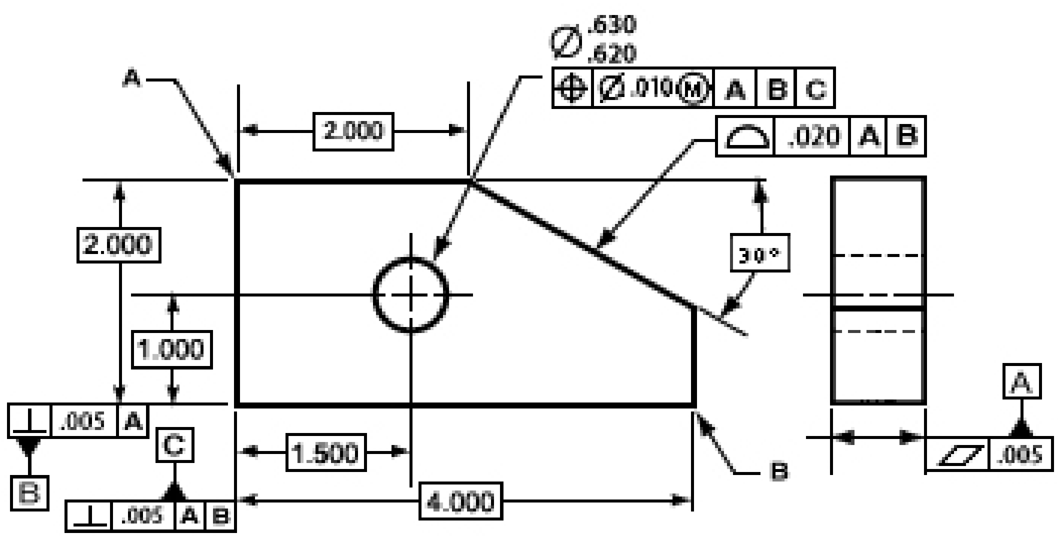
(GD&T)

Content

- The Feature Control Frame
- Types of Tolerances
- GENERAL RULES OF GD&T (con't)
- Limit (+/-) Tolerancing vs. Geometric Tolerancing
- Examples

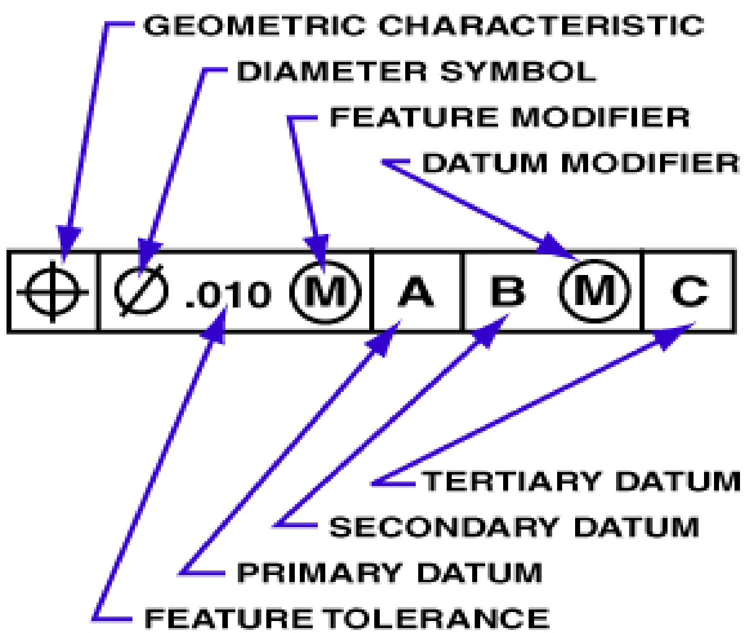
The Feature Control Frame

- GD&T instructions contain a large amount of information.
- Each feature is given a feature control frame.
- Frame reads from left to right, like a basic sentence.
- Instructions are organized into a series of symbols that fit into standardized compartments.



The Feature Control Frame (con't)

- The first compartment defines the geometric characteristic of the feature, using one of the 14 standard geometric tolerance symbols (⌖ means "position"). A second feature control frame is used if a second geometric tolerance is needed.

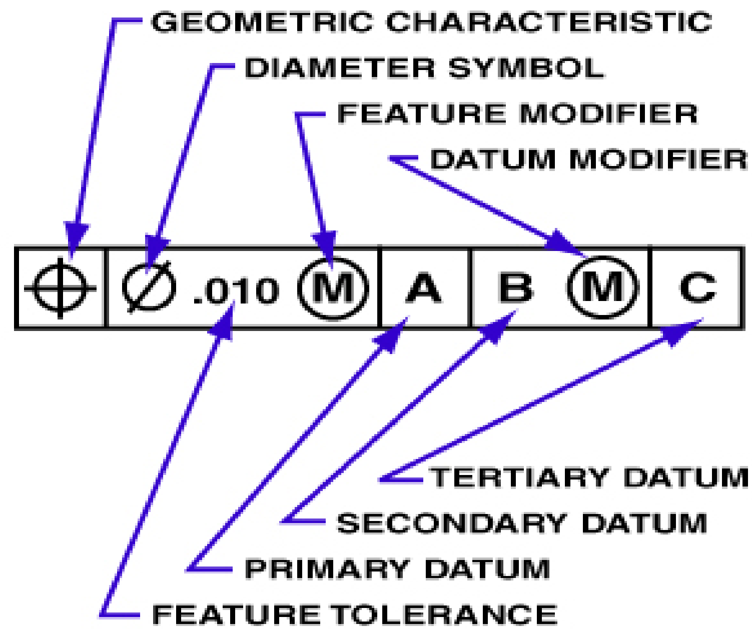


- The second compartment contains the entire tolerance for the feature, with an *additional diameter symbol* to indicate a cylindrical or circular tolerance zone. No additional symbol is needed for parallel lines or planes. If needed, material condition modifiers would also appear in the second compartment.

The Feature Control Frame (con't)

- The third compartment indicates the primary datum which locates the part within the datum reference frame. Every related tolerance requires a primary datum but independent tolerances, such as form tolerances, do not.

- The fourth and fifth compartments contain the secondary and tertiary datums. Depending on the geometric tolerance and the function of the part, secondary and tertiary datums may not be necessary.



Types of Tolerances – 5 major groups.

- ***Form Tolerances*** (flatness, circularity, cylindricity & straightness.
- ***Profile Tolerances*** (profile of surface, profile of line).

  Powerful tolerances that control several aspects.
- ***Orientation Tolerances*** (perpendicularity, parallelism, and angularity).
- ***Location Tolerances*** (concentricity, symmetry, and position).
- ***Runout Tolerances*** (circular and total). Used only on cylindrical parts.

Straight & Cylindrical Tolerances (con't)

An individual tolerance is not related to a datum. A related tolerance must be compared to one or more datums.

TYPE OF TOLERANCE	CHARACTERISTIC	SYMBOL	DATUM REFERENCE
FORM	STRAIGHTNESS	⏤	INDIVIDUAL
	FLATNESS	▱	
	CIRCULARITY (ROUNDNESS)	○	
	CYLINDRICITY	⌭	
PROFILE	PROFILE OF A LINE	⌒	INDIVIDUAL OR RELATED
	PROFILE OF A SURFACE	⌓	
ORIENTATION	ANGULARITY	∠	RELATED
	PERPENDICULARITY	⊥	
	PARALLELISM	∥	
LOCATION	POSITION	⌖	
	CONCENTRICITY	◎	
	SYMMETRY	⌯	
RUNOUT	CIRCULAR RUNOUT	↗	
	TOTAL RUNOUT	⌰	

Straightness and Flatness

- Two types of form tolerances.

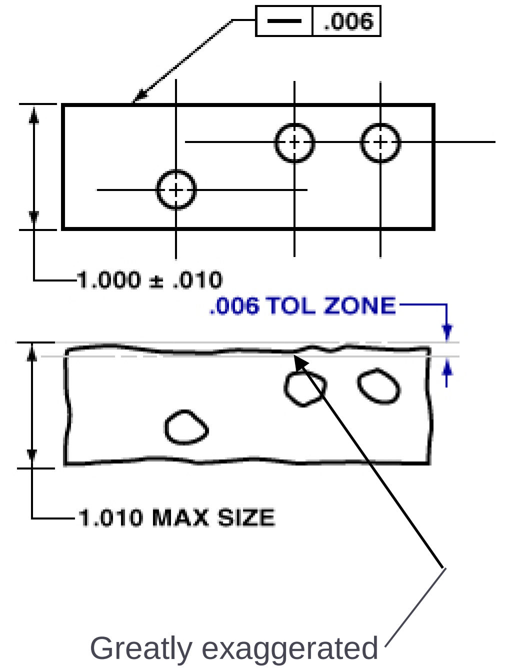
Both define a feature independently.

- Straightness is a two-dimensional tolerance.

Edge must remain within two imaginary parallel lines to meet straightness tolerance. Distance between lines is determined by size of specified tolerance.

- Most rectangular parts have a straightness tolerance.

- Edge or center axis of a cylinder may have a straightness tolerance.



Straightness and Flatness (con’t)

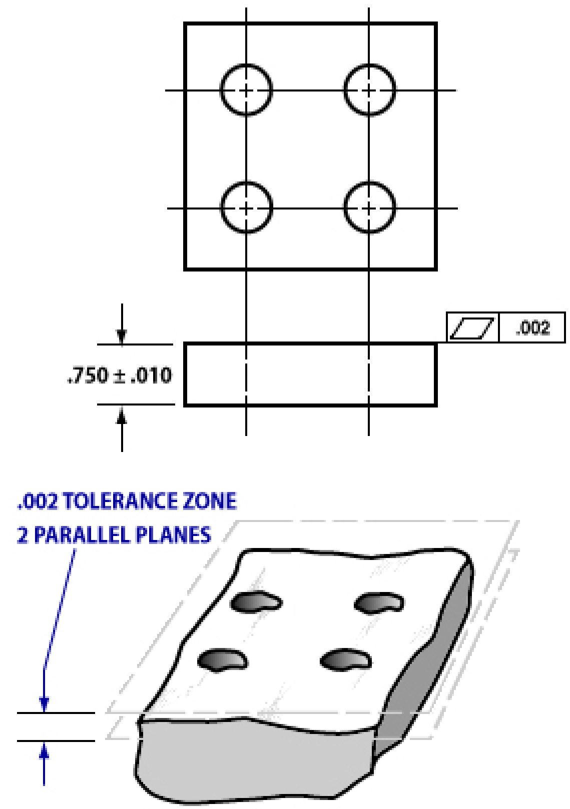
- Flatness is a three-dimensional version of straightness tolerance.

  - Requires a surface to be within two imaginary, perfectly flat, perfectly parallel planes.

  - Only the surface of the part, not the entire thickness, is referenced to the planes.

  - Most often used on rectangular or square parts.

  - If used as a primary datum, flatness must be specified in the drawing.



Circularity and Cylindricity

- Circularity (often called roundness).

  - Two-dimensional tolerance.

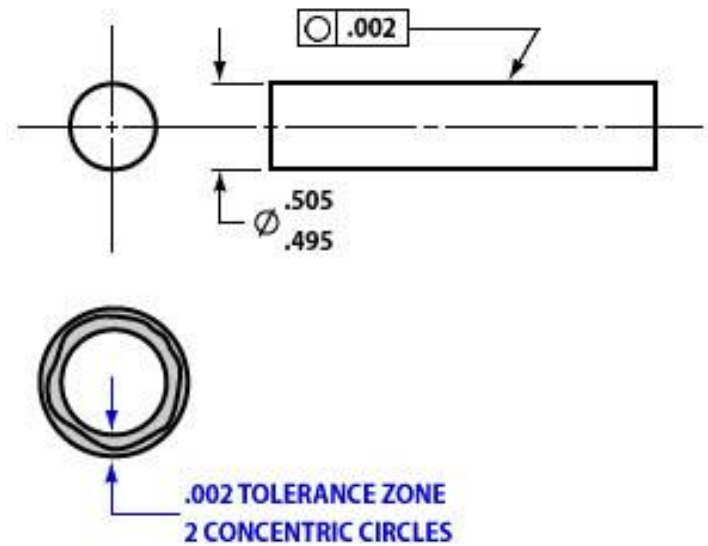
  - Most often used on cylinders.

  - Also applies to cones and spheres.

  - Demands that any two-dimensional cross-section of a round feature must stay within the tolerance zone created by two concentric circles.

  - Most inspectors check multiple cross-sections.

  - Each section must meet the tolerance on its own.



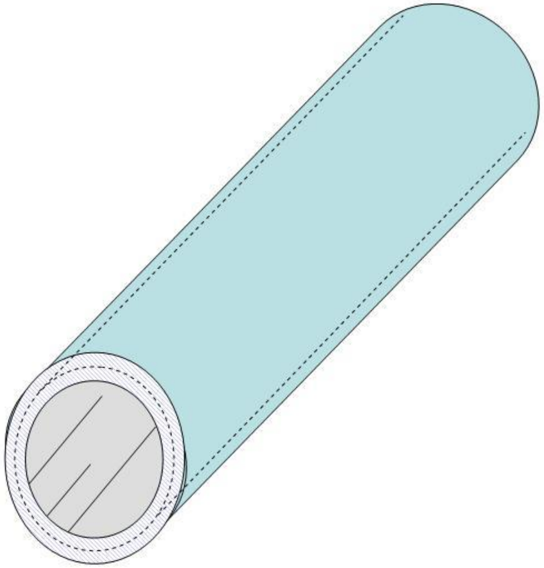
Circularity and Cylindricity (con't)

- Cylindricity specifies the roundness of a cylinder along its entire length.

  - All cross-sections of the cylinder must be measured together, so cylindricity tolerance is only applied to cylinders.

- Circularity and cylindricity cannot be checked by measuring various diameters with a micrometer.

- Part must be rotated in a high-precision spindle. Best method would be to use a Coordinate Measuring Machine (CMM).



The thickness of the wall of a pipe represents the cylindricity tolerance zone.

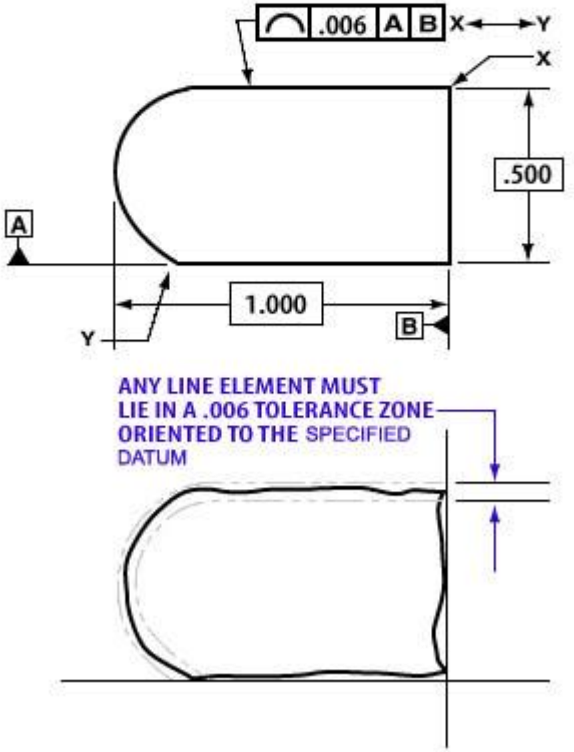
دانشگاه صنعتی شاهرود

Profile of a Line and Surface

- The two versions of profile tolerance.
- Both can be used to control features such as cones, curves, flat or irregular surfaces, or cylinders.
- A profile is an outline of the part feature in one of the datum planes.
- They control orientation, location, size and form.

- The profile of a line is a two-dimensional tolerance.

  - It requires the profile of a feature to fall within two imaginary parallel lines that follow the profile of the feature.

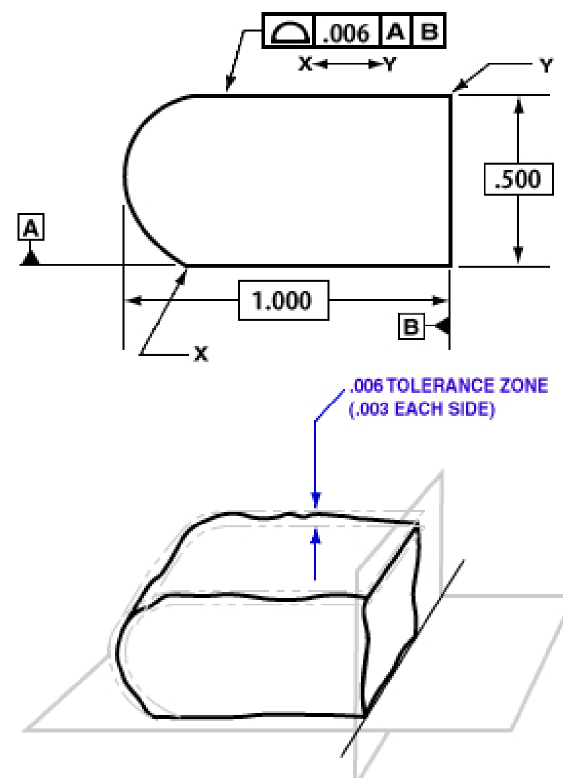


Profile of a Line and Surface (con't)

- Profile of a Surface is three-dimensional version of the line profile.

  - Often applied to complex and curved contour surfaces such as aircraft and automobile exterior parts.

  - The tolerance specifies that the surface must remain within two three-dimensional shapes.

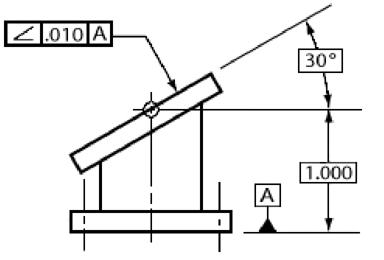


Orientation and Location Tolerances

- **Angularity, Perpendicularity, and Parallelism**

  - These tolerances define the angle and orientation of features as they relate to other features.

  - They specify how one or more datums relate to the primary toleranced feature. (Relational Tolerances)

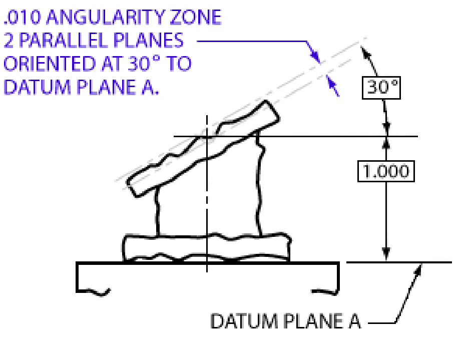


Angularity - A three-dimensional tolerance.

*** Shape of the tolerance zone depends on shape of the feature.**

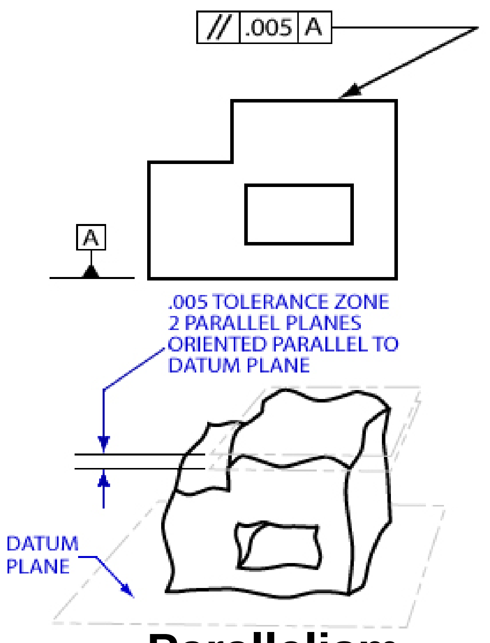
*** If applied to flat surface, tolerance zone becomes two imaginary planes, parallel to ideal angle.**

*** If applied to a hole, it is referenced to an imaginary cylinder existing around the ideal angle and center of the hole must stay within that cylinder.**

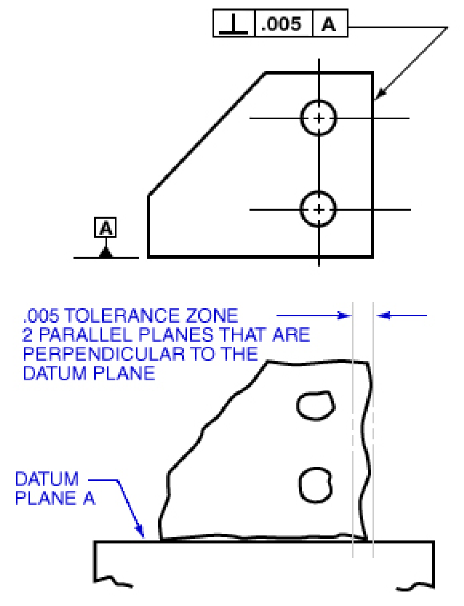


Orientation and Location Tolerances (con't)

- **Perpendicularity and Parallelism : Three-dimensional tolerances that use the same tolerance zones as angularity.**
- **Difference is that parallelism defines two features that must remain parallel to each other, while perpendicularity specifies a 90-degree angle between features.**



Parallelism



Perpendicularity

دانشگاه صنعتی شاهرود

Orientation and Location Tolerances (con't)

- Parallelism and Flatness are often confused.

  - Flatness is not related to another datum plane.

- When an orientation tolerance is applied to a flat surface, it indirectly defines the flatness of the feature.

Orientation and Location Tolerances (con't)

- **Position** is one of most common location tolerances.

- A three-dimensional, related tolerance.

- Ideal, exact location of feature is called
true position.

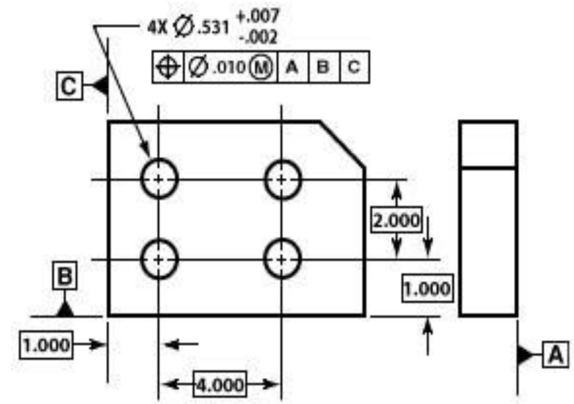
- Actual location of a feature is compared to the ideal true position.

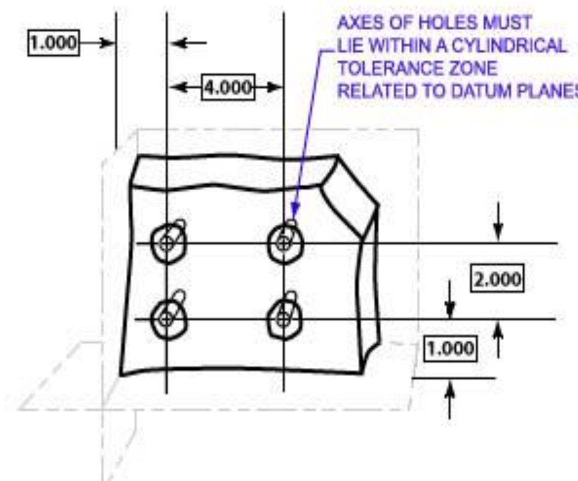
- Usually involves more than one datum to determine where true position should be.

- Has nothing to do with size, shape, or angle, but rather "where it is".

Orientation and Location Tolerances (con't)

- In the case of holes, the tolerance involves the center axis of the hole and must be within the imaginary cylinder around the intended true position of the hole.
- If toleranced feature is rectangular, the zone involves two imaginary planes at a specified distance from the ideal true position.
- Position tolerance is easy to inspect and is often done with just a functional gage (go / no-go gage).





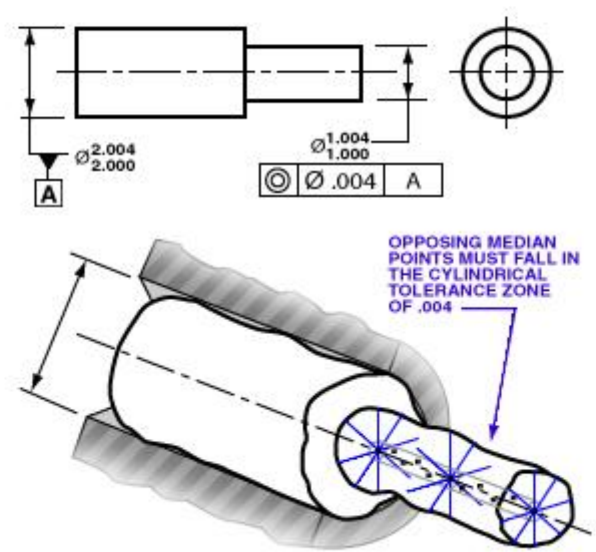
Orientation and Location Tolerances (con't)

- **Concentricity** and **Symmetry** are both three-dimensional tolerances.

- **Concentricity** is not commonly measured.

  - It relates a feature to one or more other datum features.

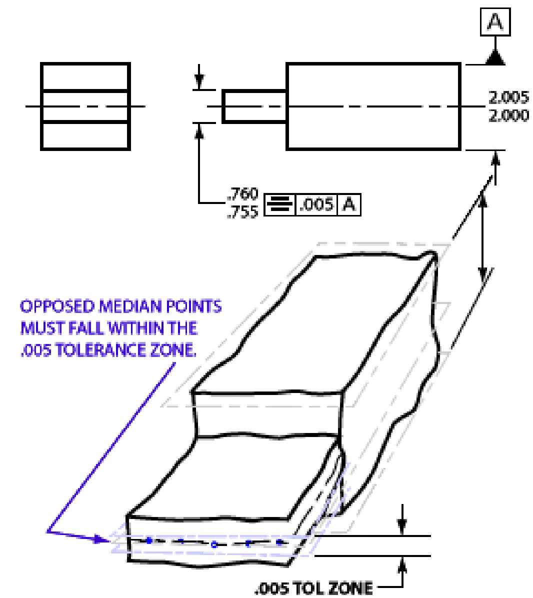
  - This shaft is measured in multiple diameters to ensure that they share a common center-axis.



Orientation and Location Tolerances (con't)

- Symmetry is much like concentricity.

* Difference is that it controls rectangular features and involves two imaginary flat planes, much like parallelism.

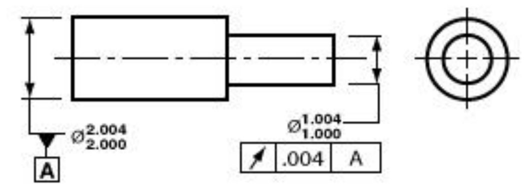


* Both symmetry and concentricity are difficult to measure and increase costs of inspection.

* When a certain characteristic, such as balance, is important, these tolerances are very effective.

Orientation and Location Tolerances (con’t)

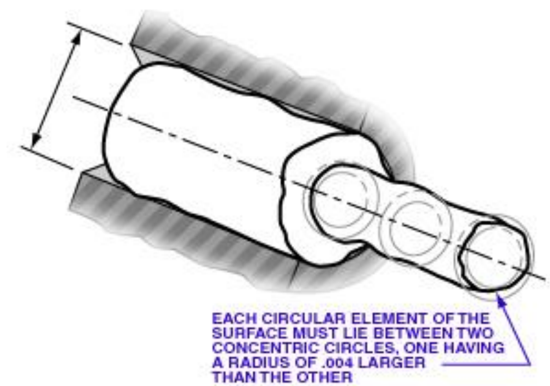
- **Circular and Total Runout are three-dimensional and apply only to cylindrical parts.**
- **Both tolerances reference a cylindrical feature to a center datum-axis, and simultaneously control the location, form and orientation of the feature.**



- **Circular runout can only be inspected when a part is rotated.**

  - Calibrated instrument is placed against the surface of the rotating part to detect the highest and lowest points.

  - The surface must remain within two imaginary circles, having their centers located on the center axis.

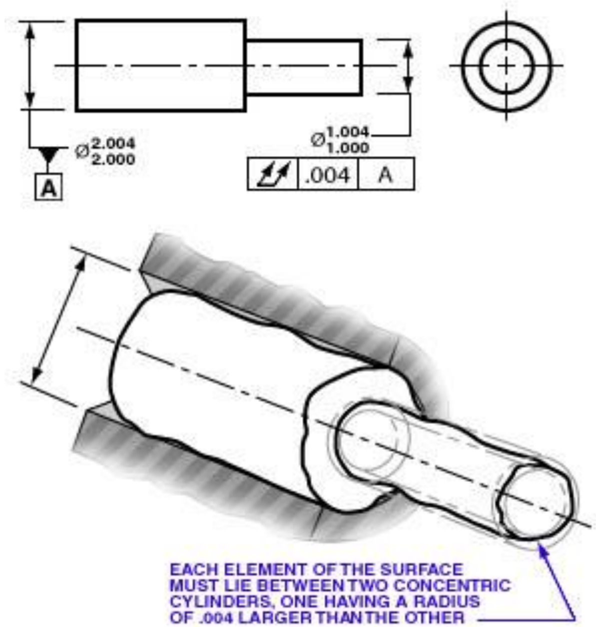


Orientation and Location Tolerances (con't)

- **Total Runout is similar to circular runout except that it involves tolerance control along the entire length of, and between, two imaginary cylinders, not just at cross sections.**

  - By default, parts that meet total runout tolerance automatically satisfy all of the circular runout tolerances.

  - Runout tolerances, especially total runout, are very demanding and present costly barriers to manufacturing and inspection.

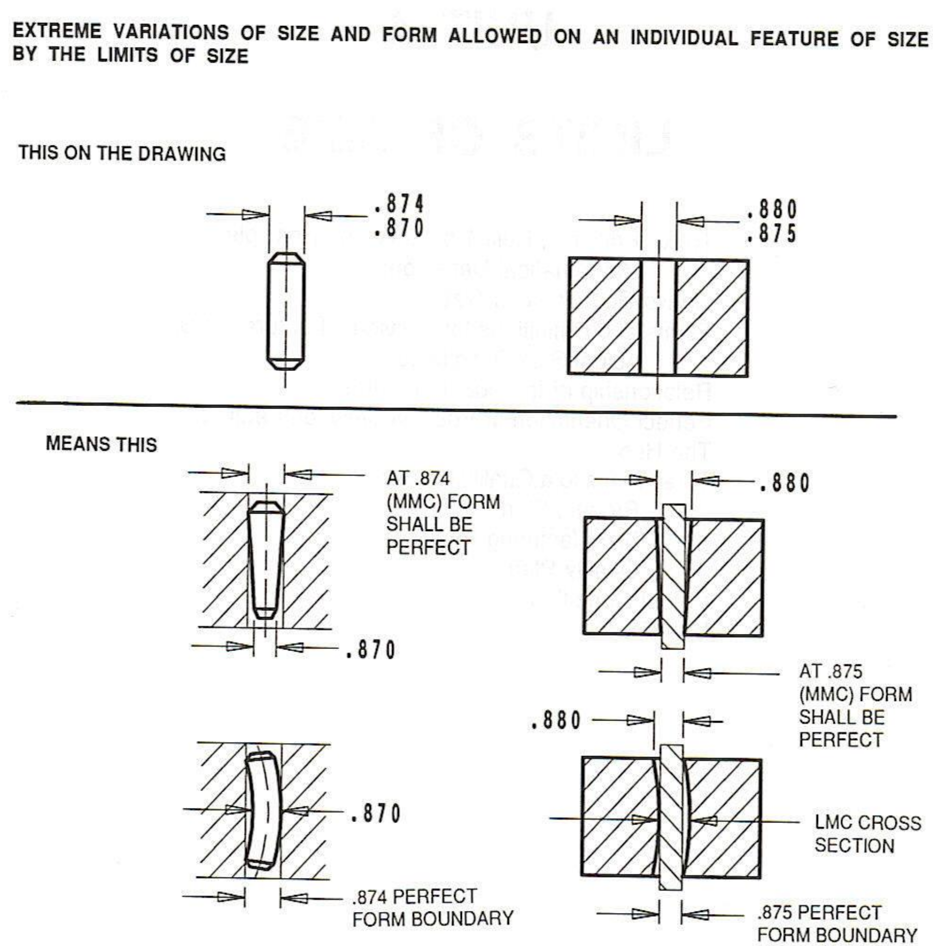


دانشگاه صنعتی شاهرود

GENERAL RULES OF GD&T

- **Geometric dimensioning and tolerancing is based on certain fundamental rules. Some of these follow from standard interpretation of the various characteristics, some govern specification, and some are General Rules applying across the entire system.**
- **Rule #1 is the Taylor Principle, attributed to William Taylor who in 1905 obtained a patent on the full form "go-gage". It is referred to as Rule #1 or "Limits of Size" in the Y14.5M, 1994 standard. The Taylor Principle is a very important concept that defines the size and form limits for an individual feature of size. In the international community the Taylor Principle is often called the "envelope principle".**

GENERAL RULES OF GD&T (con't)



- Variations in size are possible while still keeping within the perfect boundaries. The limits of size define the "size" (outside measurements) as well as the "form" (shape) of a feature. The feature may vary within the limits. That is, it may be bent, tapered, or out of round, but if it is produced at its maximum material condition, the form must be perfect. (or, as close as possible)

دانشگاه صنعتی شاهرود

GENERAL RULES OF GD&T (con't)

- **Individual Feature of Size:**
  - When 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 size are allowed.
- **Variation of Size:**
  - The actual size of an individual feature at any cross section shall be within the specified tolerance size.

دانشگاه صنعتی شاهرود

GENERAL RULES OF GD&T (con't)

- **Variation of Form:**

The form of an individual feature is controlled by its limits of size to the extent prescribed in the following paragraph and illustration.

- **The surface or surfaces of a feature shall not extend beyond a boundary (envelope) of perfect form at Maximum Material Condition (MMC). This boundary is the true geometric form represented by the drawing. No variation is permitted if the feature is produced at its MMC limit of size. (Plain English- *If the part is produced at Maximum Material Condition, it shall not be bigger than the perfect form of the drawing.*)**

- **Where the actual size of a feature has departed from MMC toward LMC, a variation in form is allowed equal to the amount of such departure.**

- **There is no requirement for a boundary of perfect form at LMC. Thus, a feature produced at LMC limit of size is permitted to vary from true form to the maximum variation allowed by the boundary of perfect form at MMC.**

دانشگاه صنعتی شاهرود

GENERAL RULES OF GD&T (con't)

- **Rule #2 – Applicability of MMC, LMC, & RFS :**
- In the current ASME Y14.5M-1994, Rule # 2 governs the applicability of modifiers in the Feature Control Frame. The rule states that "Where no modifying symbol is specified with respect to the individual tolerance, datum reference, or both, then RFS (Regardless of Feature Size) automatically applies and is assumed. Since RFS is implied, it is not necessary to include the symbol. Therefore, the symbol **S** has been eliminated from the current standard.
- MMC and LMC must be specified where required.
- **Rule #3 – Eliminated:**
- **Rule #4 & #5 - Eliminated:**

GENERAL RULES OF GD&T (con't)

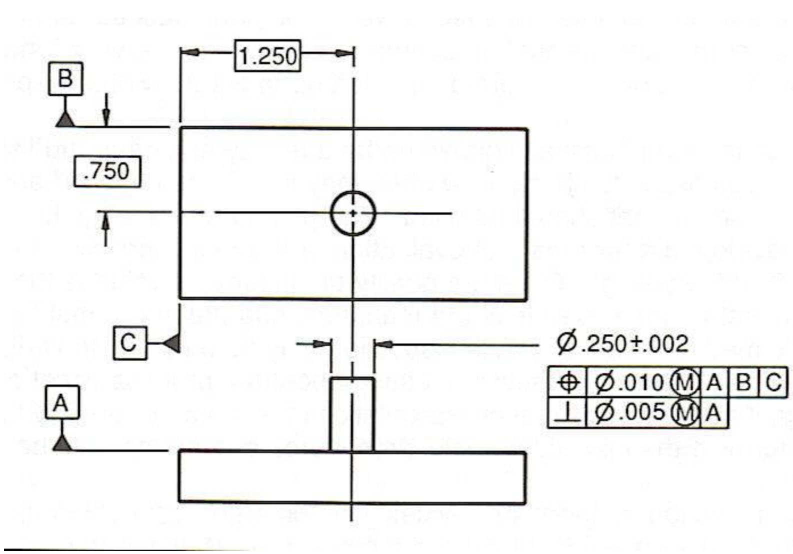
- ## What is Virtual Condition ?

Depending upon its intended purpose, a feature may be controlled by tolerances such as form, size, orientation and location. The collective (total) effects of these factors determine the clearances between mating parts and they establish gage feature sizes. The collective effect of these factors is called "virtual condition".

Virtual condition is a constant boundary created by the total effects of a "size" feature based on its MMC or LMC condition and the geometric tolerance for that material condition.

GENERAL RULES OF GD&T (con't)

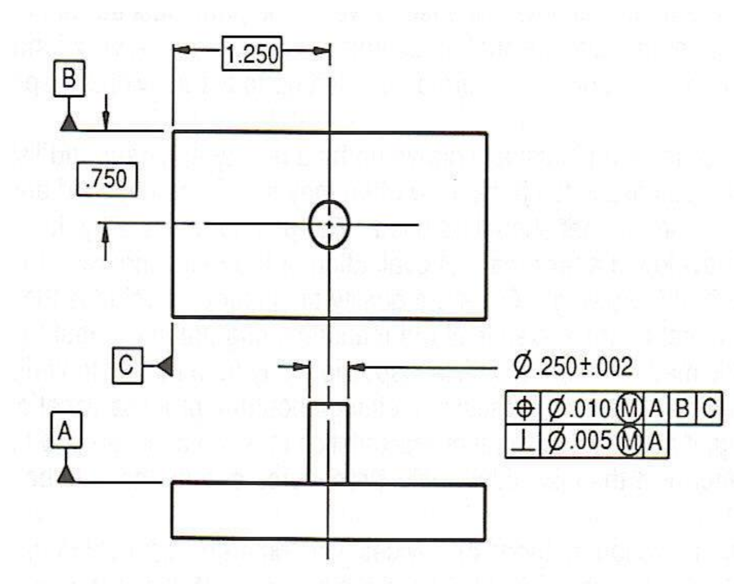
- The size tolerance for the pin (.250 ± .002) and the location and perpendicularity tolerances listed in the Feature Control Frame combine to create two possible virtual sizes. First, regardless of its position or angle, the pin must still lie within the .002 boundary specified for its width. However, the tolerance for perpendicularity allows a margin of .005. So, if the part were produced at MMC to .252 and it deviates from perpendicularity by the .005 allowed, the total virtual size of the pin can be considered to be **.257** in relation to datum A.



GENERAL RULES OF GD&T (con't)

- Second, the position tolerance of .010 combined with the size tolerance of .002 would produce a virtual size of **.262** in relation to datums A, B and C.

  This means that an inspection gage would have to have a hole of .262 to allow for the combined tolerances, even though the pin can be no more than .252 diameter. Therefore, three inspections would be necessary in order to check for size, perpendicularity, and location.



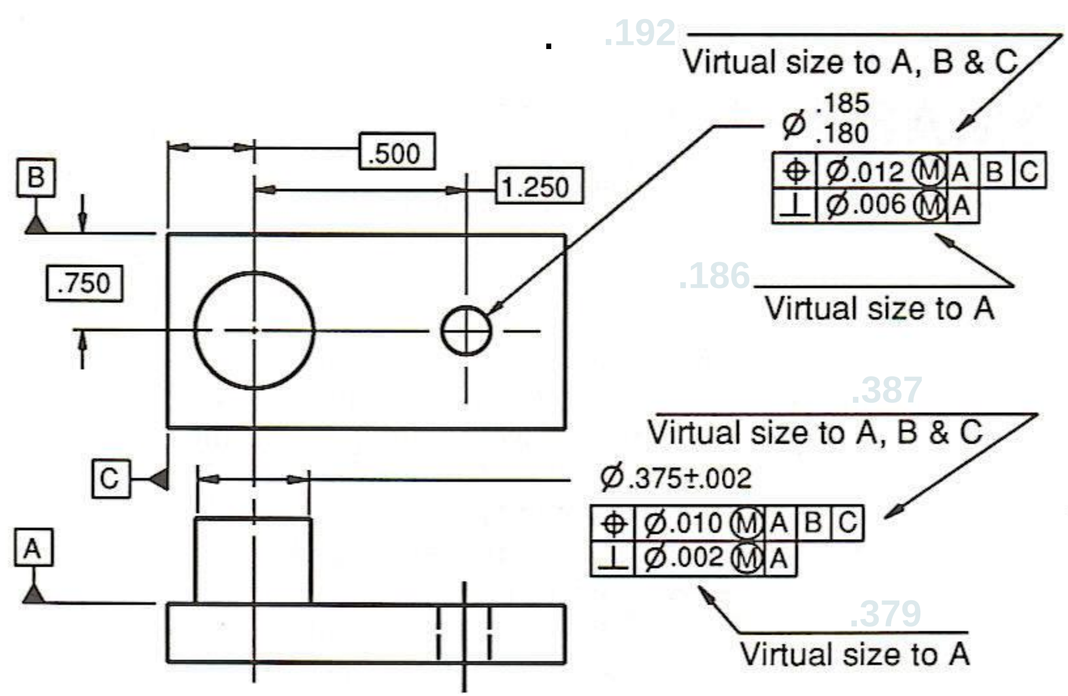
دانشگاه صنعتی شاهرود

GENERAL RULES OF GD&T (con't)

- Virtual size of a hole
  - When calculating the virtual size of a hole, you must remember the rule concerning Maximum Material Condition (MMC) and Least Material Condition (LMC) of holes. Recall that when machining a hole, MMC means the "most material that can remain in the hole". Therefore, a hole machined at MMC will be smaller and a hole machined at LMC will be larger. It is important to read the Feature Control Frame information carefully to make sure you understand which feature is specified and what material conditions are required.

GENERAL RULES OF GD&T (con't)

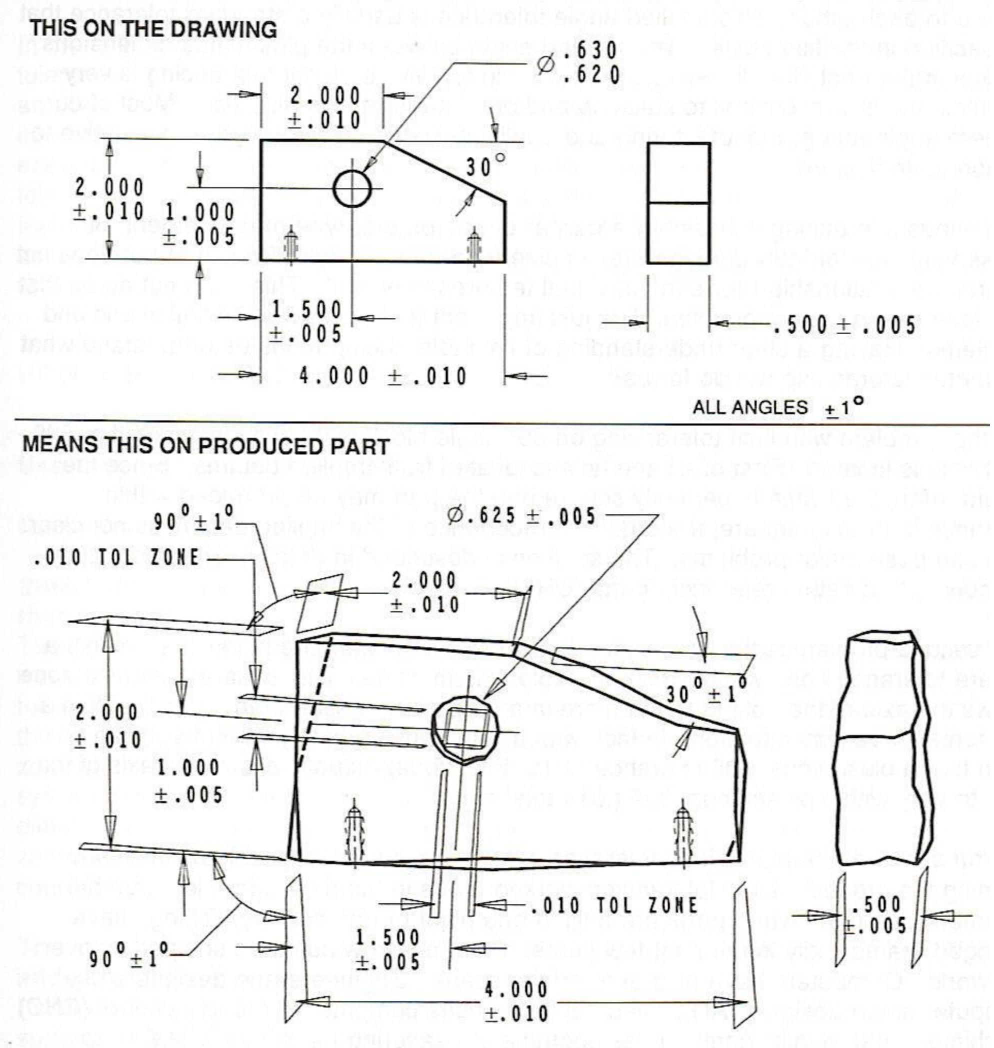
Calculate the virtual sizes for the indicated features.



Limit (+/-) Tolerancing vs. Geometric Tolerancing

دانشگاه صنعتی شاهرود

- **Limit Tolerancing (+/-) is restricted when inspecting all features of a part and their relationships.**
  - **(+/-) is basically a two-dimensional tolerancing system (a caliper/ micrometer type measurement.**
  - **Works well for individual features.**
  - **Does not control the relationship between individual features.**

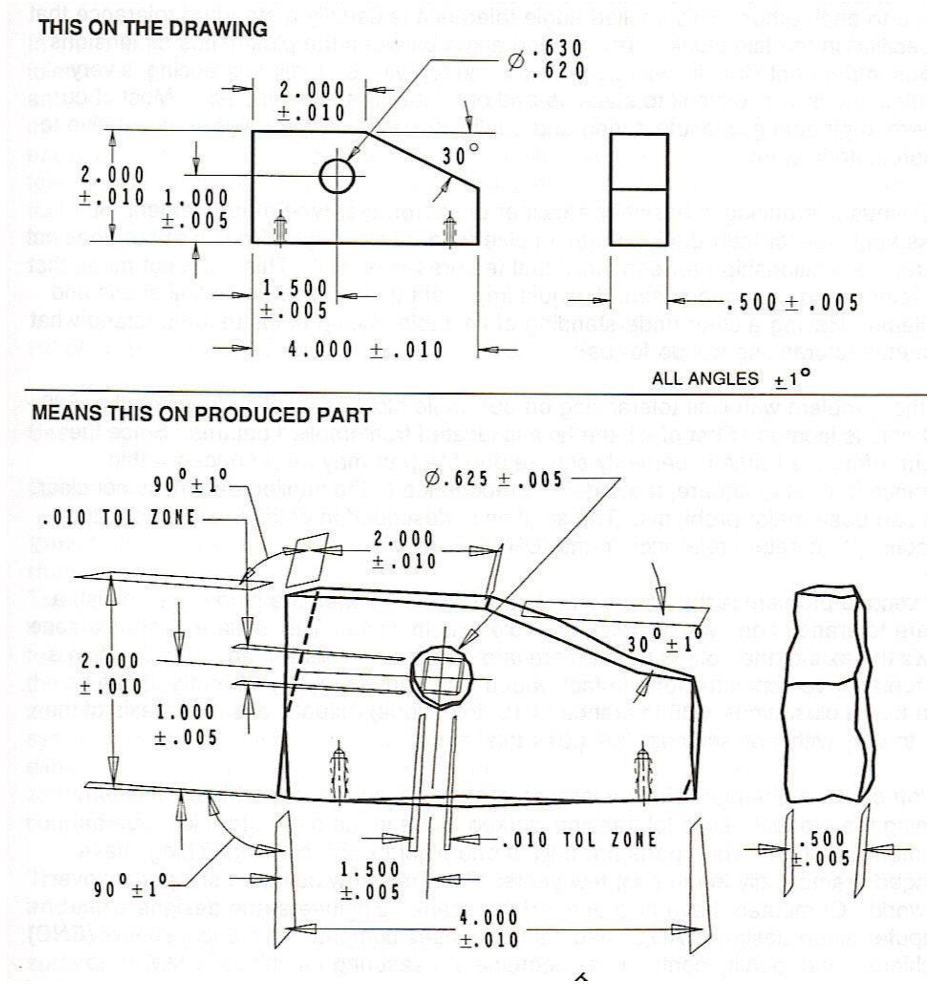


Limit (+/-) Tolerancing vs. Geometric Tolerancing

دانشگاه صنعتی شاهرود

- **Visually, the block will look straight and square. The variations will be so small that they are undetectable with the human eye. However, when the parts are inspected using precision measuring equipment such as a CMM, the angle block starts to look like the bottom drawing (greatly exaggerated).**

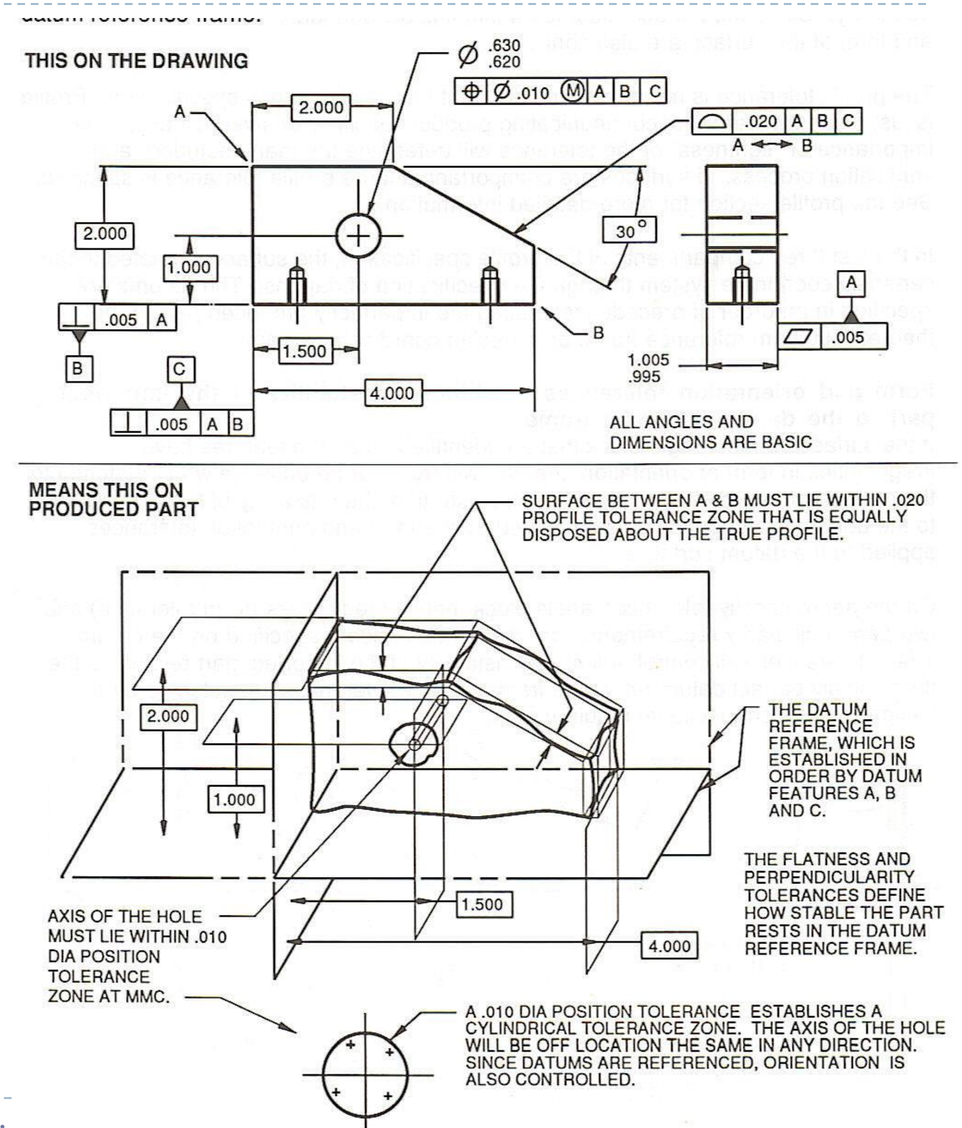
- **The block is not square in either view. The surfaces are warped and not flat. The hole is not square to any surface and it is not round. It is at this point that the limit system of tolerance breaks down. Plus/minus tolerances are two dimensional; the actual parts are three dimensional. Limit tolerances usually do not have an origin or any location or orientation relative to datums. The datums are usually implied. Most of our modern engineering, manufacturing and quality systems all work square or relative to a coordinate system. Parts must be described in a three dimensional mathematical language to ensure clear and concise communication of information relating to product definition. That is why we need geometric tolerancing.**



دانشگاه صنعتی شاهرود

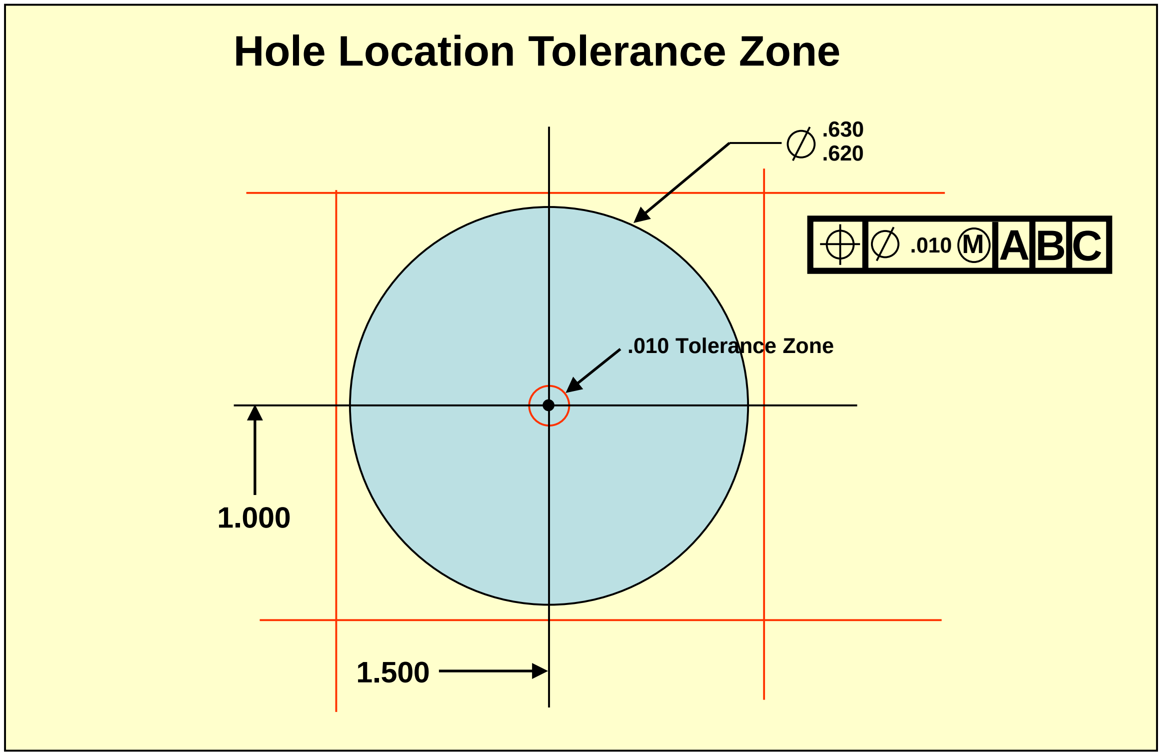
Limit (+/-) Tolerancing vs. Geometric Tolerancing

- The same angle block is now done with geometrics.
  - Notice that datums A, B and C have been applied to features on the part establishing a X,Y and Z Cartesian coordinate system.
  - Geometrics provides a very clear, concise three dimensional mathematical language for product definition.



Limit (+/-) Tolerancing vs. Geometric Tolerancing

- **A close-up look at the angle block shows how the features are controlled. For example, the hole location is controlled by the feature control frame shown below.**



The MMC condition dictates a smaller position tolerance. If the hole is made to the Least Material Condition (LMC), resulting in a larger hole, then the hole location can be farther off and still align with the mating pin.

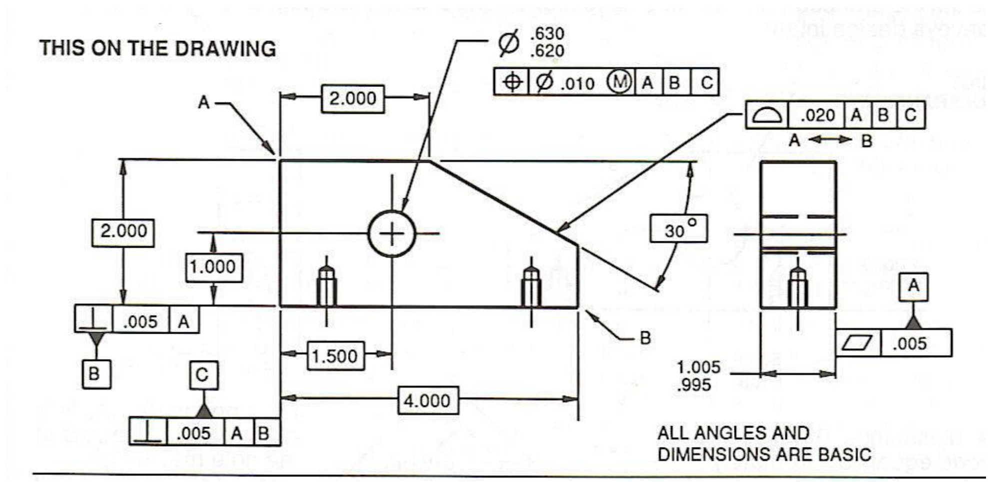
.010 when hole size is .620 (MMC)

.020 when hole size is .630 (LMC)

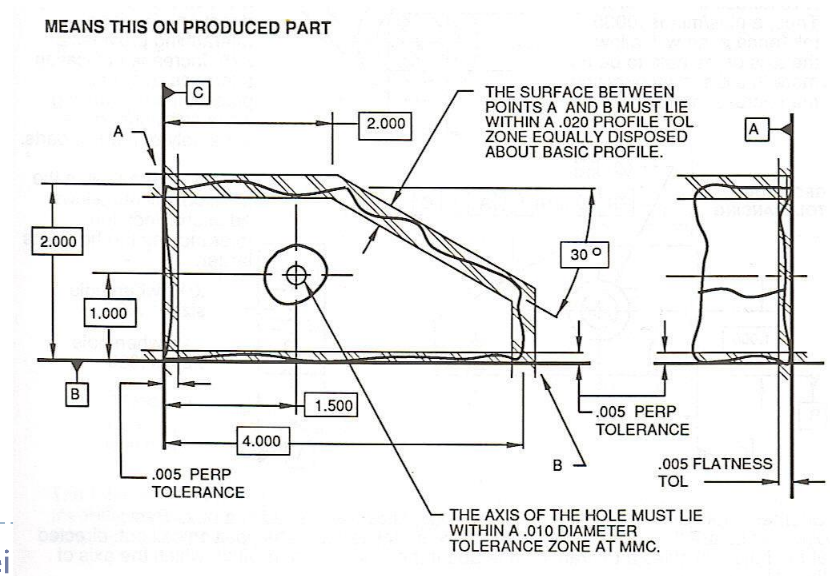
دانشگاه صنعتی شاهرود

Limit (+/-) Tolerancing vs. Geometric Tolerancing

- **Geometric Tolerancing Applied to an Angle Block – 2D View**

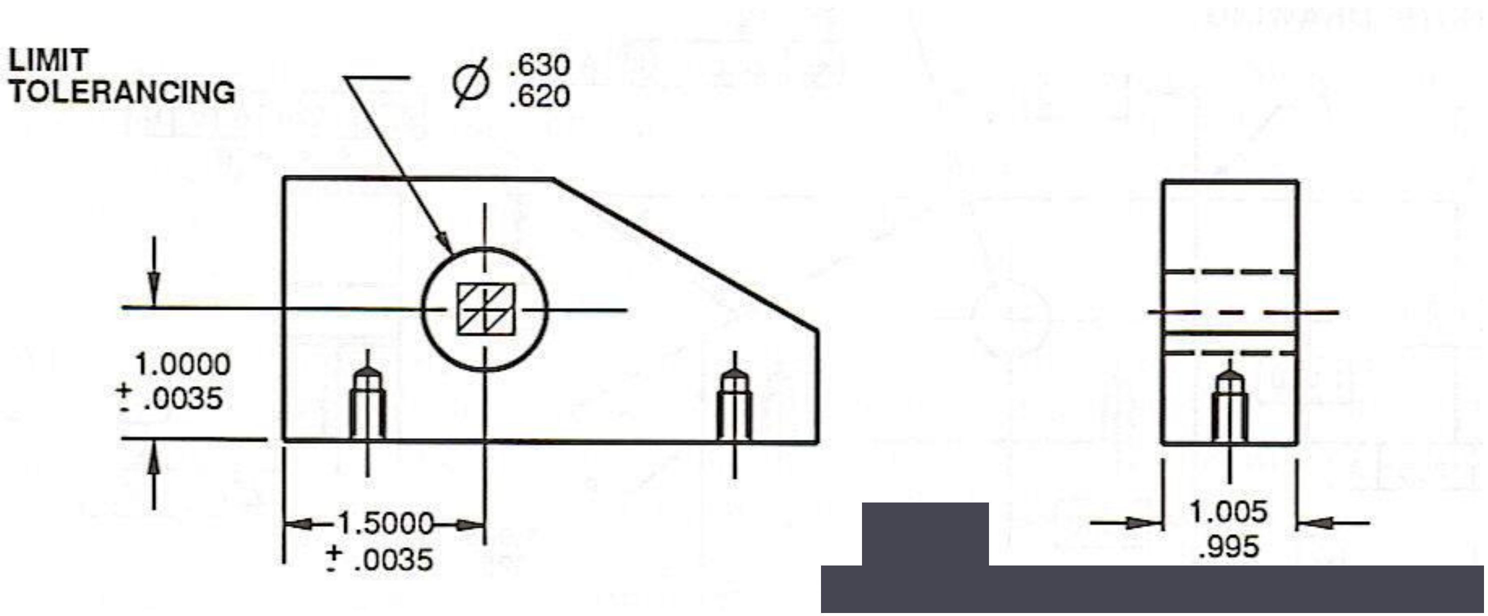


The above drawing depicts the part as the designer intended it to be. In reality, no part can ever be made perfect. It will always be off by a few millionths of an inch. With that in mind, the drawing on the right illustrates how the GD&T instructions control the features of the part. The drawing is *greatly exaggerated* to show what would be undetectable by the naked eye.



Limit (+/-) Tolerancing vs. Geometric Tolerancing

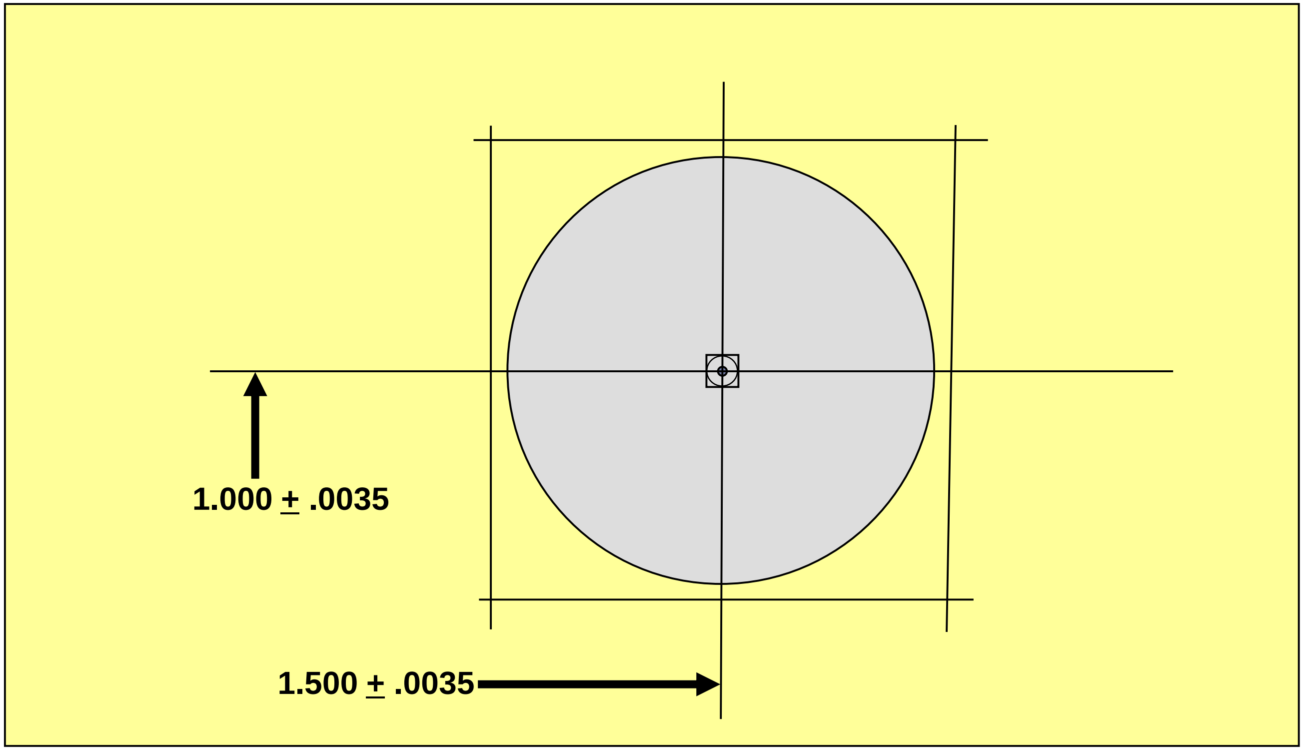
- **Geometric Tolerancing –vs- Limit Tolerancing – What's The Difference?**
  - **This drawing is produced using limit tolerancing. There is no feature control frame, so the design relies on the limits established by the ± dimensions, and the datums are all "implied".**



دانشگاه صنعتی شاهرود

Limit (+/-) Tolerancing vs. Geometric Tolerancing

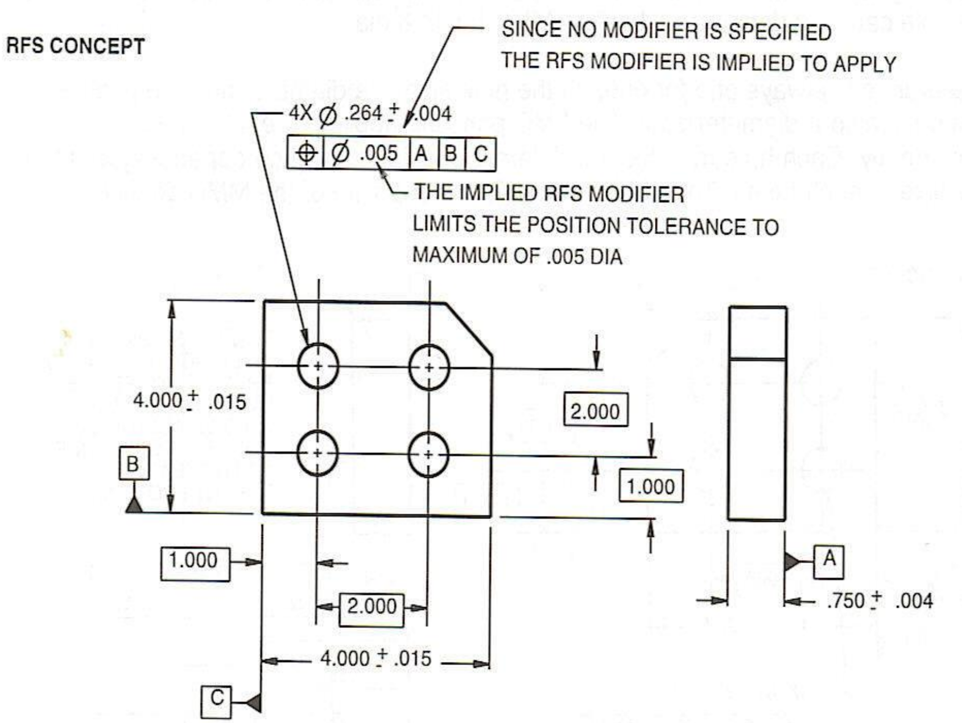
- **Notice that the position of the hole is implied as being oriented from the lower left hand corner. Because we are forced to use the plus/minus .0035 limit tolerance, the hole tolerance zone ends up looking like a square. A close look at the part reveals that the axis of the hole can be off farther in a diagonal direction than across the flat sides.**



Limit (+/-) Tolerancing vs. Geometric Tolerancing

Regardless of Feature Size – RFS

- **Modifier rule # 2 states that unless otherwise specified, all geometric tolerances are by default implied to be RFS – Regardless of Feature Size. Since all unspecified tolerances apply at RFS, there is no need for a RFS symbol. The drawing below illustrates how RFS affects the location tolerance of a feature.**



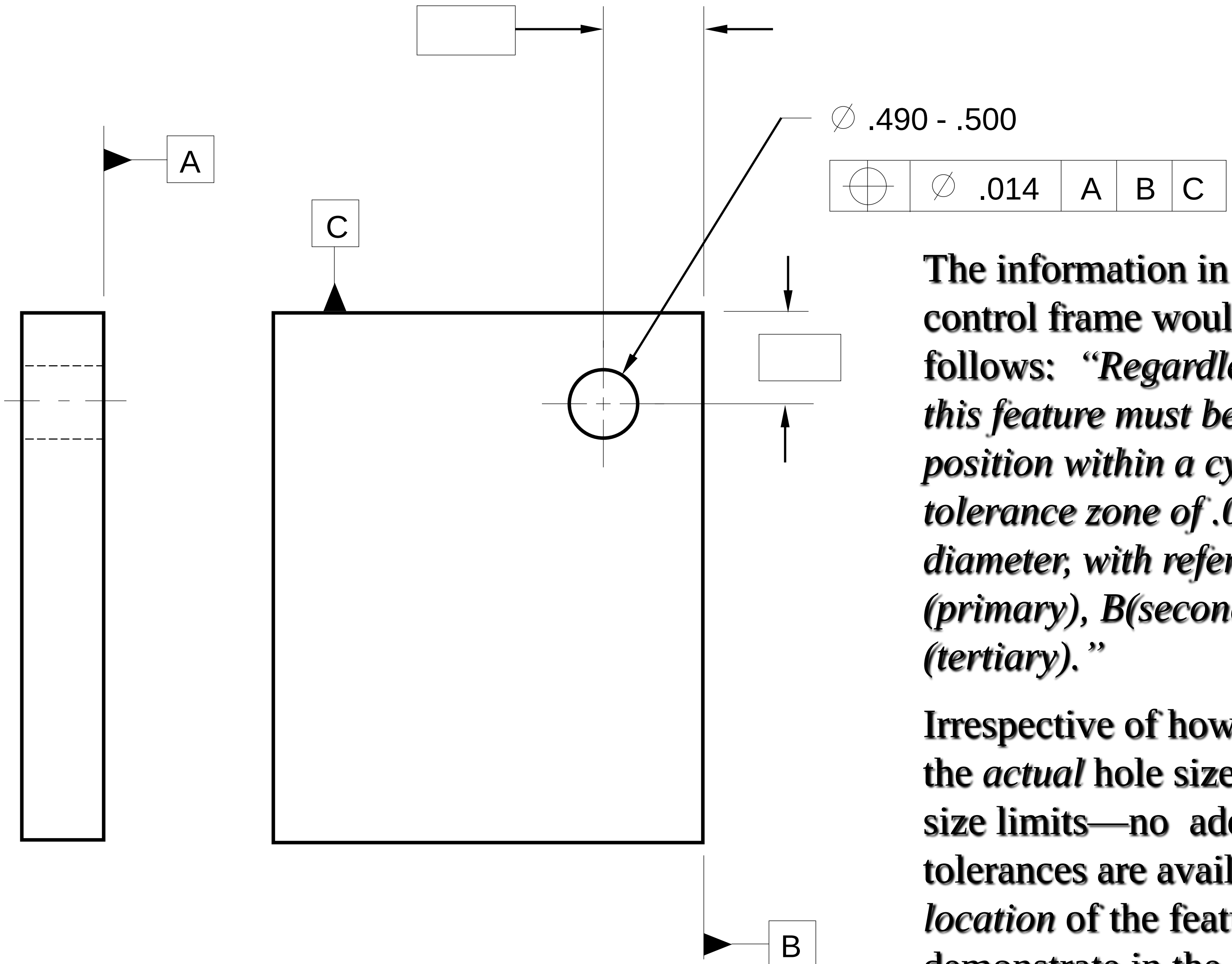
What this means to the machinist is that no matter if the holes are machined at the upper limit of .268 or the lower limit of .260, their location is still restricted to the .005 position tolerance zone.

دانشگاه صنعتی شاهرود

Examples

- Industrial Sketchs
  - Bearing
  - Gaskets
  - Valve Assembly (Valve, Valve Guide, Valve Seat)
- Tolerance of Position: RSF Versus MMC

Tolerance of Position -- RFS

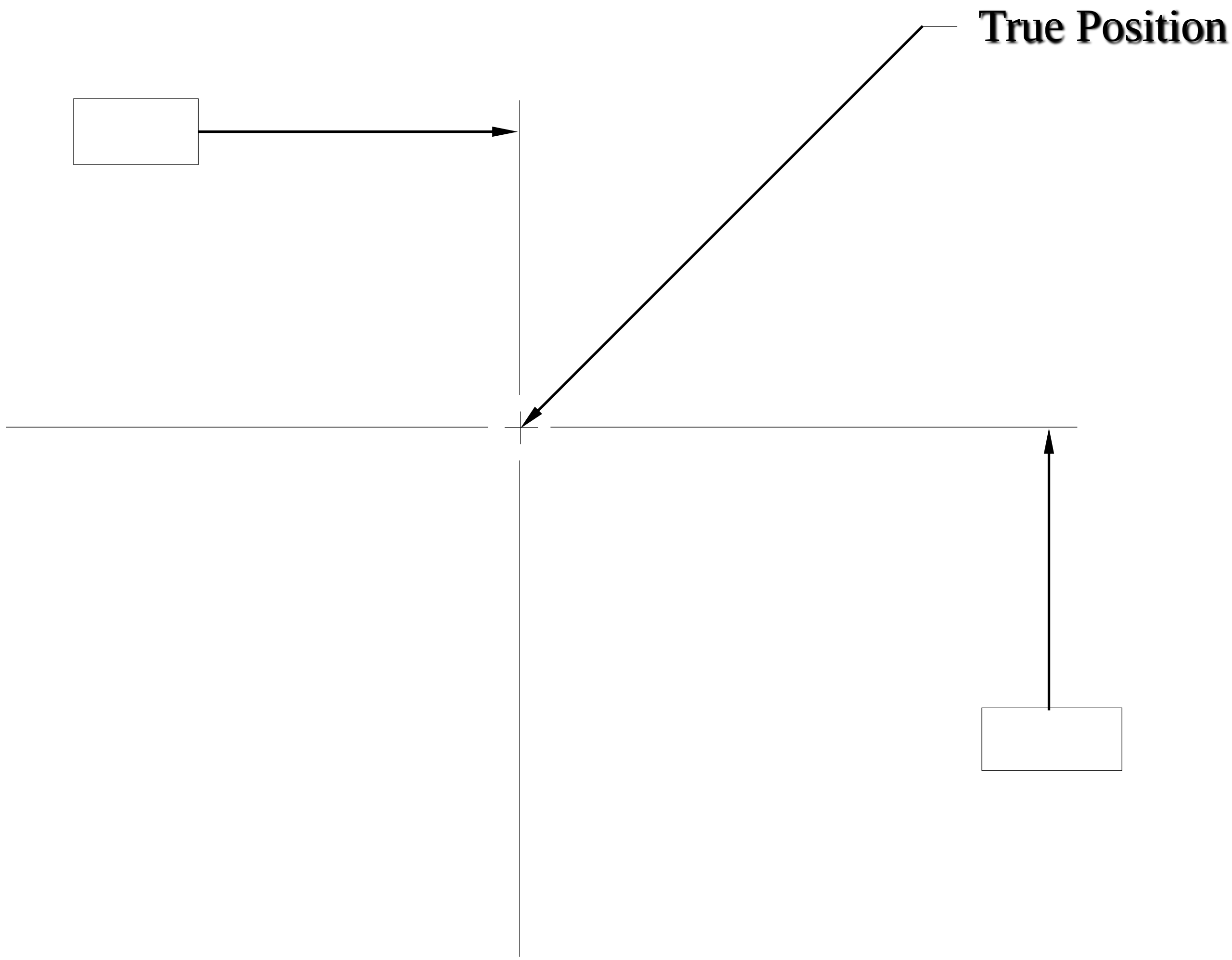


The information in the feature control frame would be read as follows: *"Regardless of feature size, this feature must be located on true position within a cylindrical tolerance zone of .014 in. on diameter, with reference to datums A (primary), B(secondary), and C (tertiary)."*

Irrespective of how large or small the *actual* hole size is—within its size limits—no additional tolerances are available for the *location* of the feature. I'll demonstrate in the next few slides.

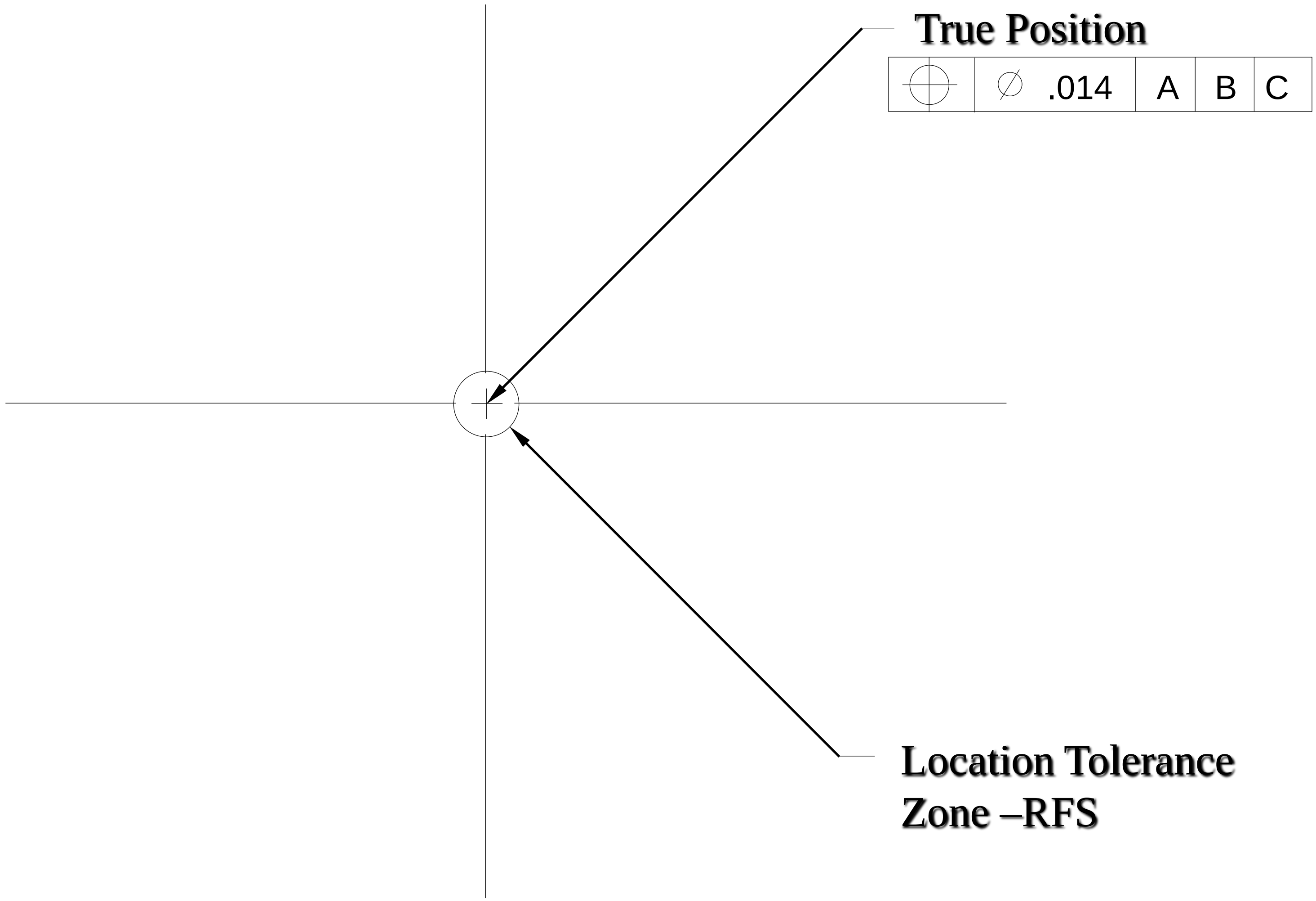
Tolerance of Position -- RFS

The exact location of the hole is established with basic dimensions.



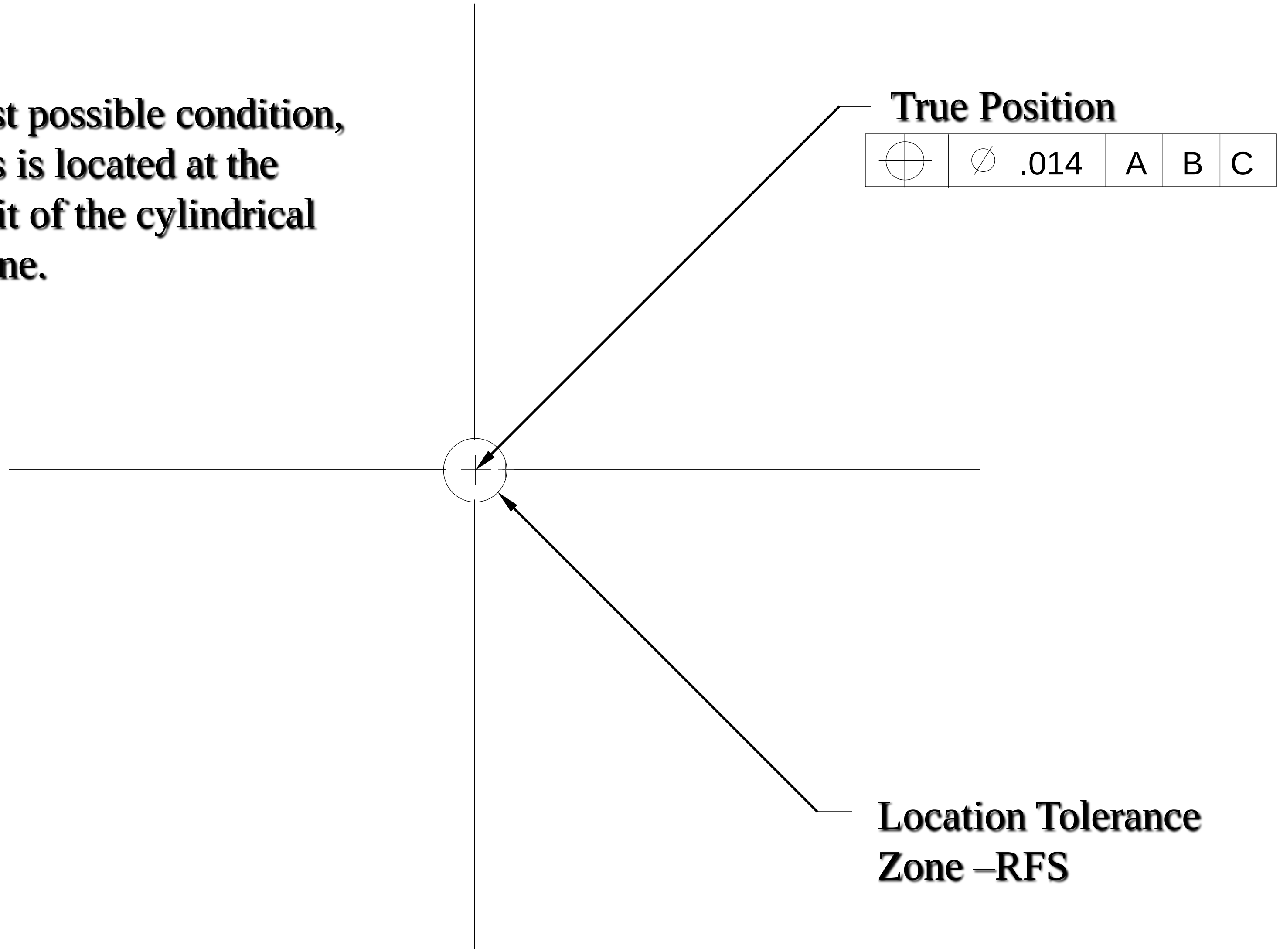
Tolerance of Position -- RFS

The cylindrical tolerance zone is established in the feature control frame –(∅ .014).



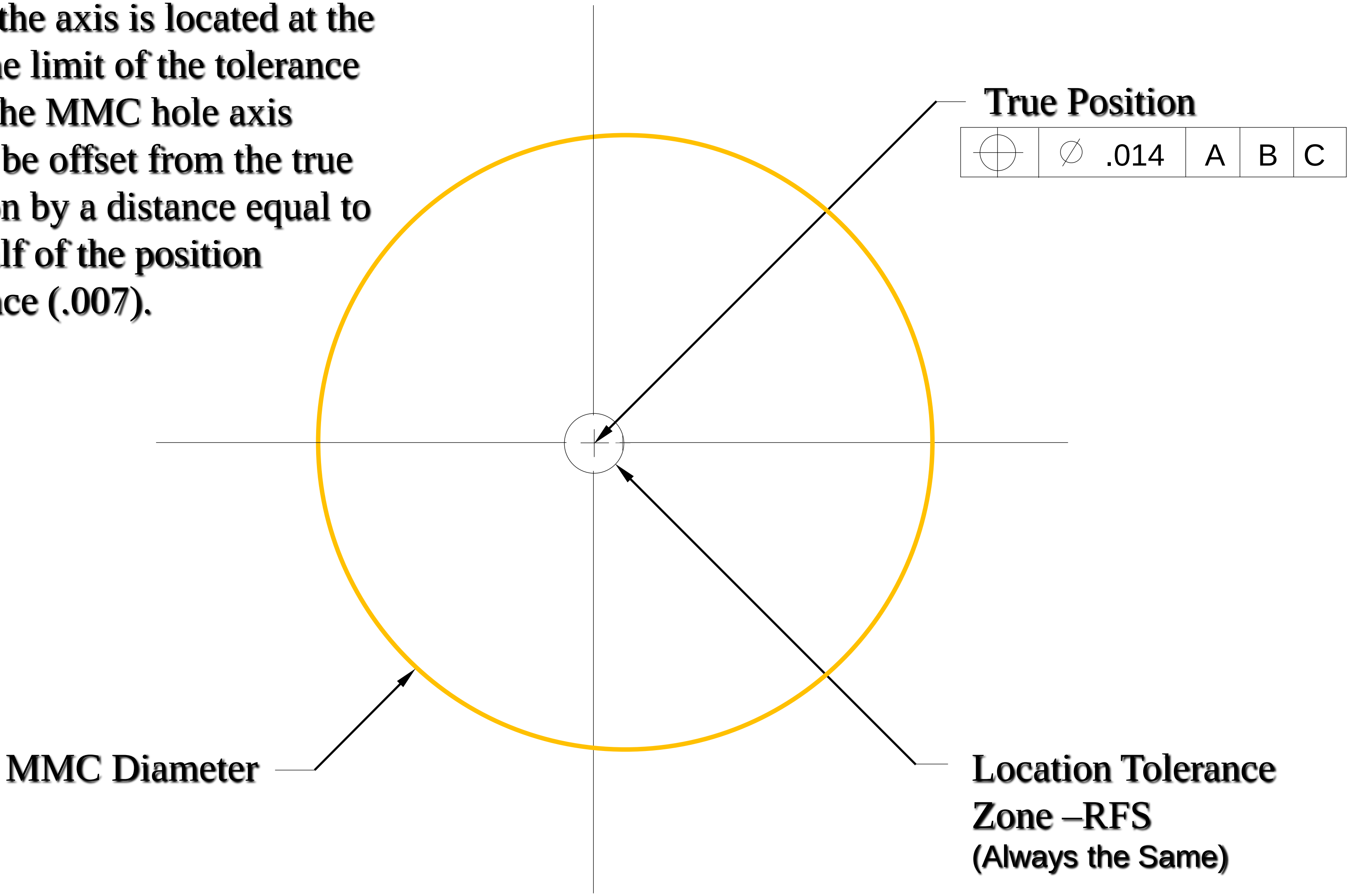
Tolerance of Position -- RFS

For the worst possible condition, the hole axis is located at the extreme limit of the cylindrical tolerance zone.



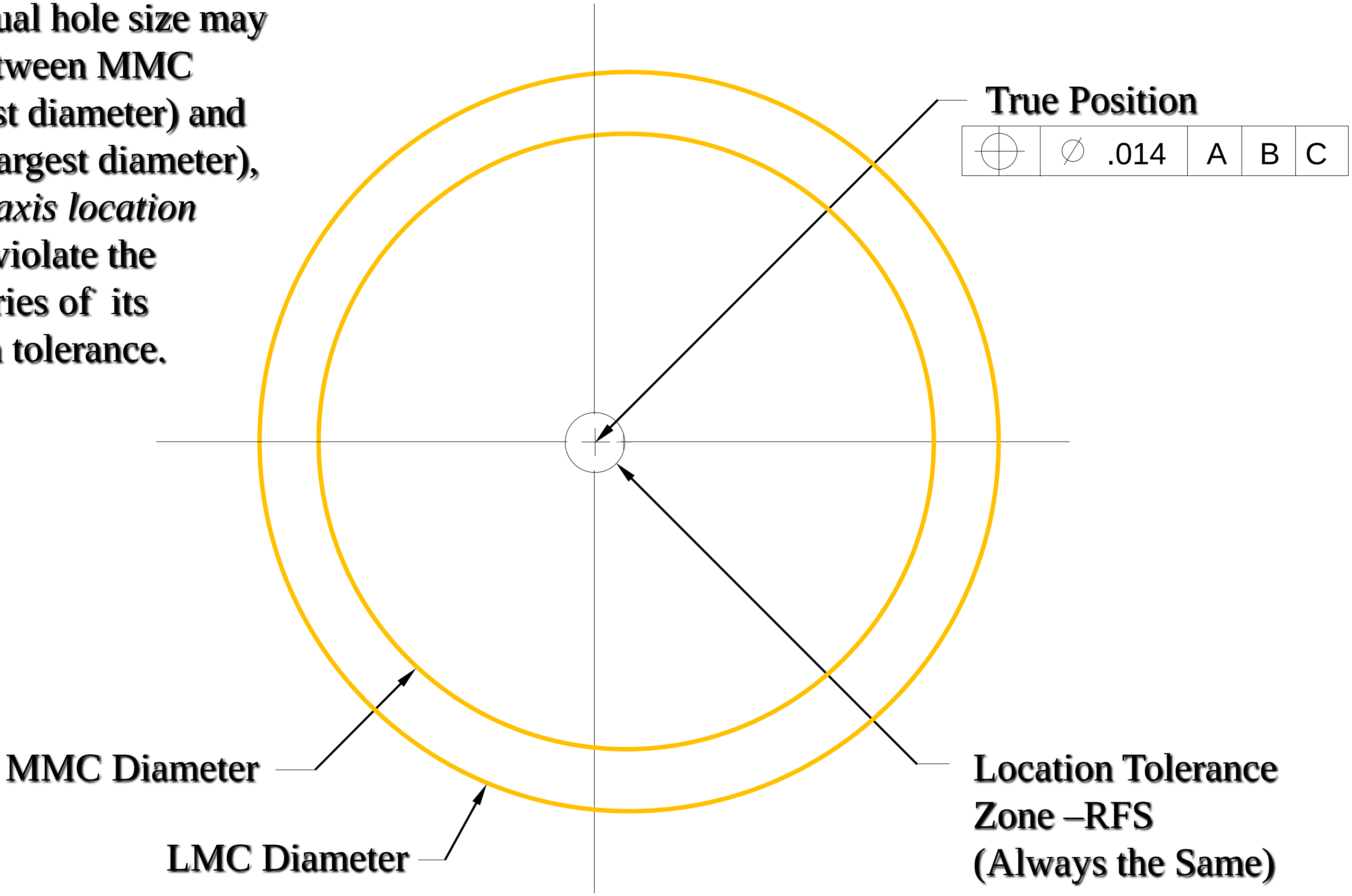
Tolerance of Position -- RFS

When the axis is located at the extreme limit of the tolerance zone, the MMC hole axis would be offset from the true position by a distance equal to one-half of the position tolerance (.007).



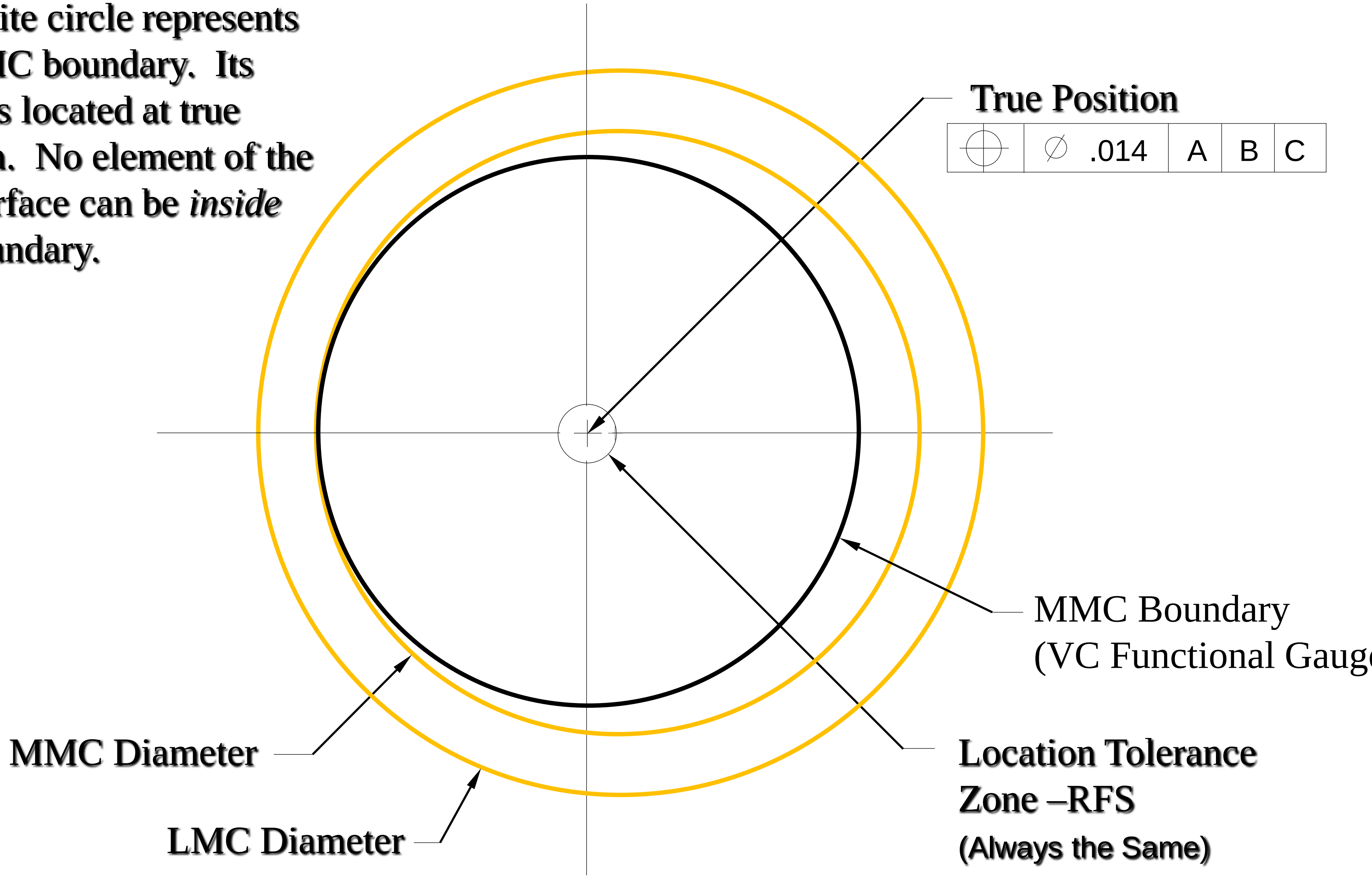
Tolerance of Position -- RFS

The actual hole size may vary between MMC (smallest diameter) and LMC (largest diameter), but the *axis location* cannot violate the boundaries of its location tolerance.



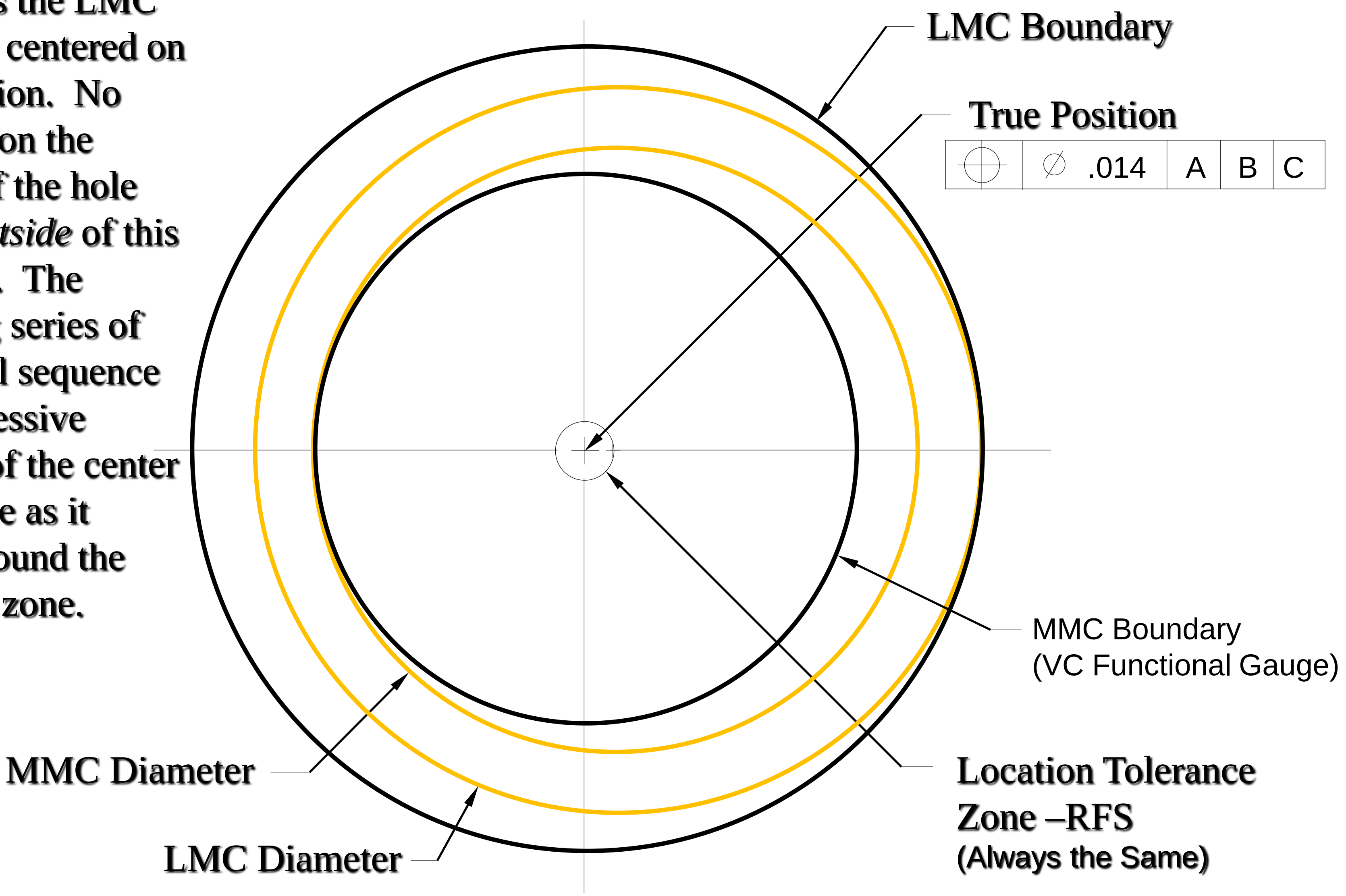
Tolerance of Position -- RFS

The white circle represents the MMC boundary. Its center is located at true position. No element of the hole surface can be *inside* this boundary.



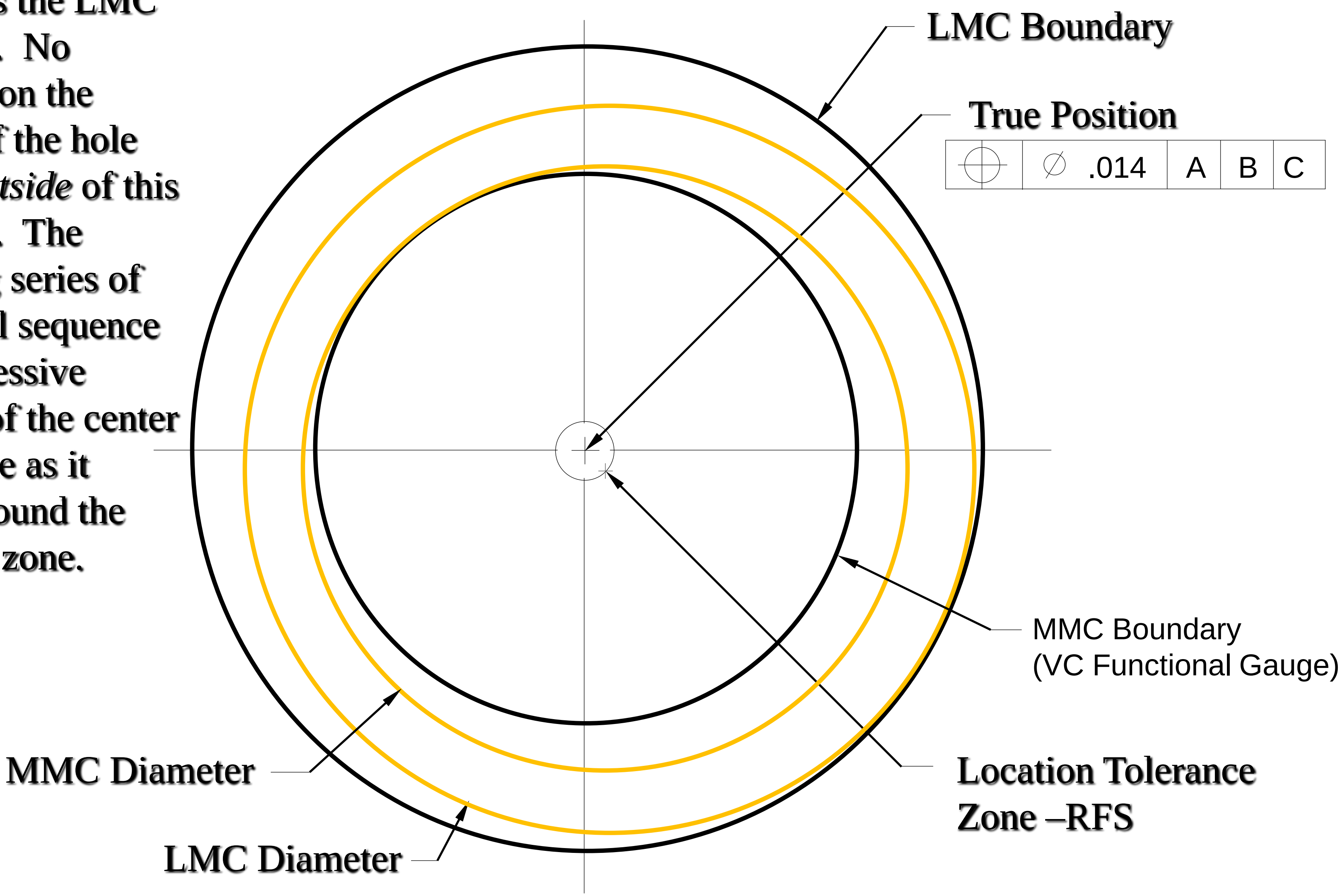
Tolerance of Position -- RFS

The outer white circle represents the LMC boundary centered on true position. No elements on the surface of the hole can be *outside* of this boundary. The following series of slides will sequence the progressive position of the center of the hole as it moves around the tolerance zone.



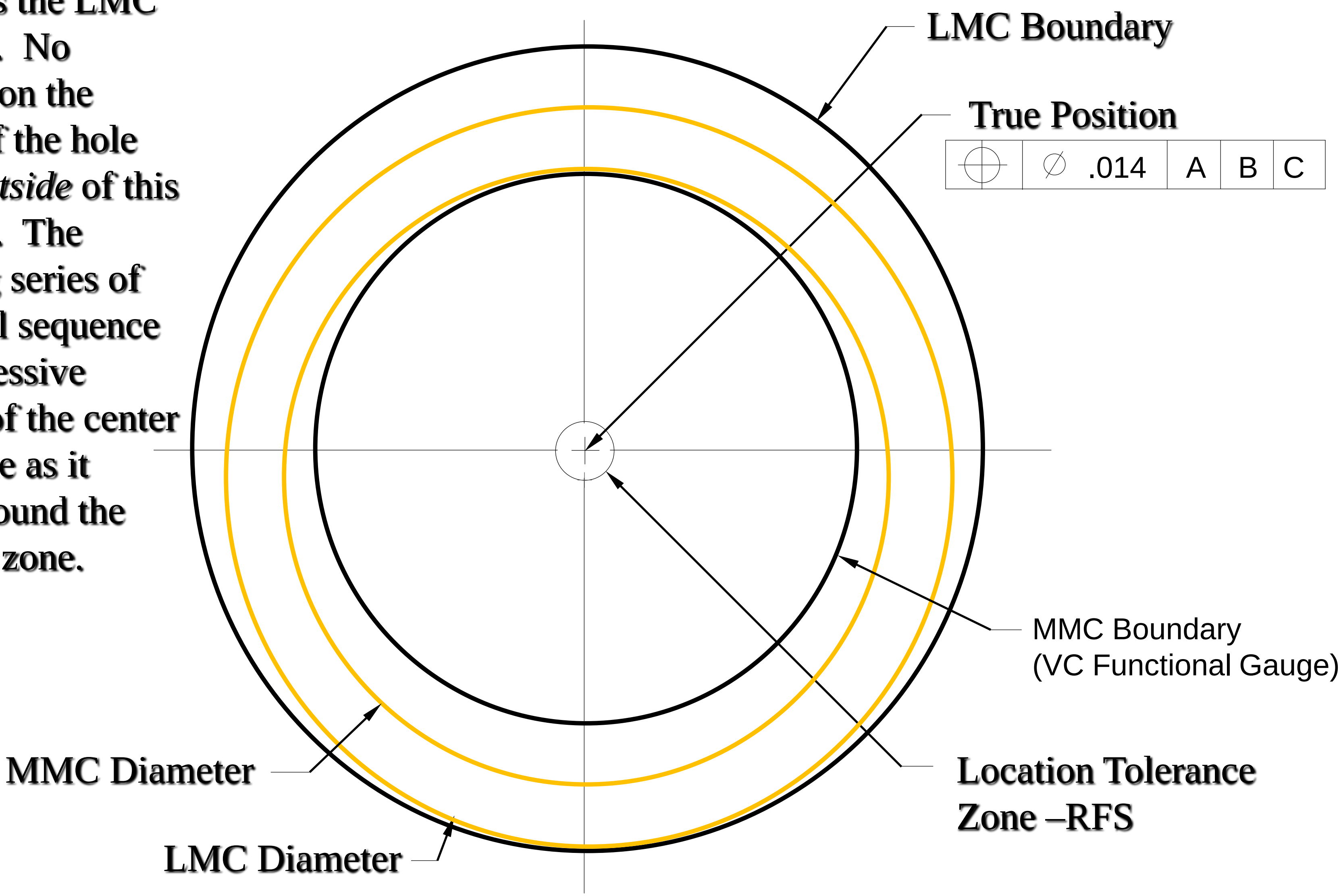
Tolerance of Position -- RFS

The outer white circle represents the LMC boundary. No elements on the surface of the hole can be *outside* of this boundary. The following series of slides will sequence the progressive position of the center of the hole as it moves around the tolerance zone.



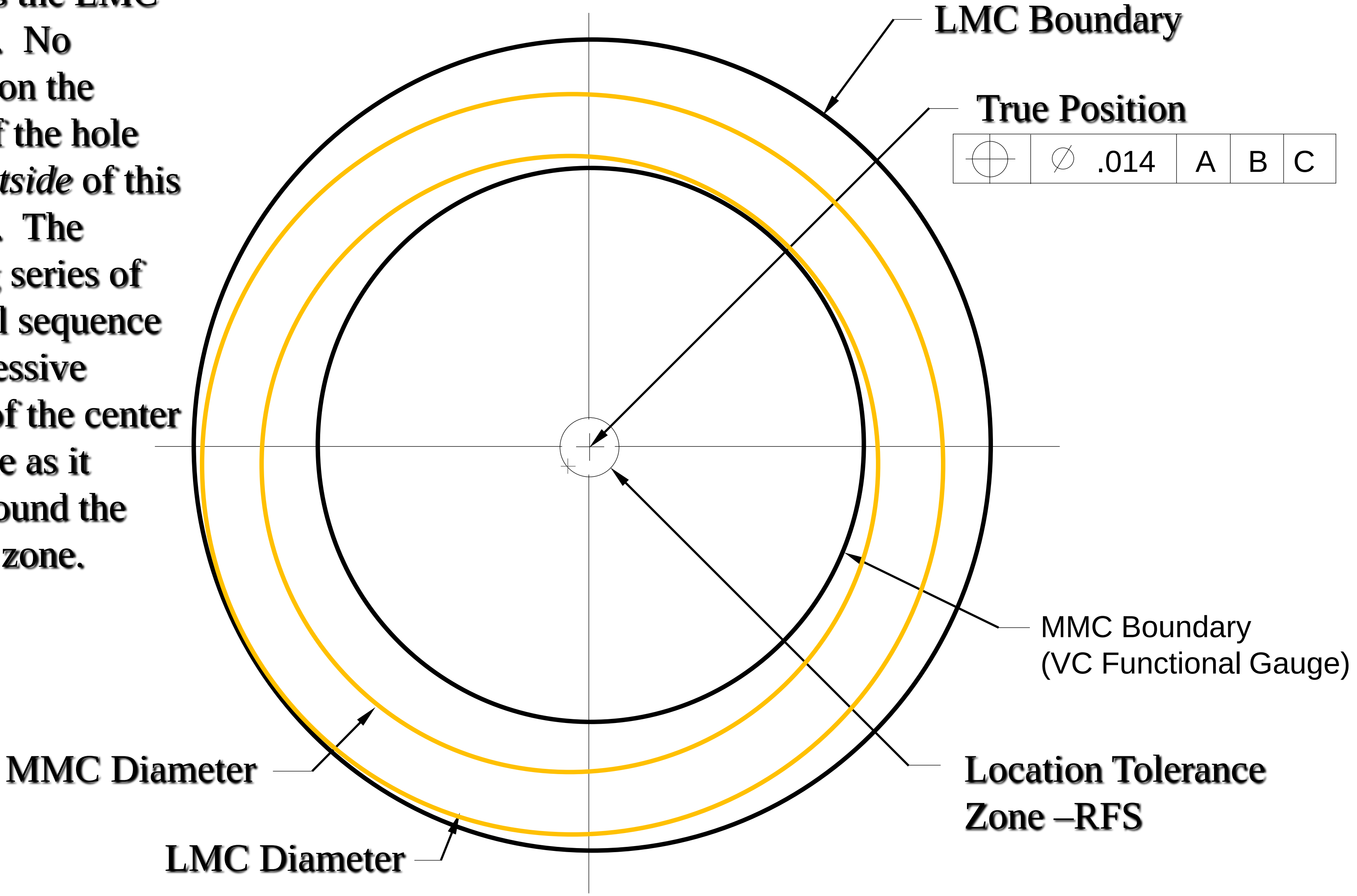
Tolerance of Position -- RFS

The outer white circle represents the LMC boundary. No elements on the surface of the hole can be *outside* of this boundary. The following series of slides will sequence the progressive position of the center of the hole as it moves around the tolerance zone.



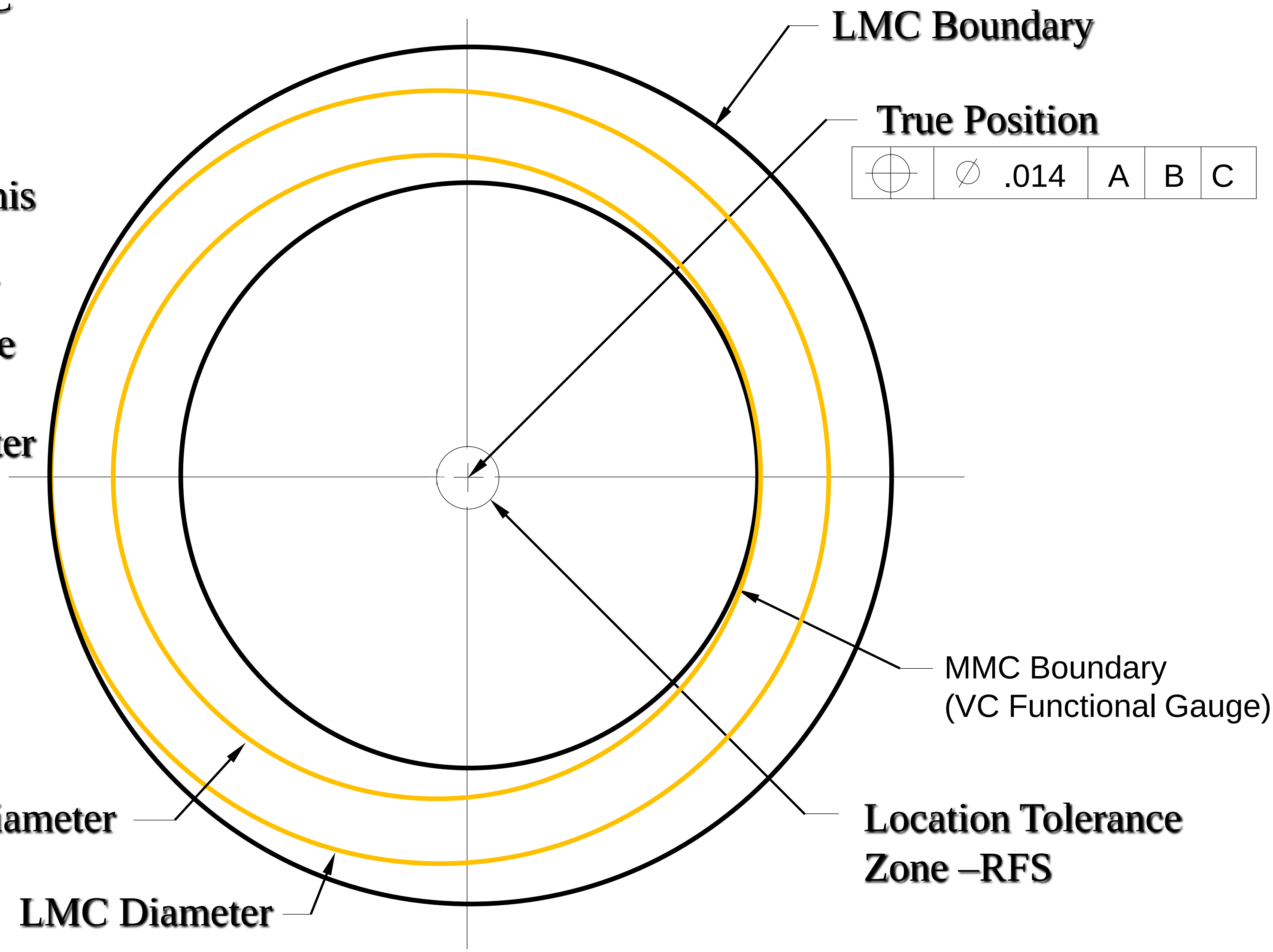
Tolerance of Position -- RFS

The outer white circle represents the LMC boundary. No elements on the surface of the hole can be *outside* of this boundary. The following series of slides will sequence the progressive position of the center of the hole as it moves around the tolerance zone.



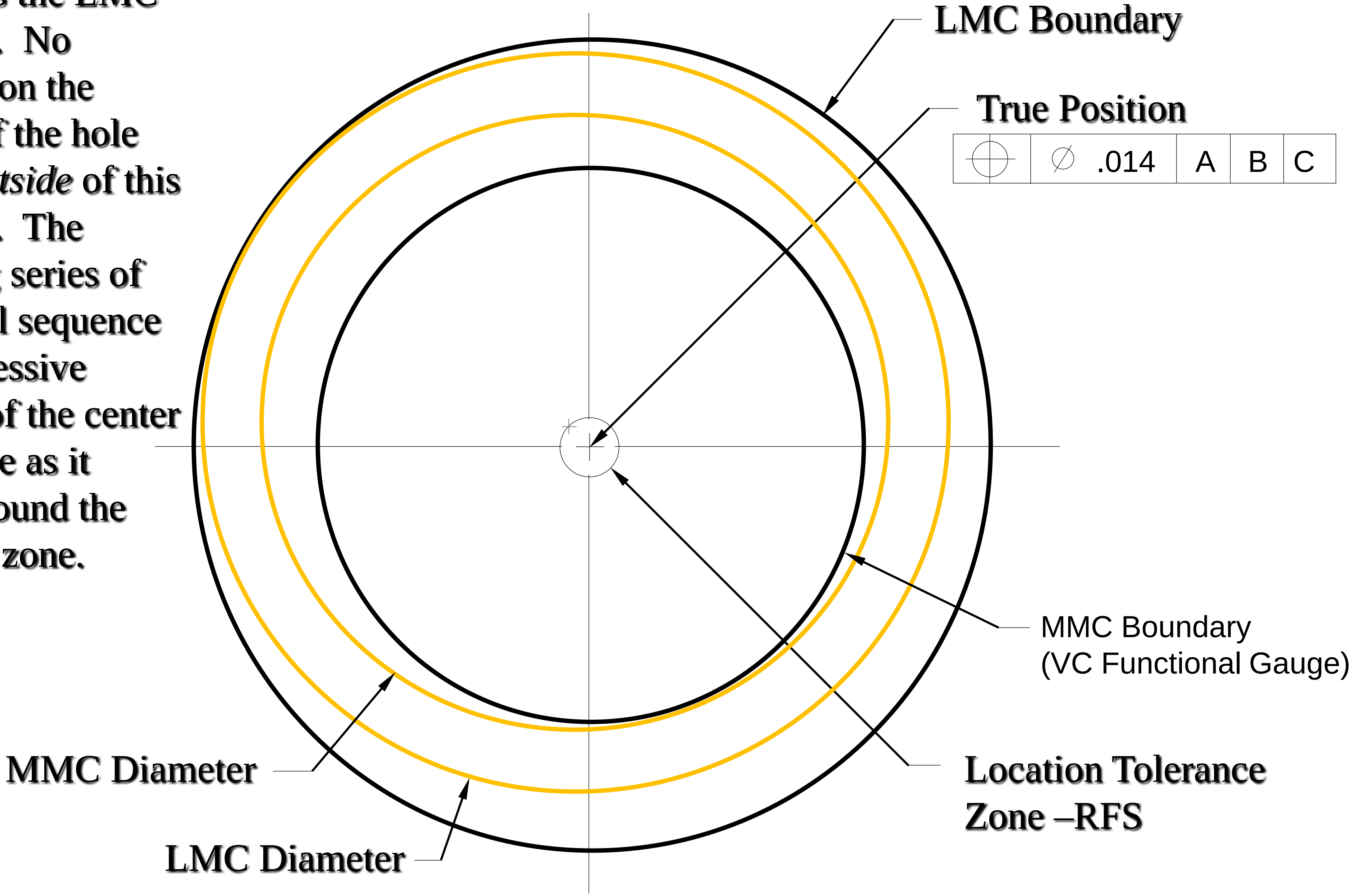
Tolerance of Position -- RFS

The outer white circle represents the LMC boundary. No elements on the surface of the hole can be *outside* of this boundary. The following series of slides will sequence the progressive position of the center of the hole as it moves around the tolerance zone.



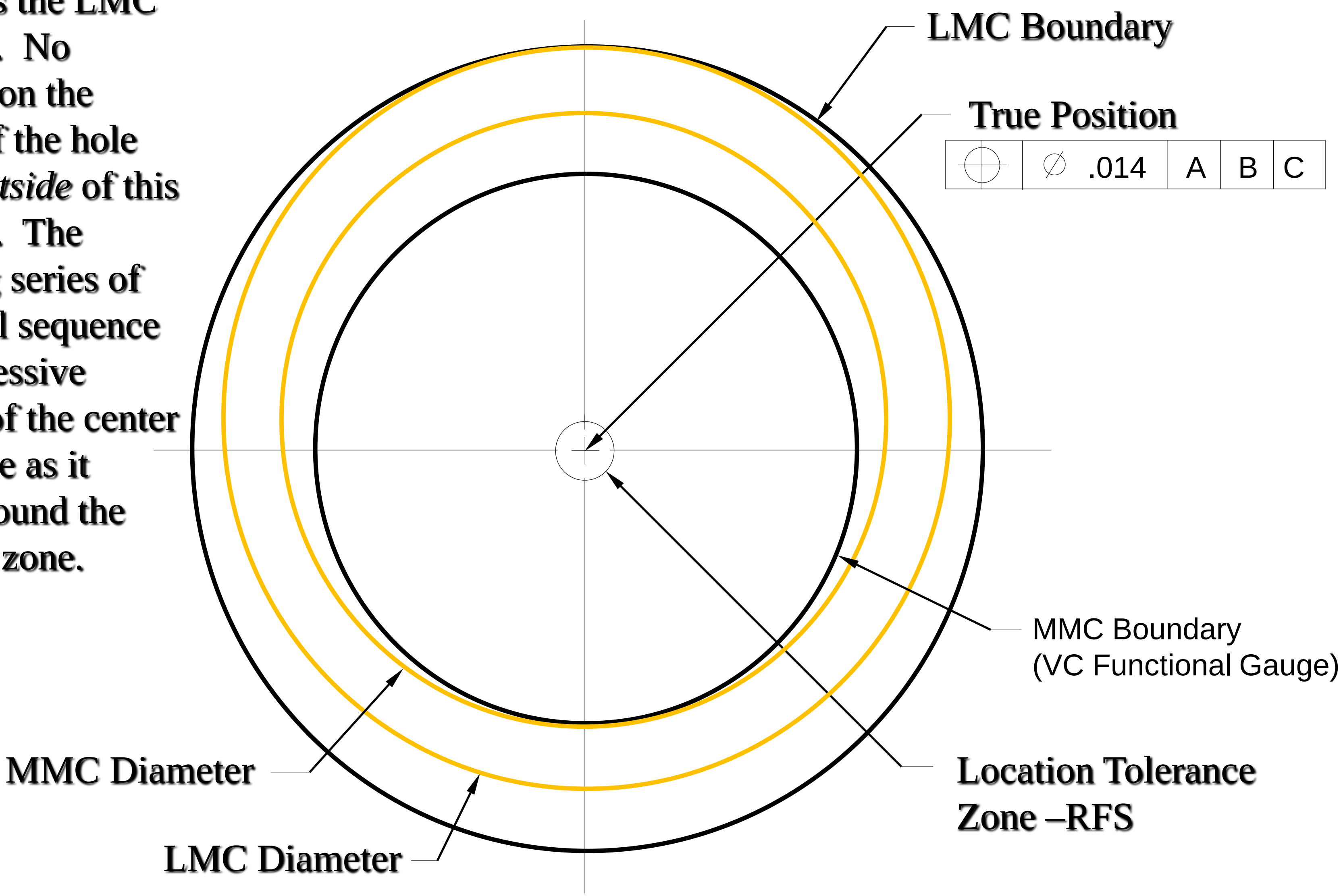
Tolerance of Position -- RFS

The outer white circle represents the LMC boundary. No elements on the surface of the hole can be *outside* of this boundary. The following series of slides will sequence the progressive position of the center of the hole as it moves around the tolerance zone.



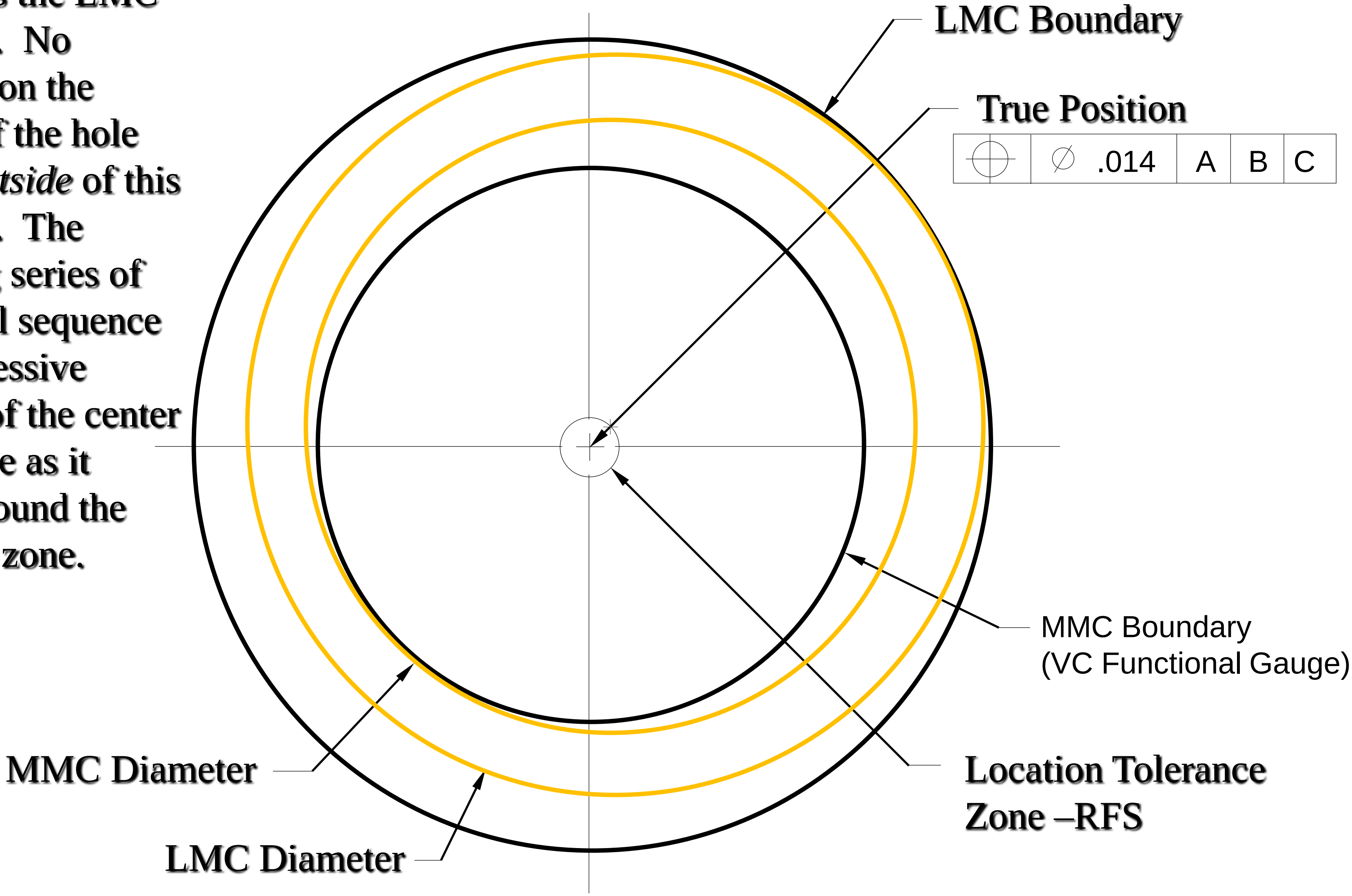
Tolerance of Position -- RFS

The outer white circle represents the LMC boundary. No elements on the surface of the hole can be *outside* of this boundary. The following series of slides will sequence the progressive position of the center of the hole as it moves around the tolerance zone.



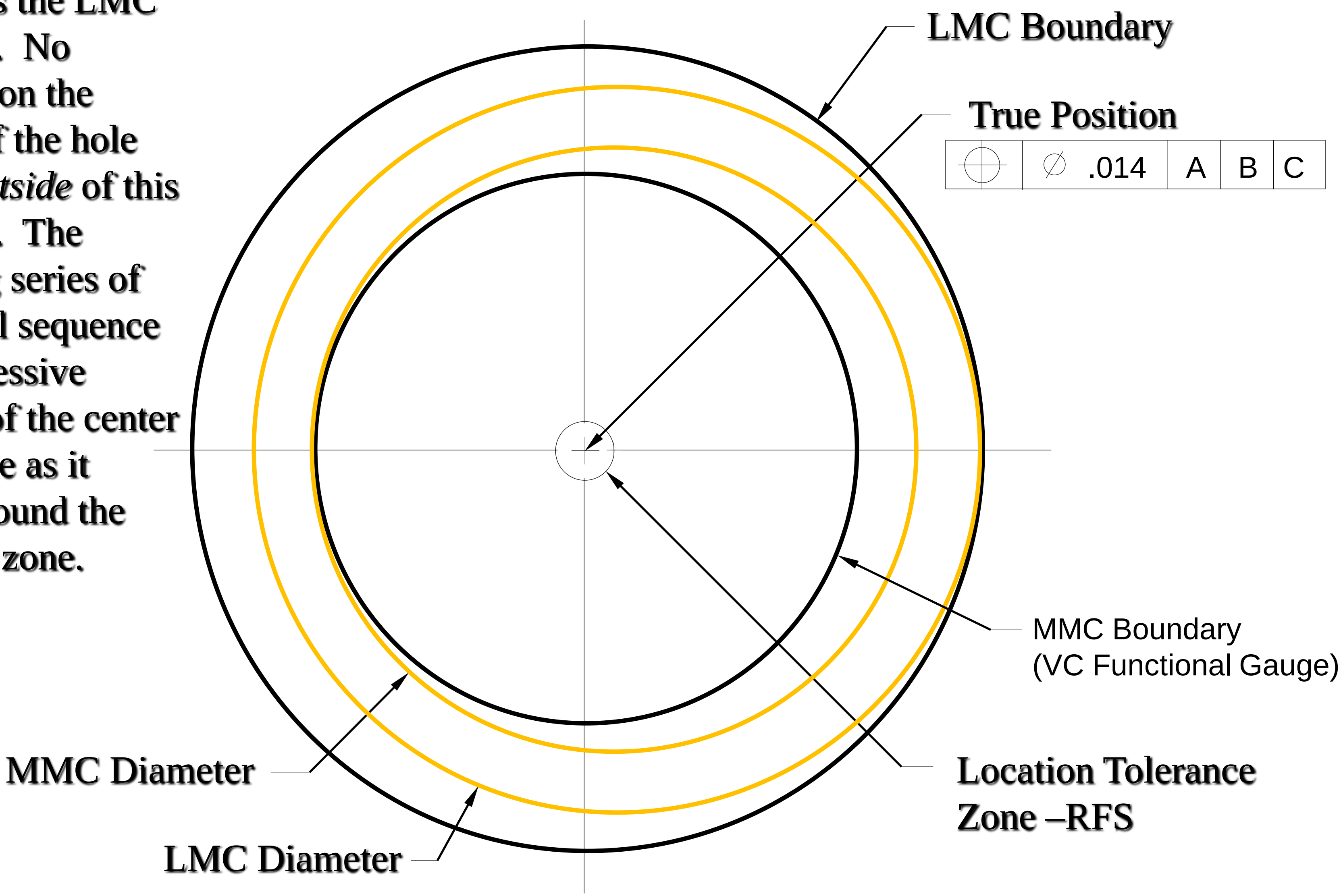
Tolerance of Position -- RFS

The outer white circle represents the LMC boundary. No elements on the surface of the hole can be *outside* of this boundary. The following series of slides will sequence the progressive position of the center of the hole as it moves around the tolerance zone.



Tolerance of Position -- RFS

The outer white circle represents the LMC boundary. No elements on the surface of the hole can be *outside* of this boundary. The following series of slides will sequence the progressive position of the center of the hole as it moves around the tolerance zone.

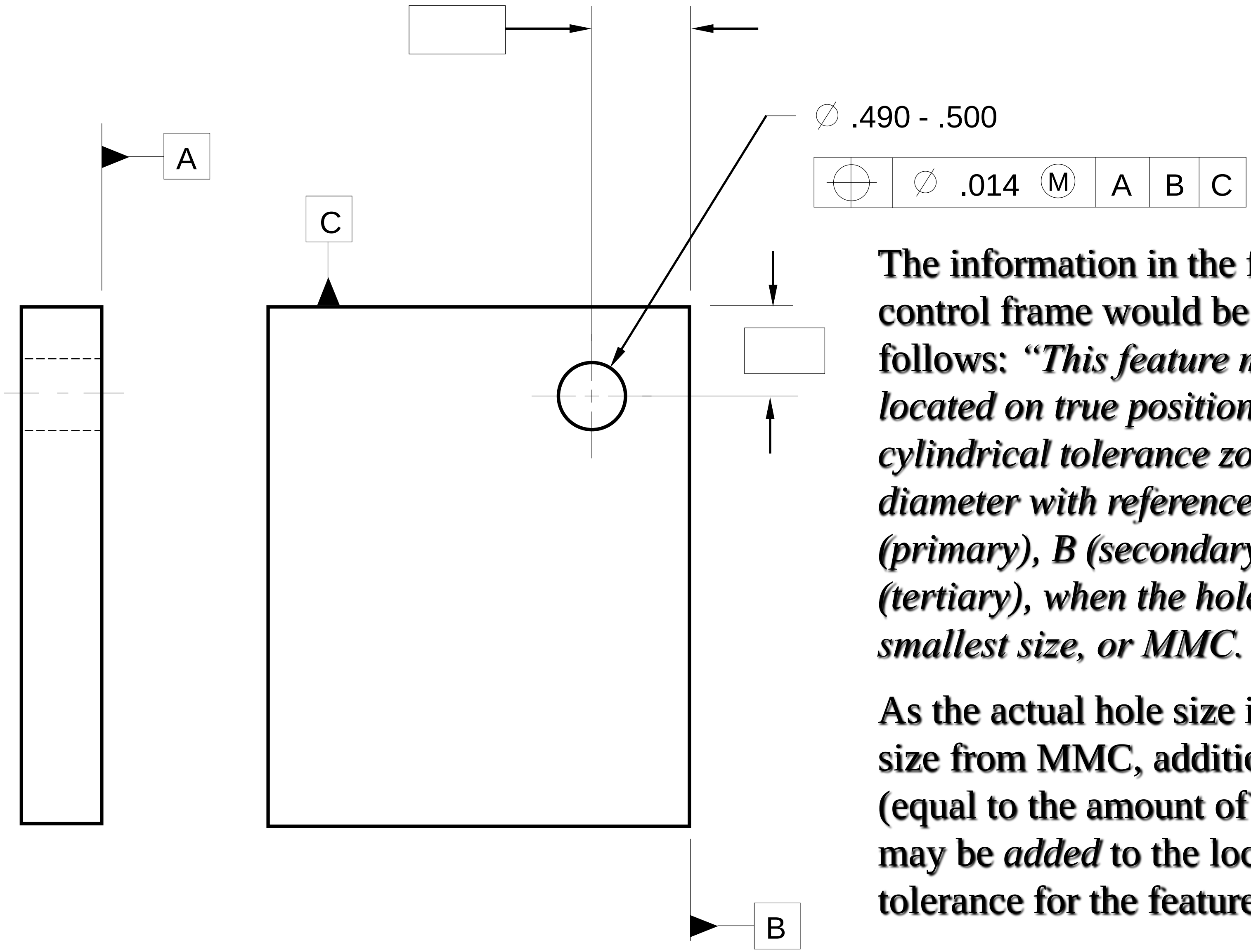


TOLERANCE OF POSITION AT MAXIMUM MATERIAL CONDITION

Tolerance of Position -- MMC

The next example will illustrate the concept of bonus tolerance, in connection with position tolerances. We will use the same drawing example that was used to discuss tolerances of position, when applied regardless of feature size (RFS). *One of the significant differences you will see is the advantages of defining the tolerance zone for the axis of a hole as we did before—but this time, we will add the modifier for maximum material condition (MMC) to the tolerance specification in the feature control frame. Notice the changes that occur in location tolerances when modifiers are used, and as departure from MMC occurs.*

Tolerance of Position -- MMC

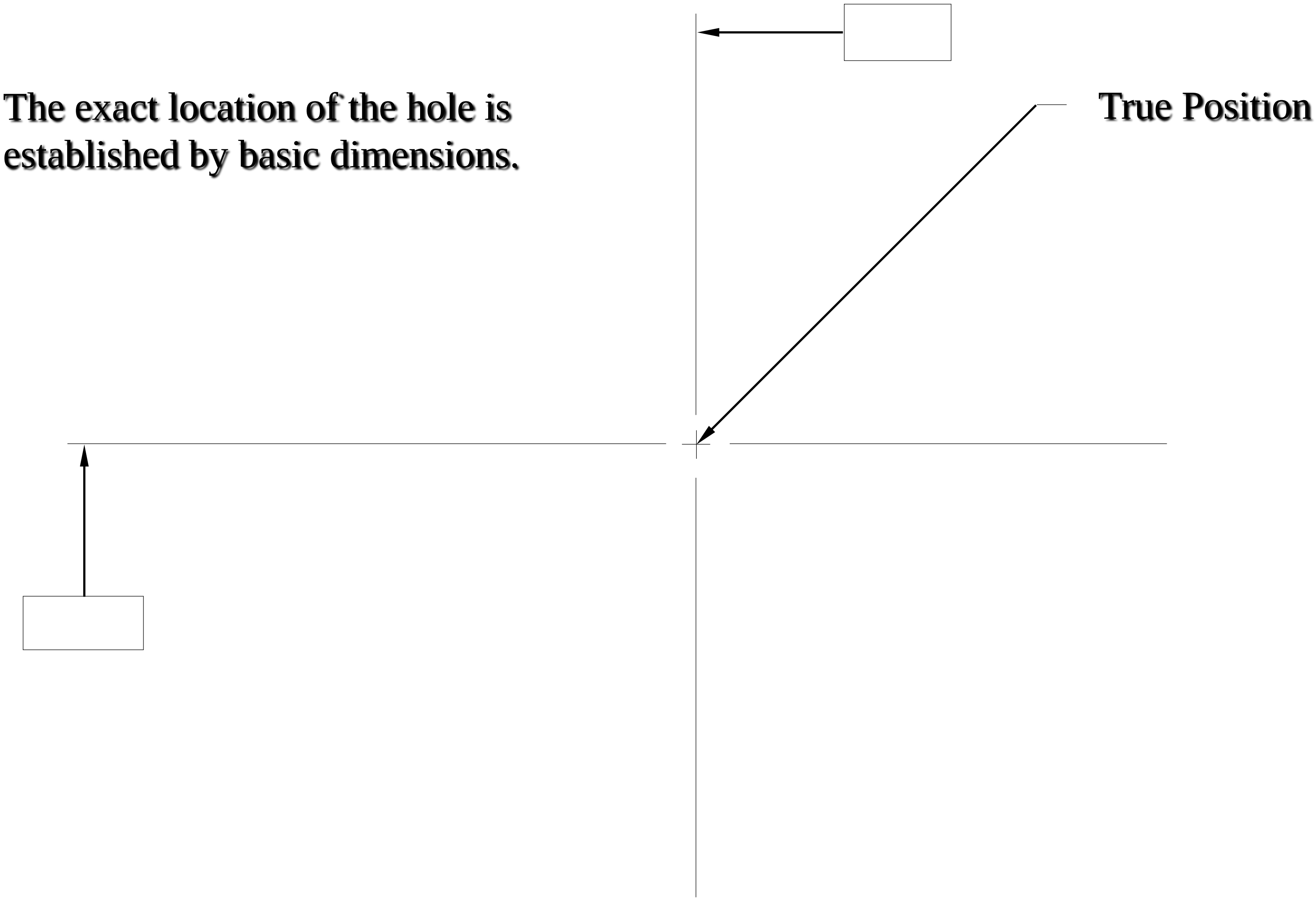


The information in the feature control frame would be read as follows: *"This feature must be located on true position within a cylindrical tolerance zone of .014 on diameter with reference to datums A (primary), B (secondary), and C (tertiary), when the hole is at its smallest size, or MMC."*

As the actual hole size increases in size from MMC, additional tolerance (equal to the amount of departure) may be *added* to the location tolerance for the feature.

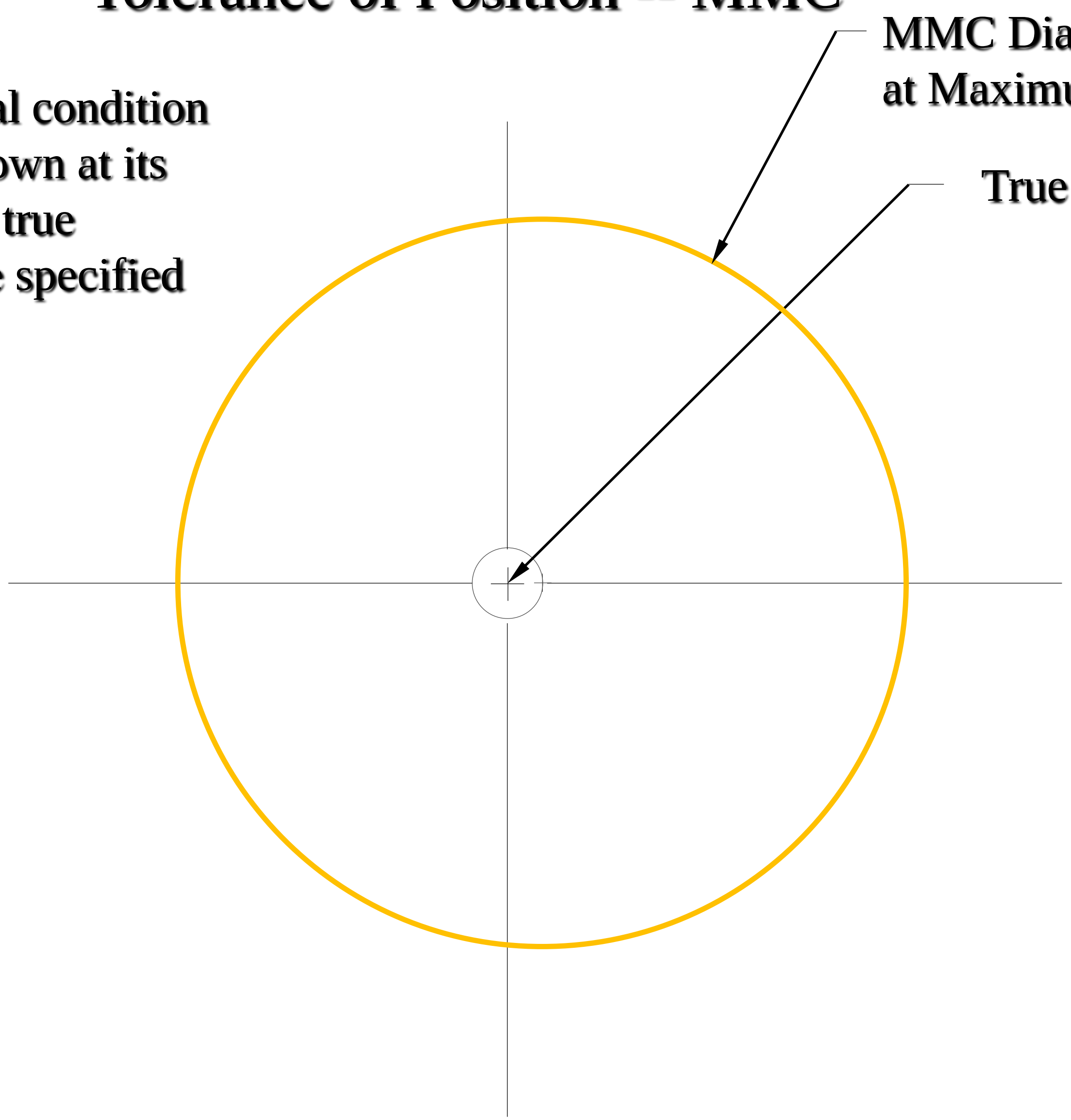
Tolerance of Position -- MMC

The exact location of the hole is established by basic dimensions.



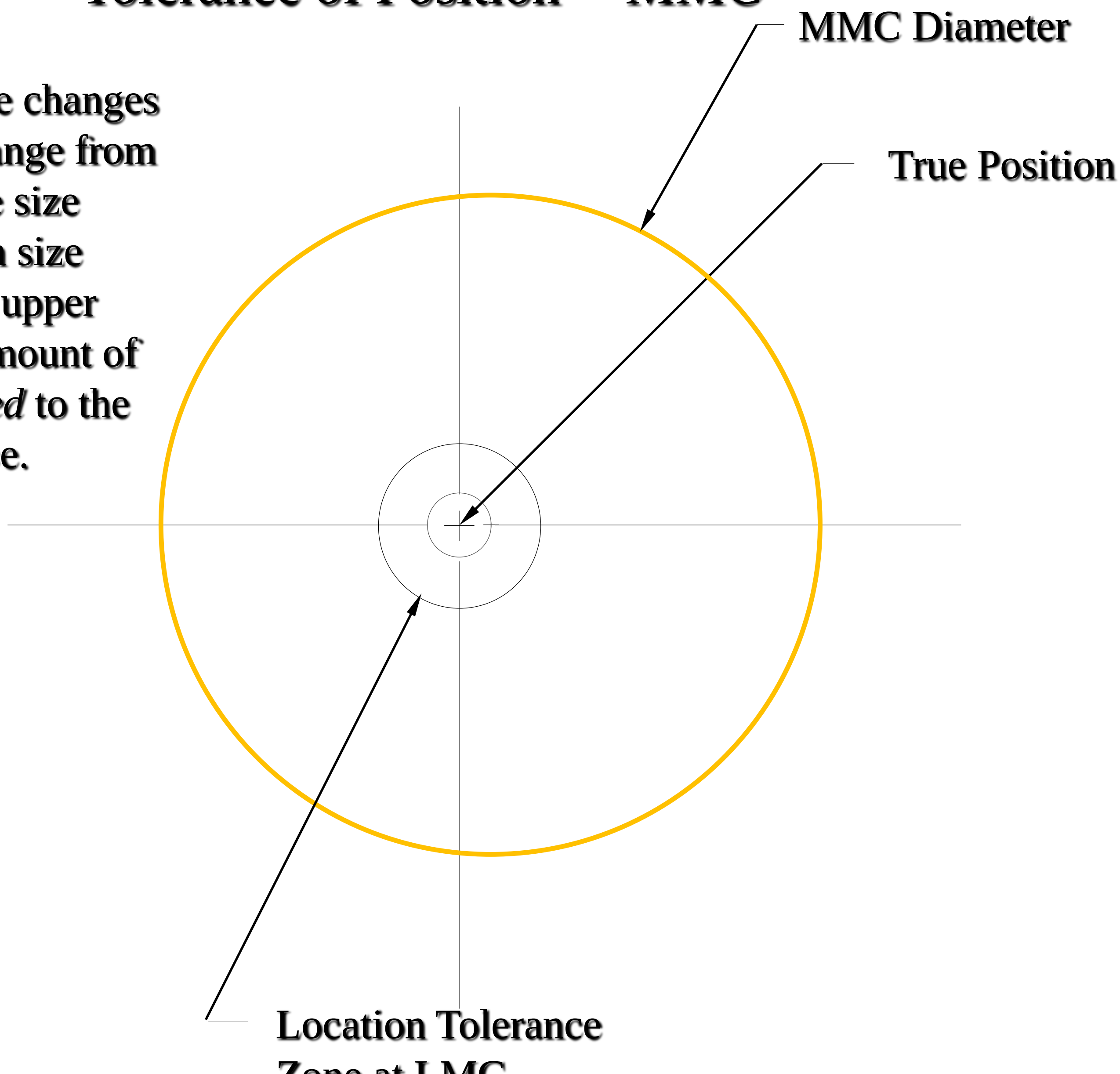
Tolerance of Position -- MMC

The maximum material condition diameter of .490 is shown at its maximum offset from true position—one-half the specified location tolerance.



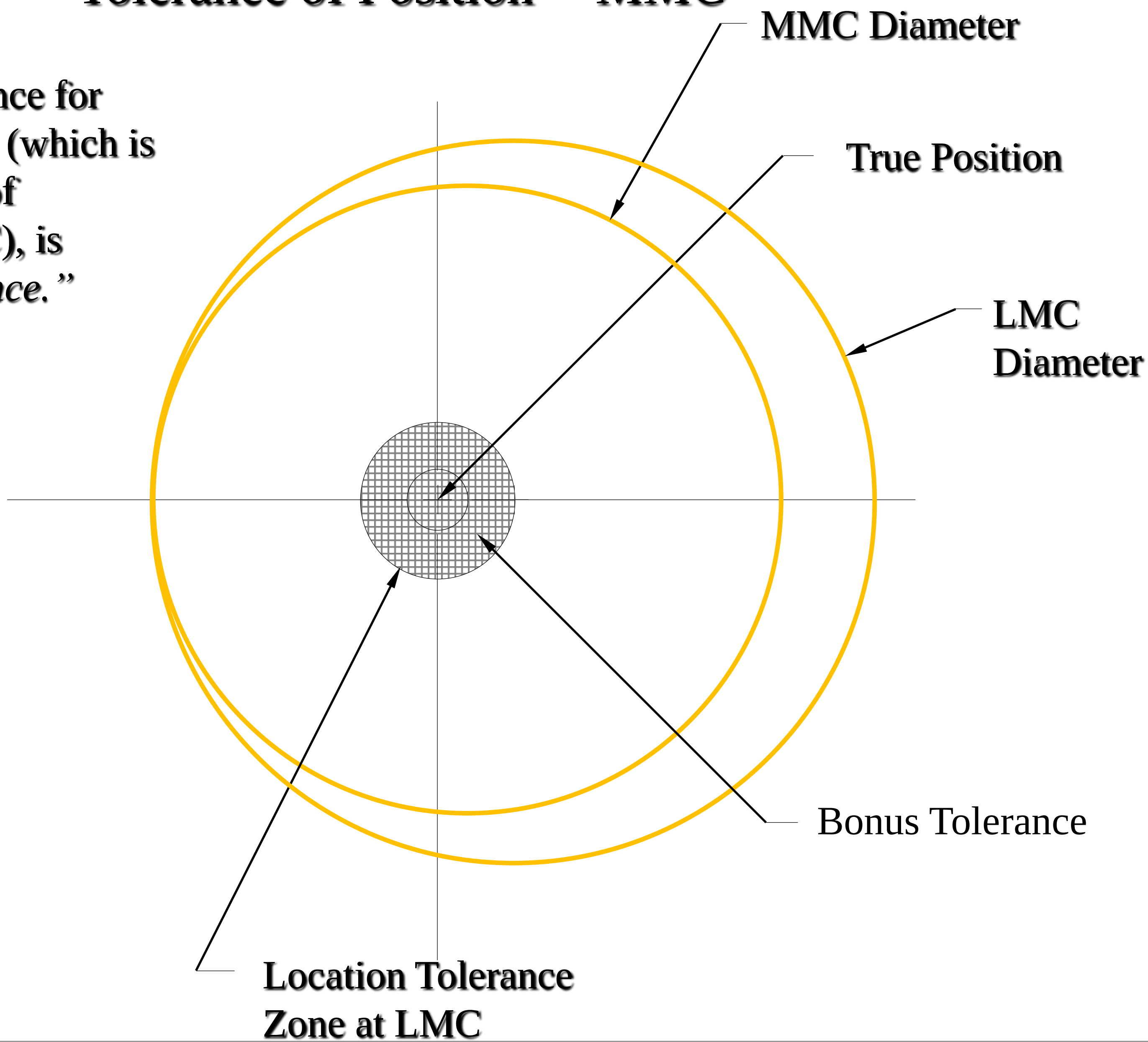
Tolerance of Position -- MMC

As the size of the hole changes within its tolerance range from MMC—smallest hole size limit, and increases in size towards the LMC, or upper size limit, an equal amount of tolerance can be *added* to the axis *location* tolerance.



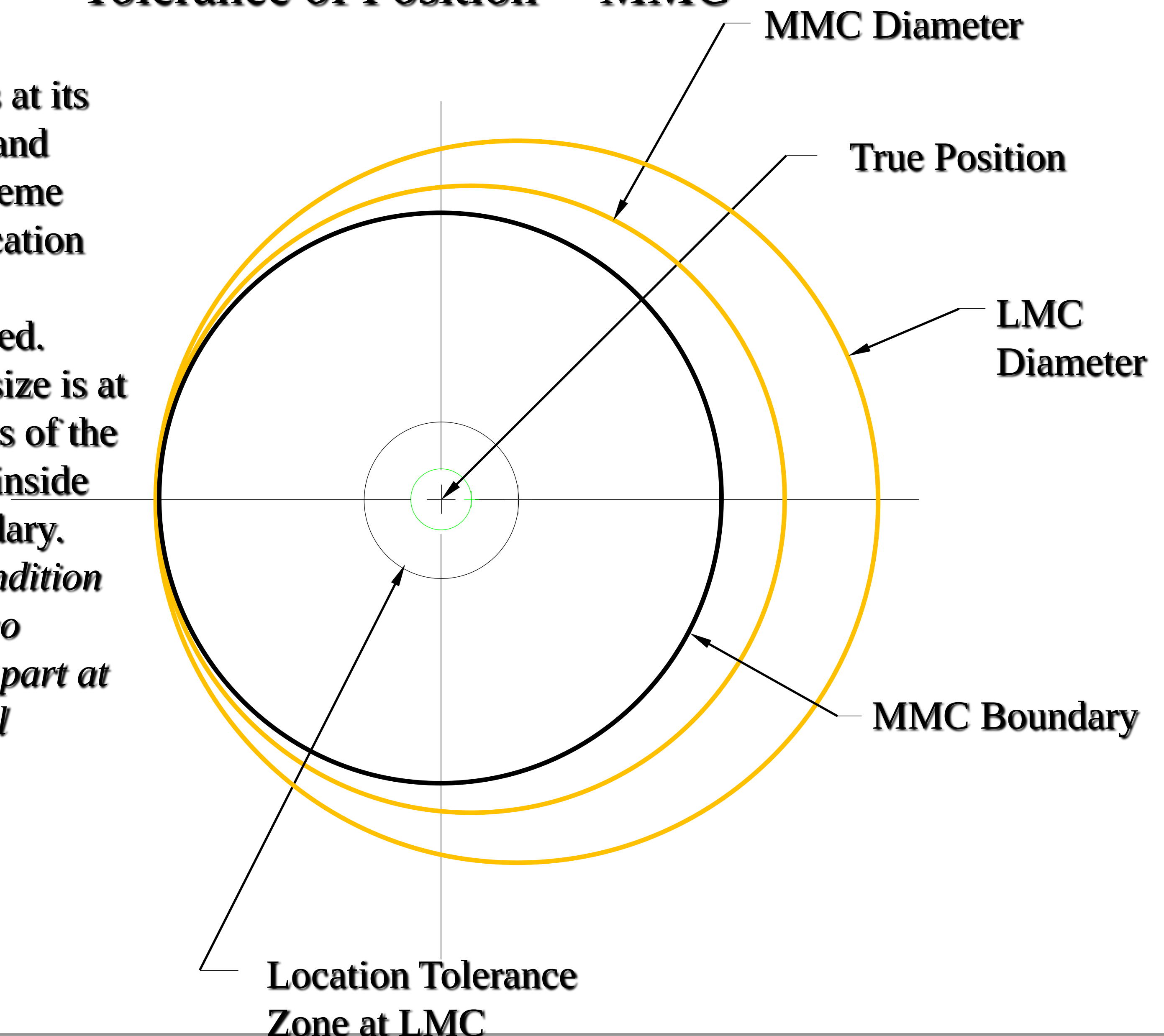
Tolerance of Position -- MMC

The additional tolerance for the hole axis location (which is equal to the amount of departure from MMC), is called *"bonus tolerance."*



Tolerance of Position -- MMC

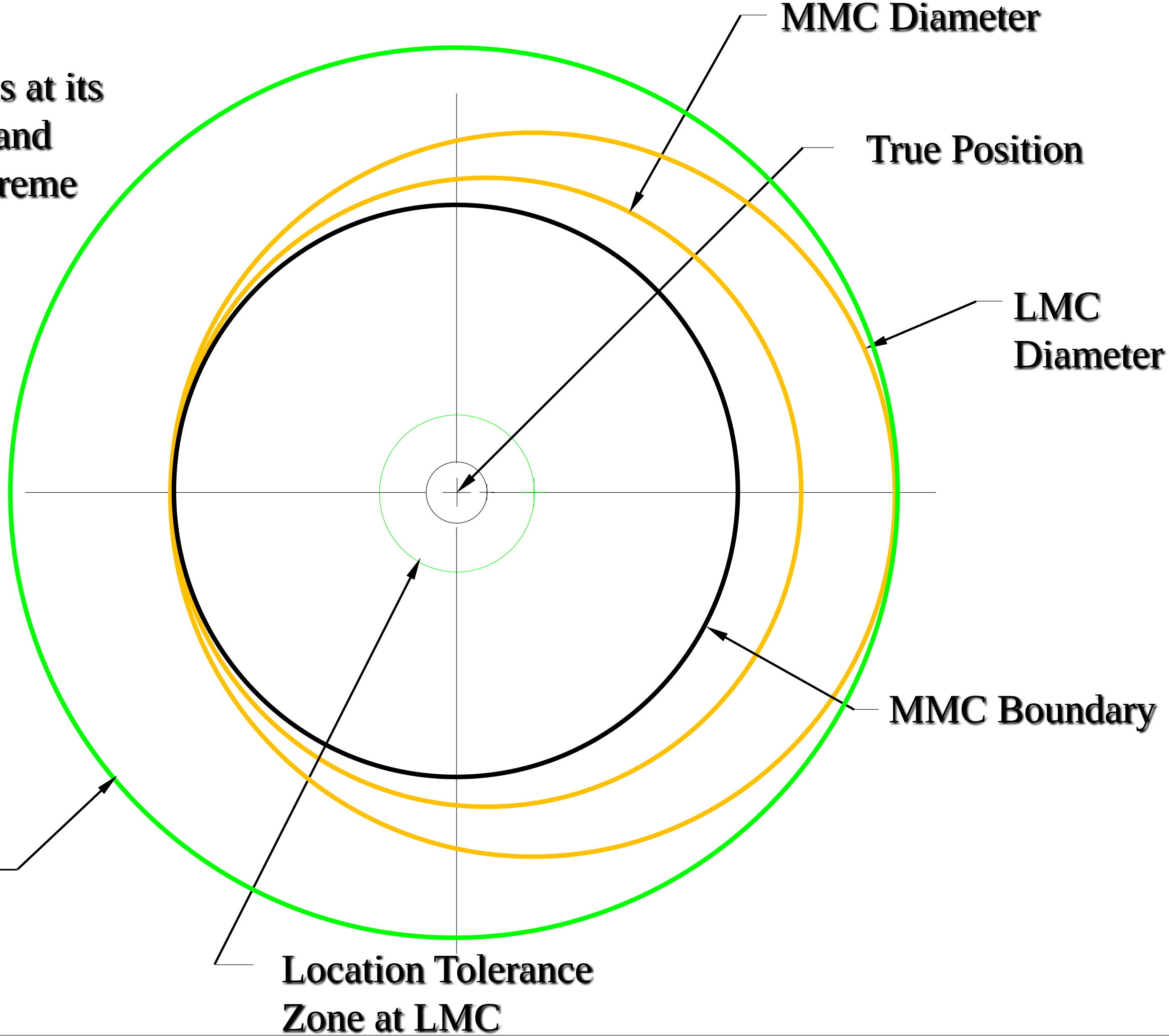
When the hole size is at its lower limit (MMC), and positioned at the extreme limit of the MMC location tolerance, the MMC boundary is established. When the feature of size is at this limit, no elements of the hole *surface* may be inside this theoretical boundary. *This is the virtual condition of the hole, which also simulates the mating part at its maximum material condition.*



Tolerance of Position -- MMC

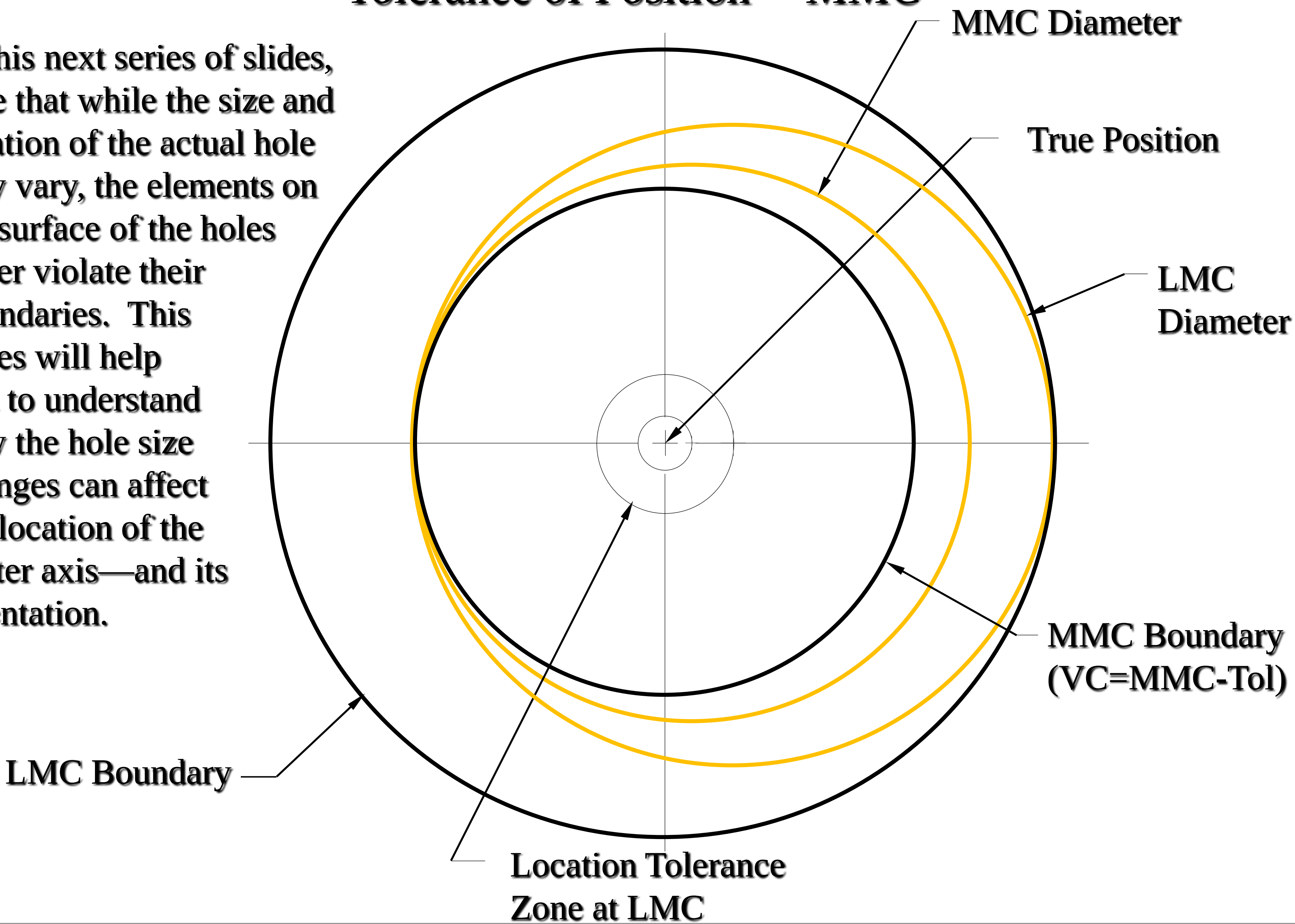
When the hole size is at its upper limit (LMC), and positioned at the extreme limit of the location tolerance, the LMC boundary is established.

No elements of the hole surface can be out-side this boundary.



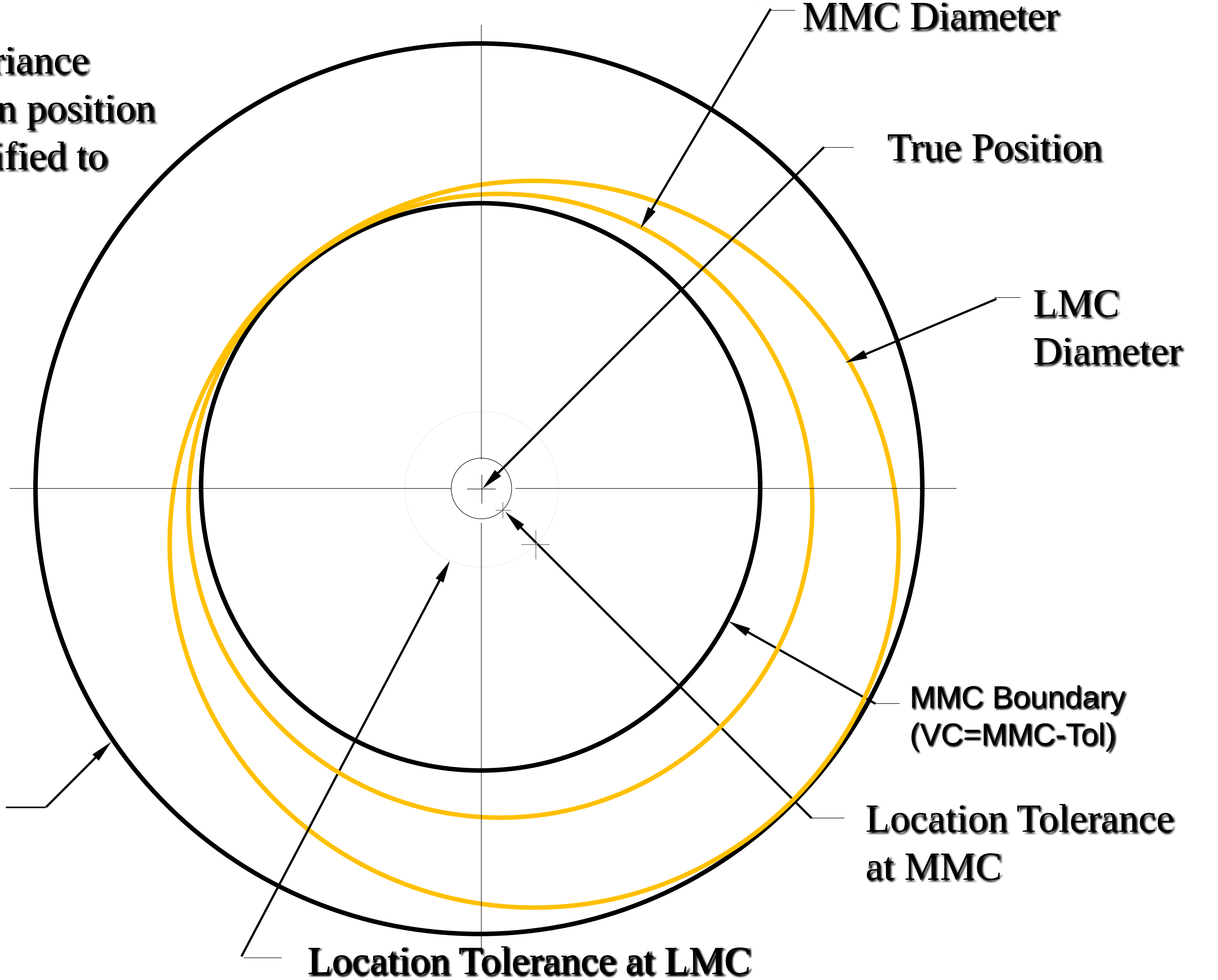
Tolerance of Position -- MMC

In this next series of slides, note that while the size and location of the actual hole may vary, the elements on the surface of the holes never violate their boundaries. This series will help you to understand how the hole size changes can affect the location of the center axis—and its orientation.



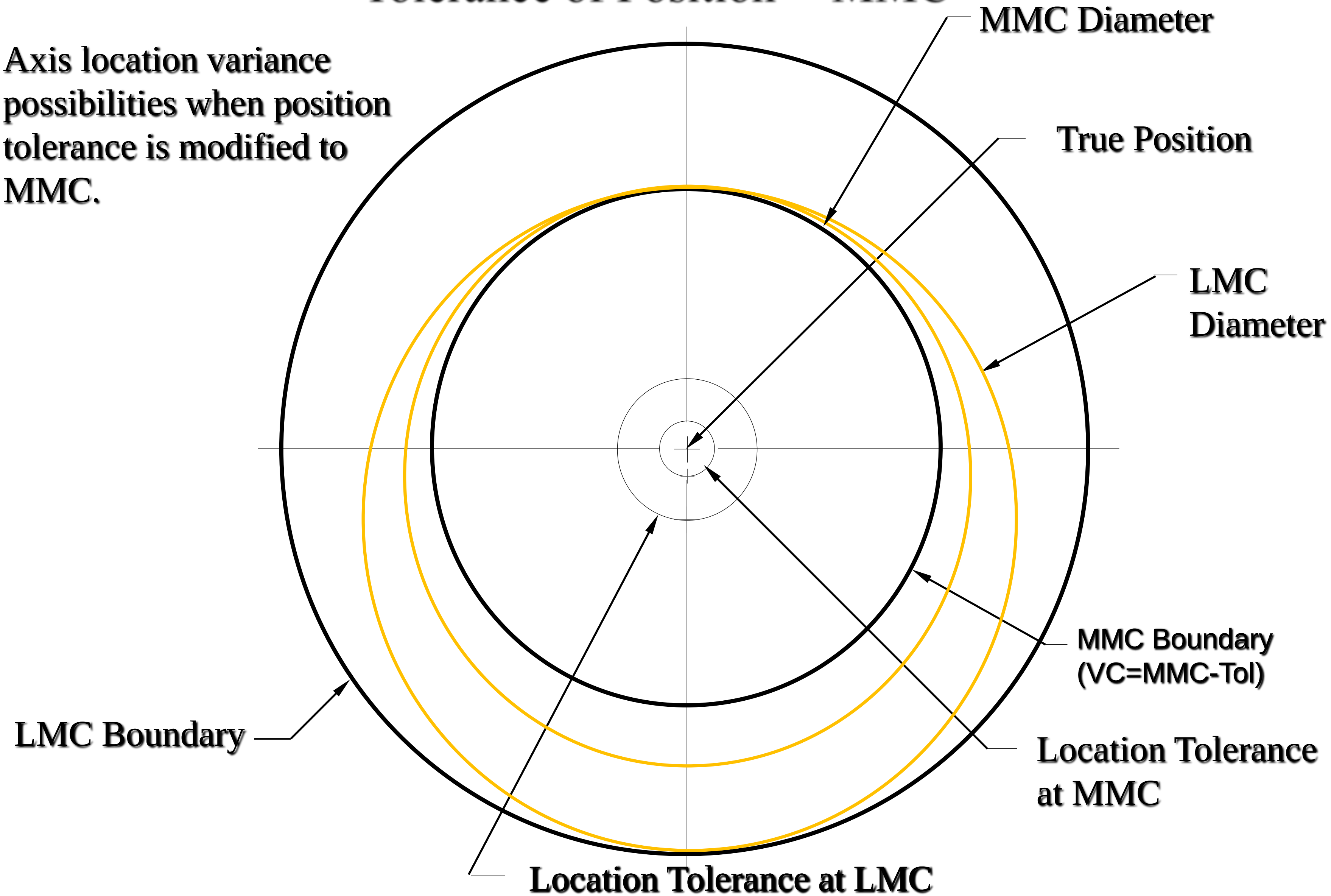
Tolerance of Position -- MMC

Axis location variance possibilities when position tolerance is modified to MMC.



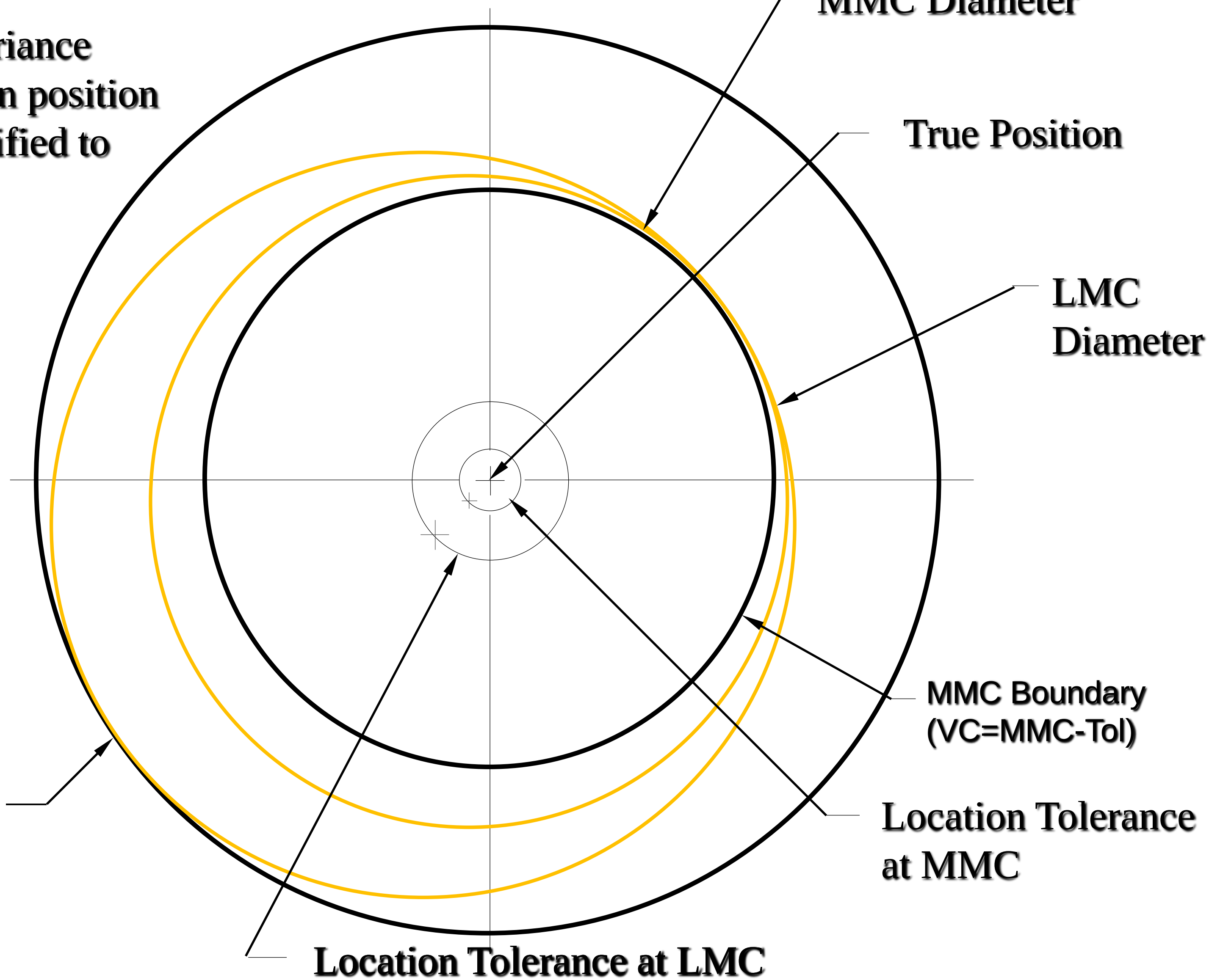
Tolerance of Position -- MMC

Axis location variance possibilities when position tolerance is modified to MMC.



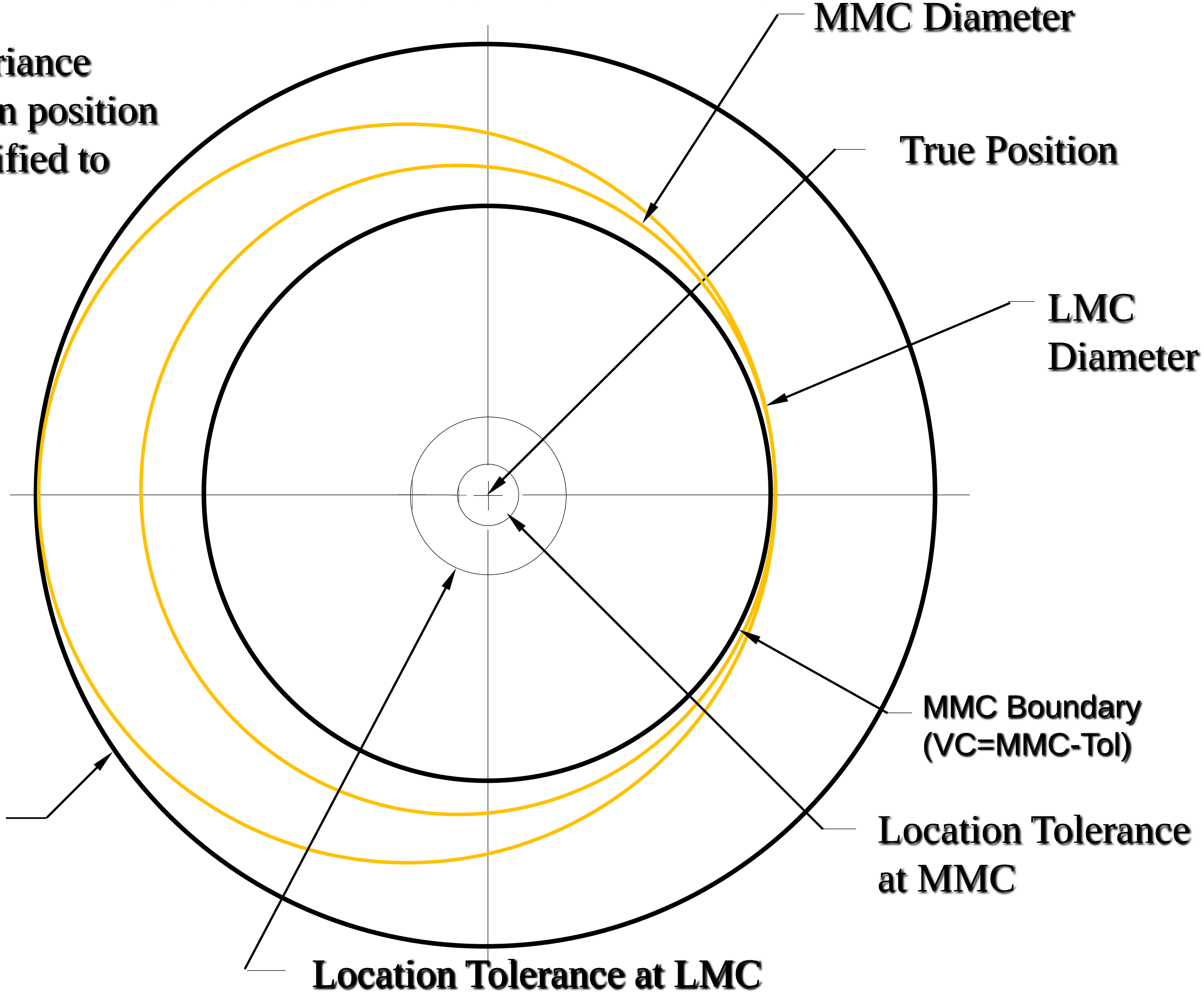
Tolerance of Position -- MMC

Axis location variance possibilities when position tolerance is modified to MMC.



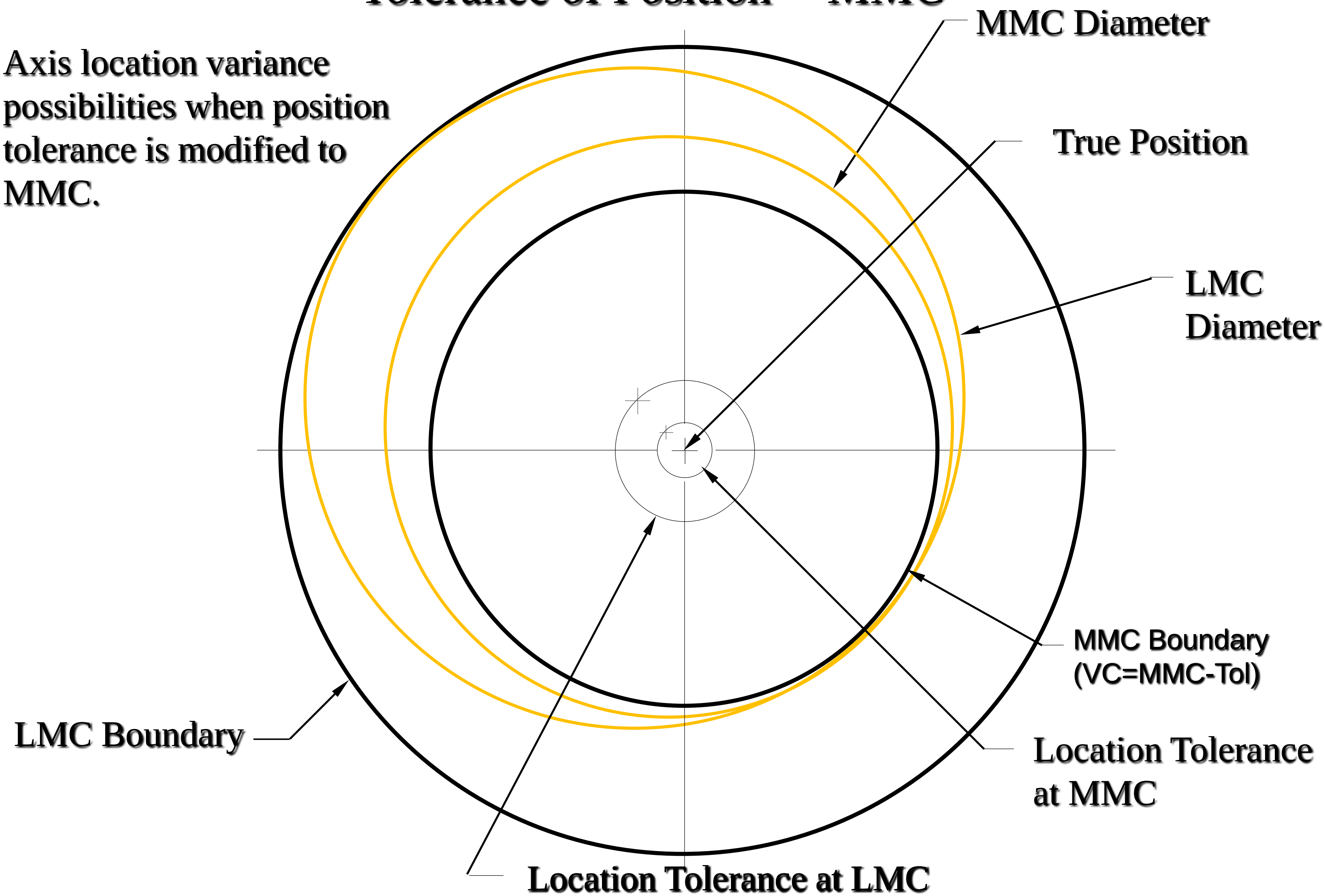
Tolerance of Position -- MMC

Axis location variance possibilities when position tolerance is modified to MMC.



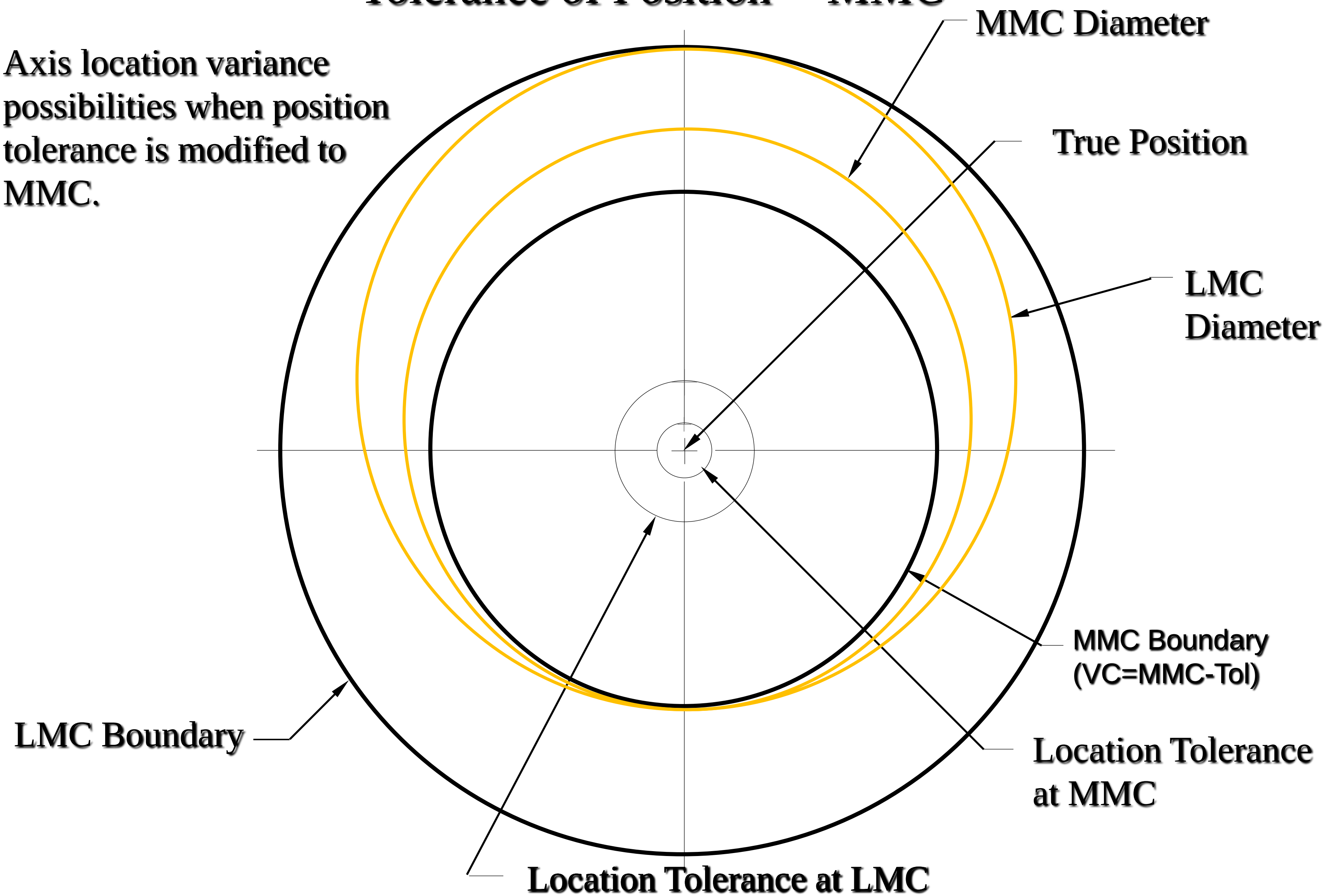
Tolerance of Position -- MMC

Axis location variance possibilities when position tolerance is modified to MMC.



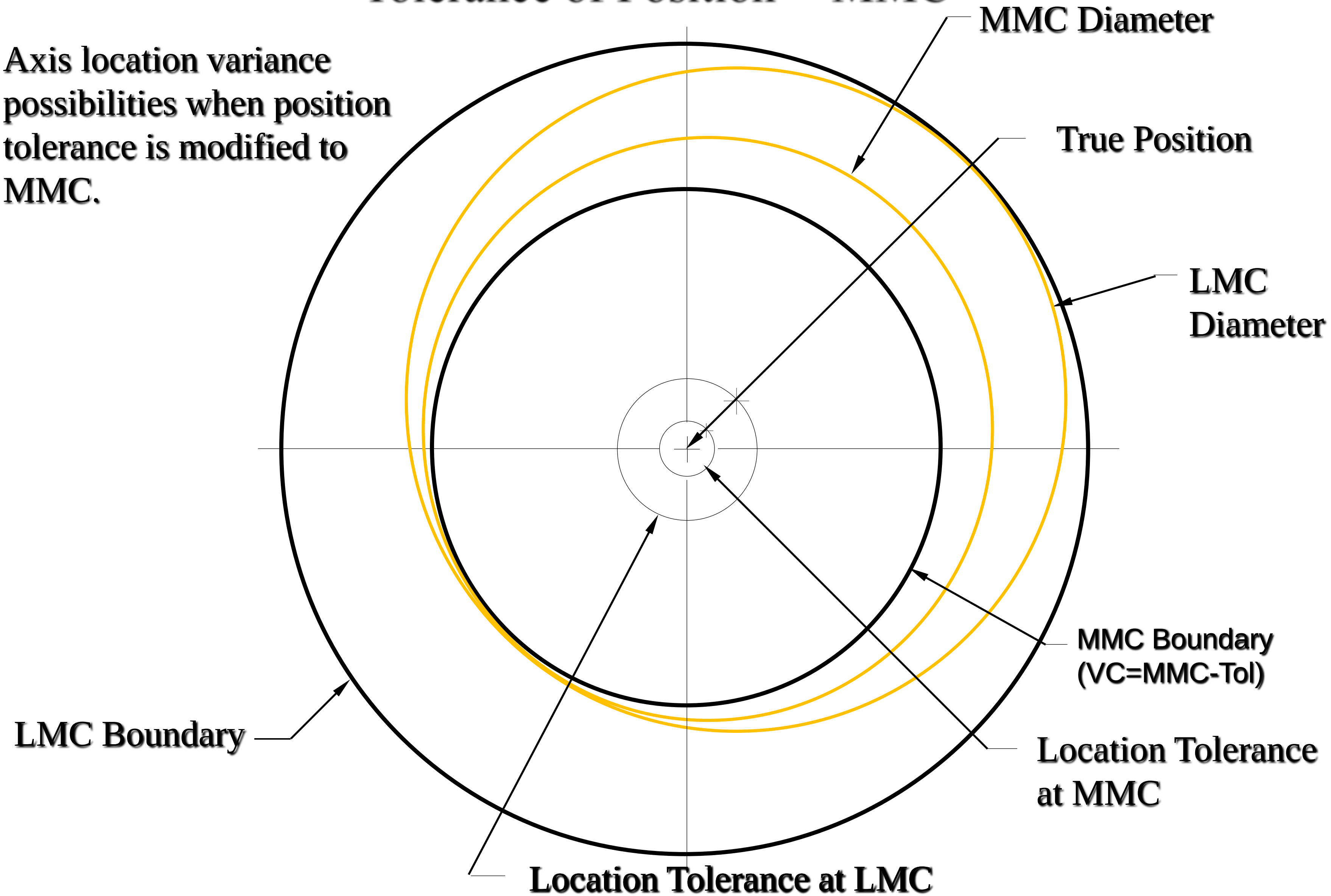
Tolerance of Position -- MMC

Axis location variance possibilities when position tolerance is modified to MMC.



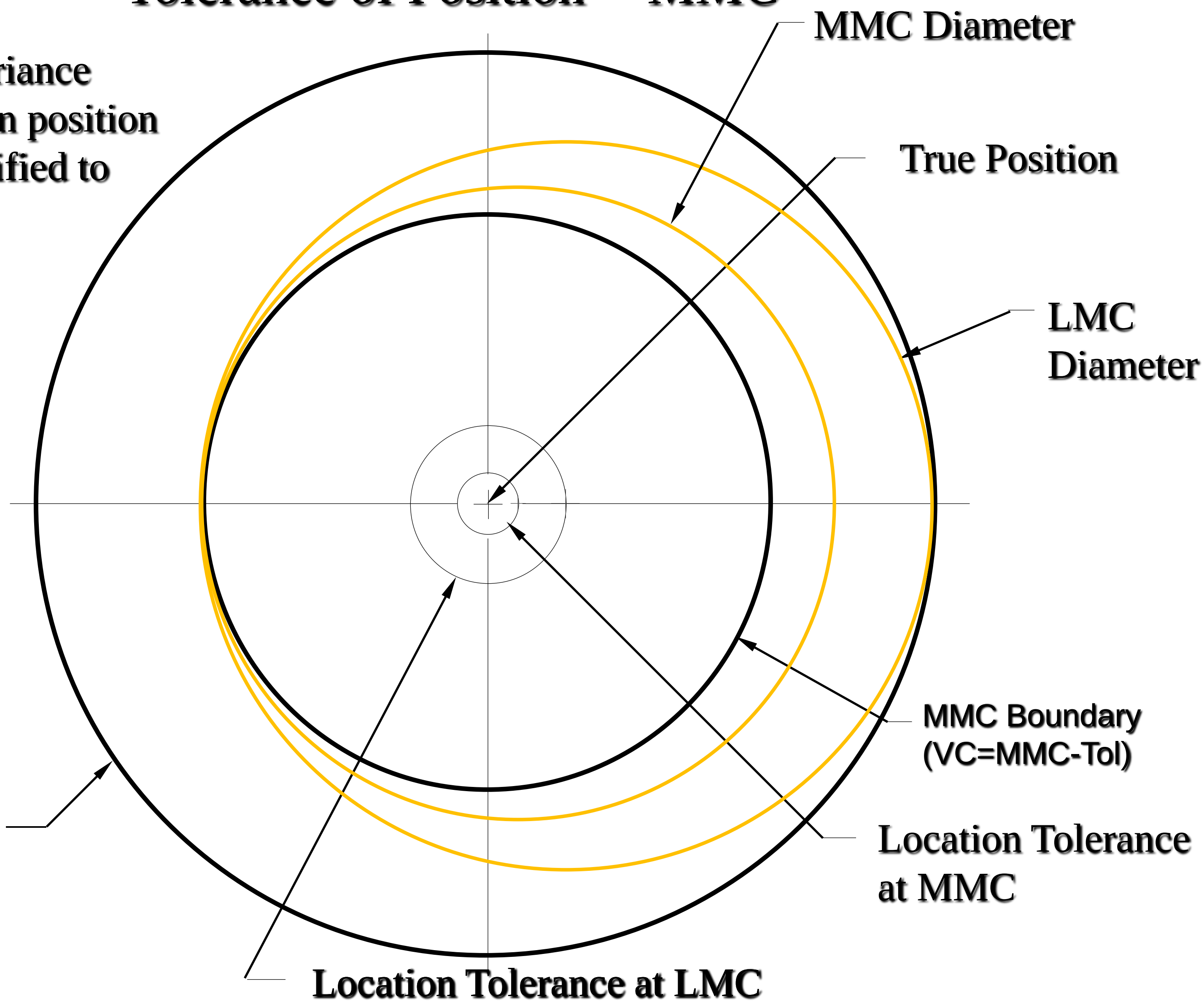
Tolerance of Position -- MMC

Axis location variance possibilities when position tolerance is modified to MMC.



Tolerance of Position -- MMC

Axis location variance possibilities when position tolerance is modified to MMC.



دانشگاه صنعتی شاهرود

Summary

- GD&T (geometric dimensioning and tolerancing) is an international design standard.
- Uses consistent approach and compact symbols to define and control the features of manufactured parts.
- Is derived from the two separate standards of ASME Y14.5M and ISO 1101.
- Technically, GD&T is a drafting standard.

دانشگاه صنعتی شاهرود

Summary

- Helps inspectors improve their methods by emphasizing fit, form and function.
- Compares the physical, imperfect features of a part to its perfect, imaginary form specified in the design drawing.
- Controls flatness, straightness, circularity, cylindricity, and four form tolerances that independently control a feature.
- Other tolerances, such as location, runout, and orientation must be referenced to another datum.

Summary

- The profile tolerances can define a feature independently.
- A related datum can further define the orientation and location.
- A series of internationally recognized symbols are organized into a feature control frame.
- The control frame specifies the type of geometric tolerance, the material condition modifier, and any datums that relate to the feature.