

HISTORY AND BACKGROUND

The concept of tolerancing is a relatively recent development; relative to the timeline of the Industrial Revolution. It has only been since the early 1900's that tolerances have been placed on drawings. The reader may note that this time frame corresponds roughly to the standardization of thread forms also. In the past there were no tolerances placed on drawings. Machinists were expected to build exact or "perfect" parts. If parts were not perfect (none were) and caused fitting or use difficulties, they were filed and hand fitted until they functioned as desired. Workers were considered true craftsmen and the knowledge of making parts fit was a father son tradition handed down through the generations. At the time of all this fitting and filing, measurement instruments did not exist that had accuracy enough to detect small variations of size in any case. As a result the lack of tolerancing on part prints was not deemed a hindrance in making functional parts.

As time and methods of manufacturing progressed it was found that parts did not need to be exact in size to work. In fact some variation could be tolerated and still provide functional parts for assemblies. For example, an oversize pin could still be placed in an oversize hole and still allow parts to mate. Acknowledgement of this reality is how the system of coordinate (conventional) tolerancing found its way onto part prints.

This conventional tolerancing system worked effectively until about the late 1930's. In the hurried build up of military arsenals before the outbreak of World War II (1939 in Europe), the royal torpedo factory in Scotland began to experience critical delays in manufacturing. Parts made outside tolerances specified on the print could still be made to work if selective mating of parts was manually undertaken. While preventing the scraping of some parts it was a laborious and cumbersome process that proved faster than remaking parts. Determined that a better system was possible, a factory worker named Stanly Parker came up with the idea of a round positional tolerancing system to replace the square or coordinate system in use on part drawings. This system would locate features with round tolerance zones based on part geometry that serve a discrete function in the assembly. This new system of tolerancing rejected truly bad parts but accepted parts that could function based on their geometry. Although not initially named such Geometric Tolerancing was born.

The idea caught on, gained acceptance in the United States in the 1940's (WWII) and was eventually incorporated into a military standard as MIL-STD-8 for all part prints. This system underwent revisions 8A, 8B, and 8C of this standard. The first unified American Standard ASA Y14.5 was adopted in 1966 and became accepted for use in military contracting.

In 1973, the American National Standards Institute ANSI Y14.5 standard was released and expanded the geometric system farther and converted it all to Symbology. Symbols replaced the ambiguity of words and provided a universal language understood by any



nation sharing drawings. In the global economy of manufacturing present today, this universal understanding of tolerancing for parts can not be underestimated in value.

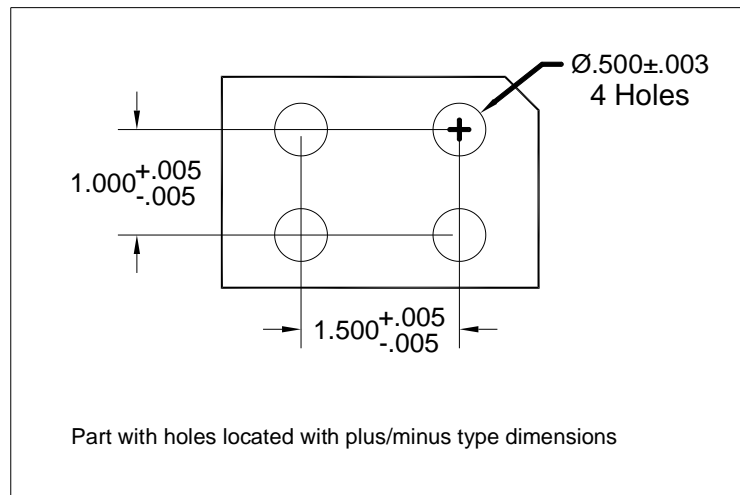


Plus / Minus VS. Geometric Tolerancing

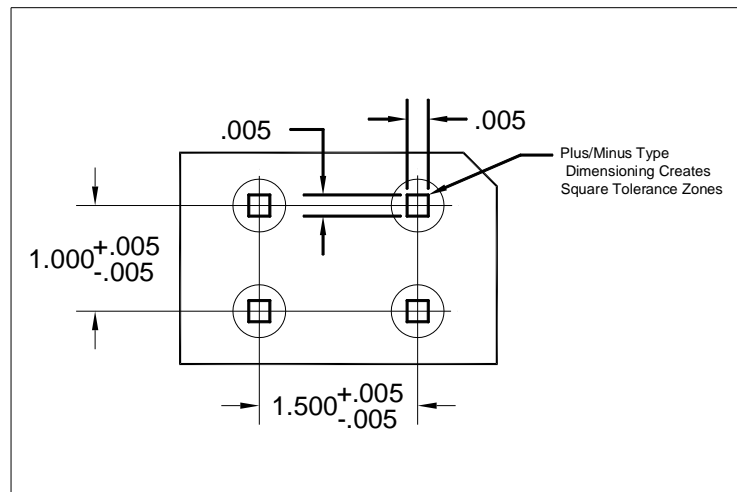
Why change from plus/minus Tolerancing to geometric Tolerancing?

There are many problems with the old system; lack of specified datums, lack of maximum material concept, non uniform interpretation, etc. But one of the most apparent and easiest to see examples is in the manner in which we locate holes.

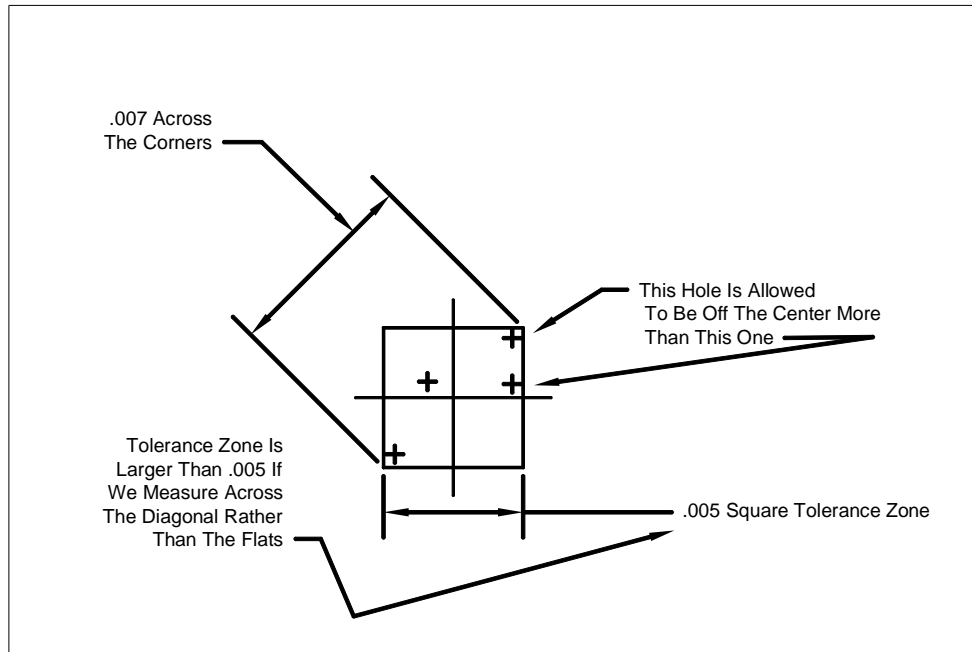
At present, in the coordinate system, we locate holes with dimensions that have plus and minus tolerance, in a manor shown below. To keep this example simple consider the distance between holes and not consider the distance to the edges at this time.



This method of tolerancing allows the axis of the hole to move plus something and minus something in the X and Y direction. This creates a square tolerance zone in which the holes (or their center axis) must lie. Observe the example below.



Notice, if the holes are located with a plus/minus square zone this allows manufacturing to place the hole centers anywhere inside this square zone. If the axes of the holes can fall anywhere inside the square zone it definitely would allow their axes to fall in the corner of the square zone. This in effect allows the holes to be off location more in a diagonal direction than across the flats as illustrated in the drawing below.



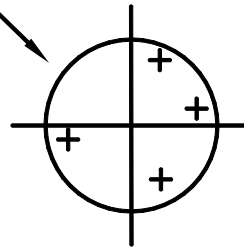
This is not logical. It is not desirable to allow the holes to be off location more in one direction than another. The hole location should be the same in any measured direction. Looking at the tolerancing method from a manufacturing standpoint it does not make sense either. When manufacturing locates the holes they do not use the tolerance. They will endeavor to put the holes in, exactly as the dimensions dictate. In other words, if the dimension allows plus/minus .005, manufacturing does not say; "We have a total of .010. Let us be off .002." They put the holes in as precise a manor as possible, exactly as the dimension requires. Then in inspection or quality processing a determination can be made to what extent the holes are out of location.

ADVANTAGES OF GEOMETRIC TOLERANCING

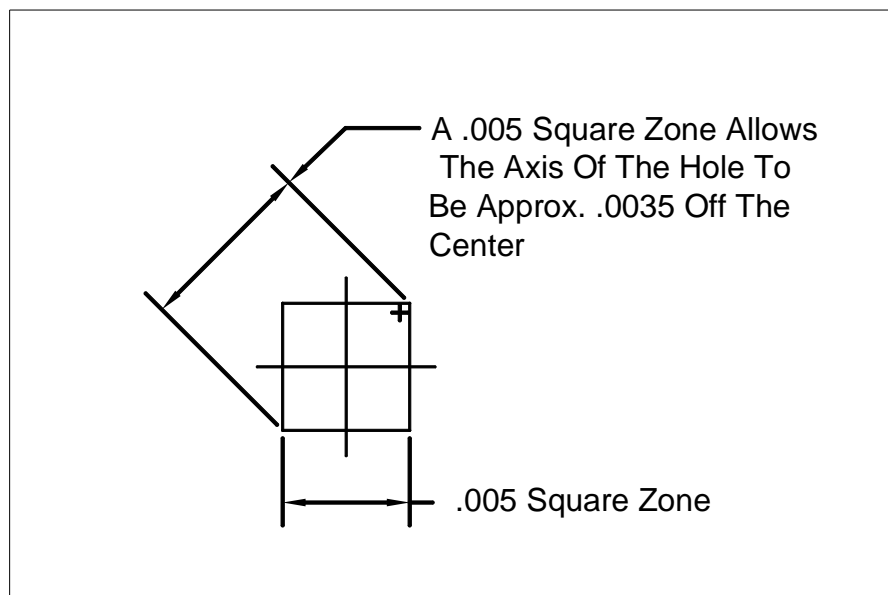
Locating holes (or any feature for that matter) should be similar to shooting at a bull's-eye at a shooting range. Aim for the exact center and afterwards check to see how much the bullet hole is off center of the intended position. The tolerance zones should be round and not square. If round tolerance zones are used the holes will be off location the same in any direction that is measured. This is the desired result. See the figure on page 3. The old system of plus/minus gave square zones and they are neither useful nor logical. In the past these was close, but not close enough for today's design requirements.



Round Tolerance Zones Do Not Allow The Axis Of The Holes To Be Off The Center More In One Direction Than Another

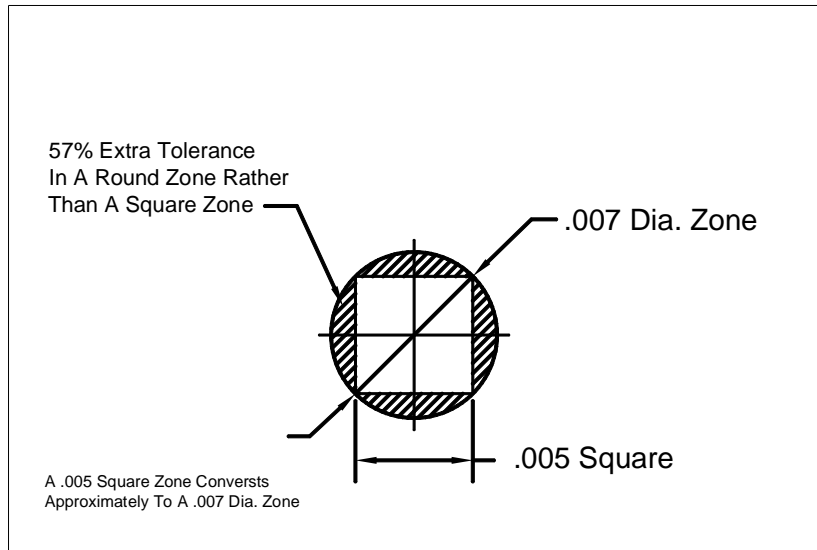


Another advantage of round tolerance zones over square zones is the opportunity to receive additional tolerance in manufacturing parts. The reasoning is as follows; in the plus/minus system the designer allows the location of the holes anywhere in the square zone. If the location of the hole axis is in the corner of a .005 square zone it is possible that the hole could be .0035 from the basic center if measured in a diagonal direction as shown in the figure below.



If the designer made calculations on two mating parts and determined that the parts would work if the holes were off their basic location by .0035 in the diagonal direction, why not let the holes be off location in any direction by .0035? A .005 square zone converts to an approximate .007 diameter zone. Notice that by switching from a .005 square zone to a .007 diameter zone, it is possible to have an increase of 57% additional tolerance while still assuring the assembly of parts.



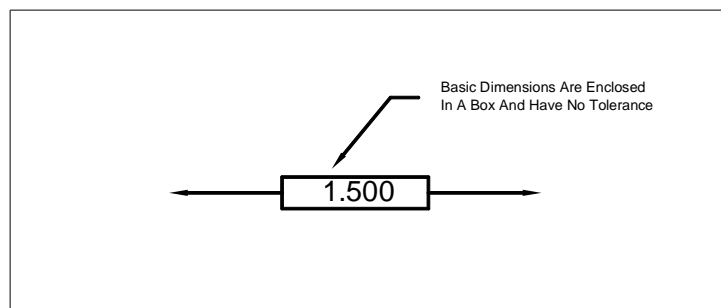


At this point it should be apparent the benefit in using round geometric tolerance over the square plus/minus system. It is more correct and logical and has an added benefit of allowing more tolerance but ensuring functional parts (the real objective).

HOW THE SYSTEM WORKS

So what does the use of geometric tolerancing and positional tolerancing actually look like? Consider a rectangular plate on which we wish to locate 4 holes to each other. On this plate there must be dimensions between holes. If tolerances are placed on the dimensions we arrive back at the square zone environment again. To define the part with round positional zones the hole location must be described in an entirely different manner.

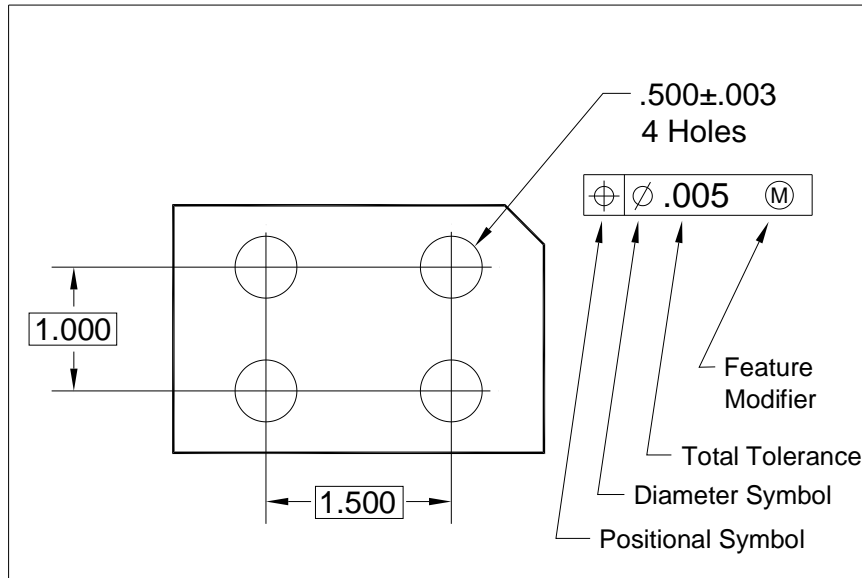
The dimensions must be described as exact or basic. They will have no tolerance; and to ensure that title block tolerance is not applied, the dimension will be enclosed in a box. This means that this dimension is basic and has no tolerance.



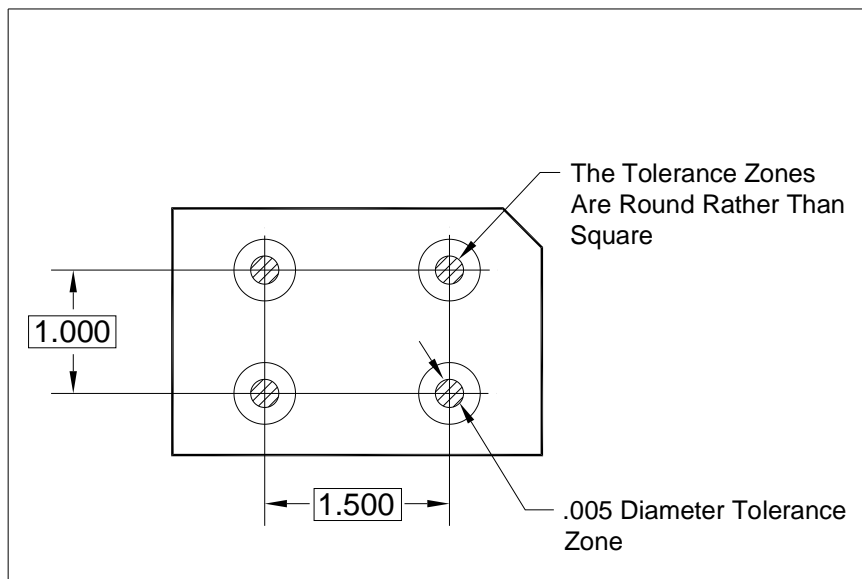
Of course it is not possible to build anything “perfect”, with no tolerance. There must be tolerance somewhere in the manufacturing process. Instead of placing tolerance on the dimensions, the tolerance is placed in a frame and placed under the holes that are to be located. Pause and think about it logically, the dimensions should not have tolerance. The basic dimensions describe the exact location where the holes are to be located. The frame under the holes contains the tolerance that the holes can be off the basic location.



The complete name for this frame is, feature control frame. In Geometric tolerancing holes, slots, tabs, surfaces, etc. are referred to as features. Hence the name feature control frame. It controls features and will give instructions for any associated feature.



In the first compartment of the feature control frame, a geometric symbol is placed. If the object were to locate the holes, the first entry would be the position symbol. In the next compartment the diameter symbol is placed indicating the tolerance zone will be a diameter. This is followed by the total tolerance.



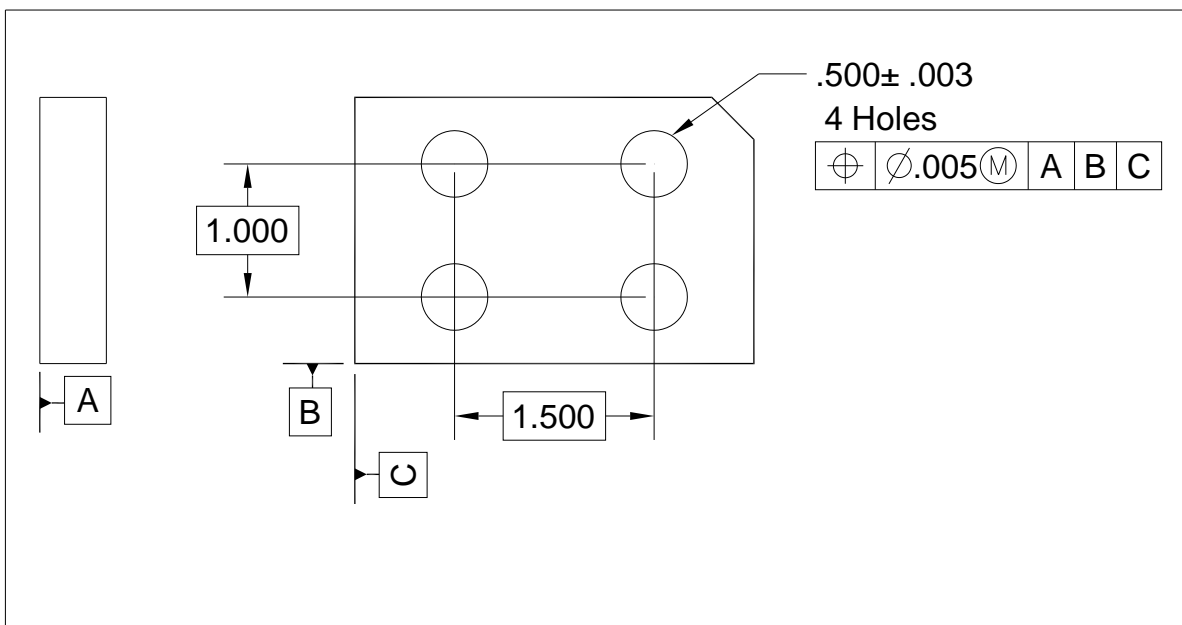
The tolerance zones are round rather than square. The axis of the holes must fall inside these round zones. Rather than stating the holes are plus/minus it can now be said the hole centers are located in a diameter of .005. These holes can only be off the basic location .0025 radiuses in any direction or a total of .005 on a diameter.



In the geometric system, there is also the concept of datums. In the past if it was desired to locate holes from edges, then the dimension lines were merely attached to the edges. This meant that, this is where the dimensions were to originate and all measurement was to start from this point. These edges were called implied datums. Without specifically noting where the measurements were to originate from, it was assumed that the user of the drawing “knew” these were the reference edges.

In geometric tolerancing implied datums are not used. Datums that are to be used are specified and placed in order; in the next compartment of the feature control frame. The reasoning for not using implied datums, beyond the obvious confusion or misinterpretation, is because the order of datums has great importance and this provides a method of hierarchy.

This is basically how the system works. The dimensions are basic and have no tolerance. Rather, the tolerance is put in a feature control frame along with any other requirements and is placed under or associated with the feature that is being located or controlled.



The requirements for the holes are as follows; the sizes of the holes are $\Phi .005 \pm .003$. The location of the holes are about the basic location within a $.005$ diameter tolerance zone at maximum material condition in relation to the datums A, B, C.

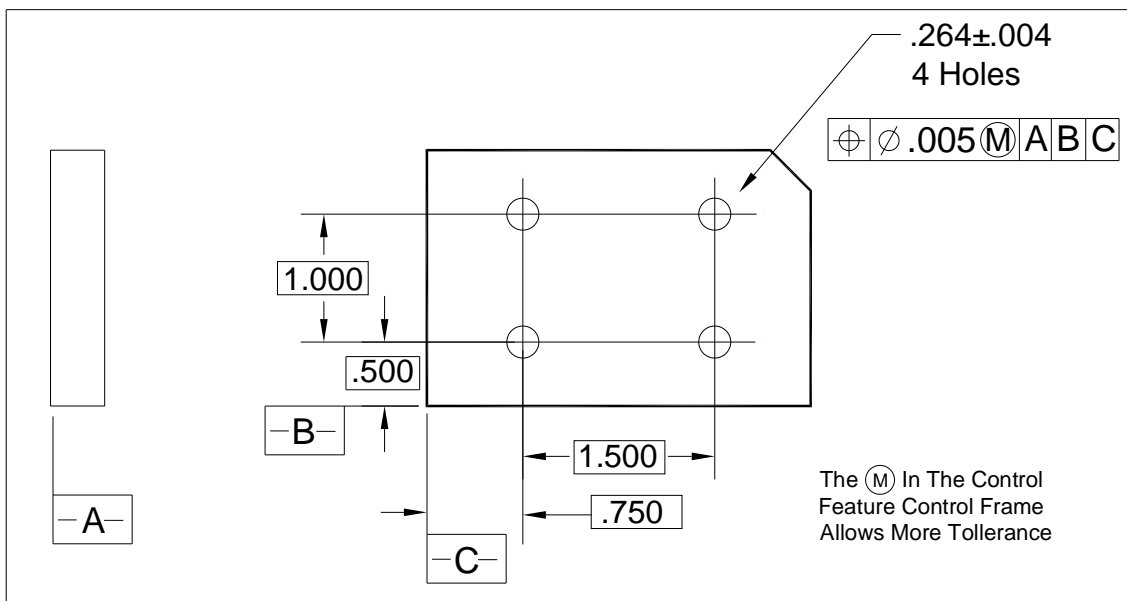


MAXIMUM MATERIAL CONDITION CONCEPT

Abbreviation = MMC

Symbol = \textcircled{M}

Previously the advantage of positional tolerance zones over plus/minus was illustrated by pointing out the shortcomings of coordinate tolerancing. The ability of increasing tolerance in some cases up to 57% by switching from a square tolerance zone to a round zone and produce acceptable parts is at the least a competitive advantage. There is another method to increase the available tolerance through the maximum material concept \textcircled{M} . Examine the graphic below. In the feature control frame it is stated that the holes have a diameter (\varnothing) tolerance zone of $.005\textcircled{M}$. A logical question would be "how do you get more tolerance"? It is obviously defined in the feature control frame that the tolerance is $\varnothing .005$. Follow this explanation for the answer.



Consider where the $\varnothing .005$ tolerance came from in the first place. Consider the question; why was the tolerance not specified $\varnothing .007$ or $\varnothing .010$? The tolerance in the feature control frame is not an arbitrary number. It is usually arrived at by formulas called the fixed and floating formulas. These will be described in detail later in this paper. Consider the following abbreviated sample.

In the figure above the $\varnothing .264 \pm .004$ holes are to match up with $\frac{1}{4}$ - 20 tapped holes in a mating part. This falls into the fixed fastener formula category. The location of the features on both parts is defined as basic. The designer/engineer then determines the largest the screw could be (major dia. = $\varnothing .250$), and then the smallest size clearance holes could be, which in this example is $\varnothing .260$. The difference between these two numbers, $\varnothing .010$ is the total allowable tolerance for location. This $\varnothing .010$ may be split and $.005$ is given for position of the tapped holes and the remaining $.005$ is given to the



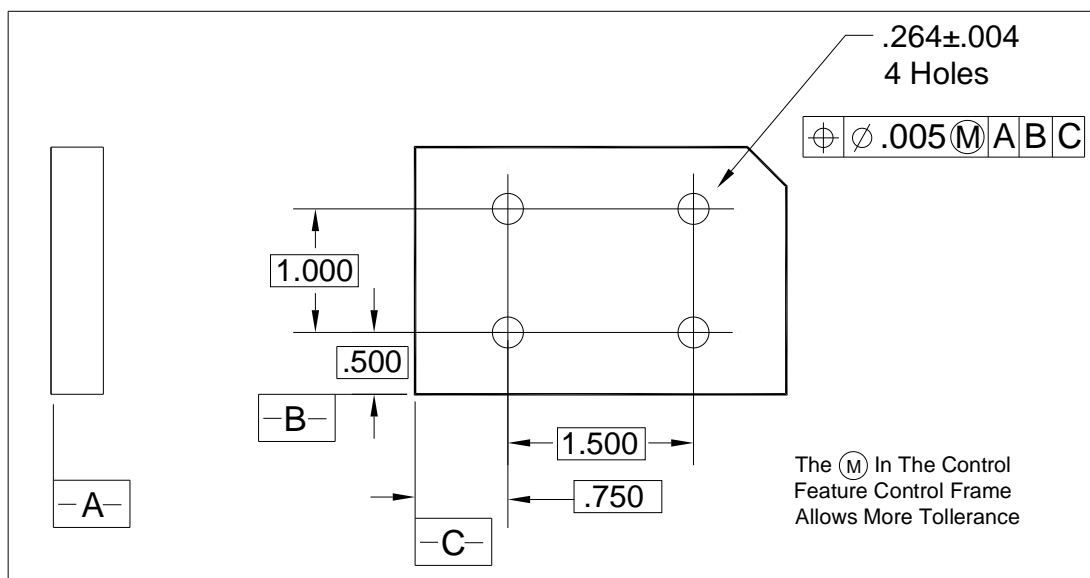
clearance holes for position. This is how the $\varnothing .005$ -position tolerance is arrived at, in the examples feature control frame.

Notice that the $\varnothing .005$ location is arrived at by determining the worst condition that can occur – the largest screw size and the smallest size clearance hole. If the hole is produced at the low side of $\varnothing .260$ it can be off location in a zone of $\varnothing .005$ and still assemble properly with the mating part. But what if the hole size is produced at $.261$? It seems reasonable if the hole is produced larger by one thousandth of an inch than the smallest size it can be off location $\varnothing .001$ and still work fine. In fact the larger the hole is produced the more location tolerance it can have. If the hole is produced at its largest size of $\varnothing .268$, it will be greater than the smallest size by $\varnothing .008$. Thus the size of the positional tolerance zone can increase by $\varnothing .008$. The original $\varnothing .005$ for location plus the $\varnothing .008$ that the hole size grew larger, gives a total location tolerance of $\varnothing .013$.

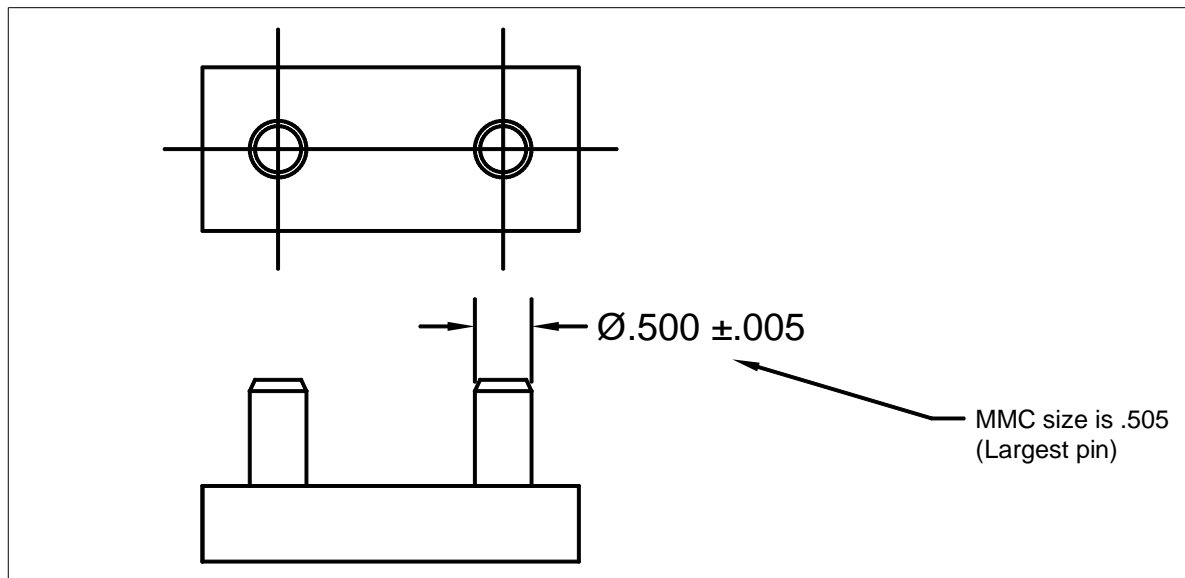
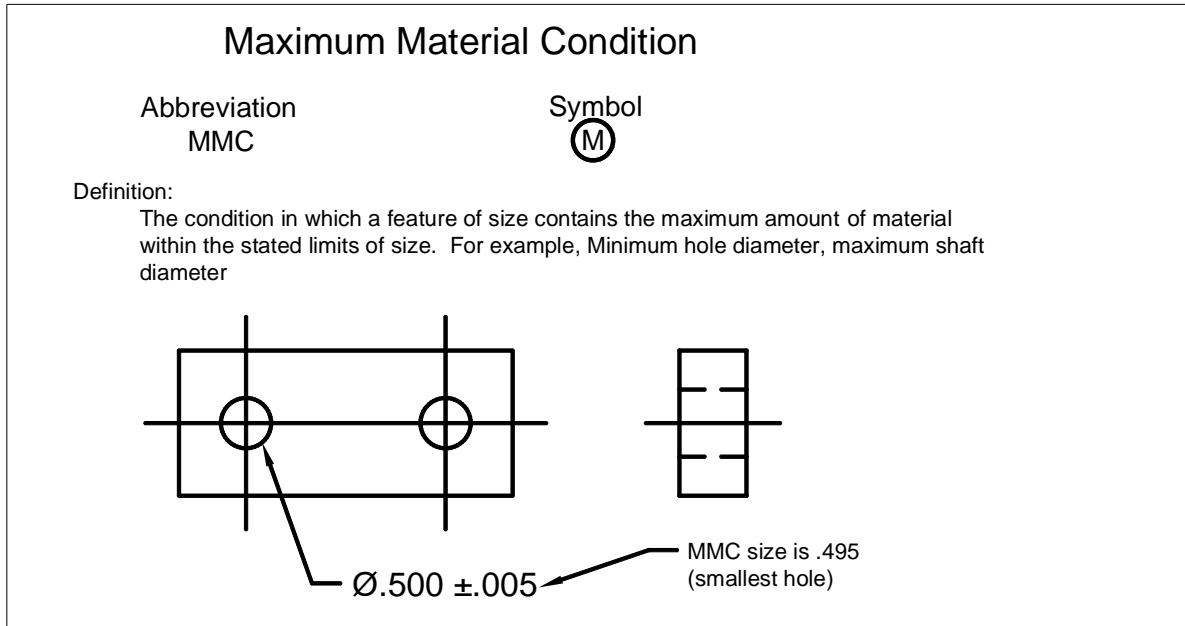
If the object is to clear a screw in a mating part and a small size clearance hole is put in place, the location of that hole must be quite close to the basic dimension. If the size of that clearance hole is made larger, then the hole location can be off more and still clear the mating part. How much more it can be off location depends on how big the hole gets. One unit larger in size—one more unit for location; two units larger—two more for location, and so on.

It is a one for one ratio, never half. The hole gets one unit larger on the diameter so the tolerance zone gets one unit larger on the diameter. Each hole works independently of each other. If only one hole gets larger, then it is the only one that has more location tolerance.

This is the essence of what is called the maximum material concept. To put this condition into effect, put an M in a circle (M) and place it in the feature control frame behind the tolerance as shown in the graphic below. This feature control frame is read; these features (the $\varnothing .260 / .268$ holes) must be positioned in a $\varnothing .005$ tolerance zone at maximum material condition (MMC) in relation to the datum's A, B and C.



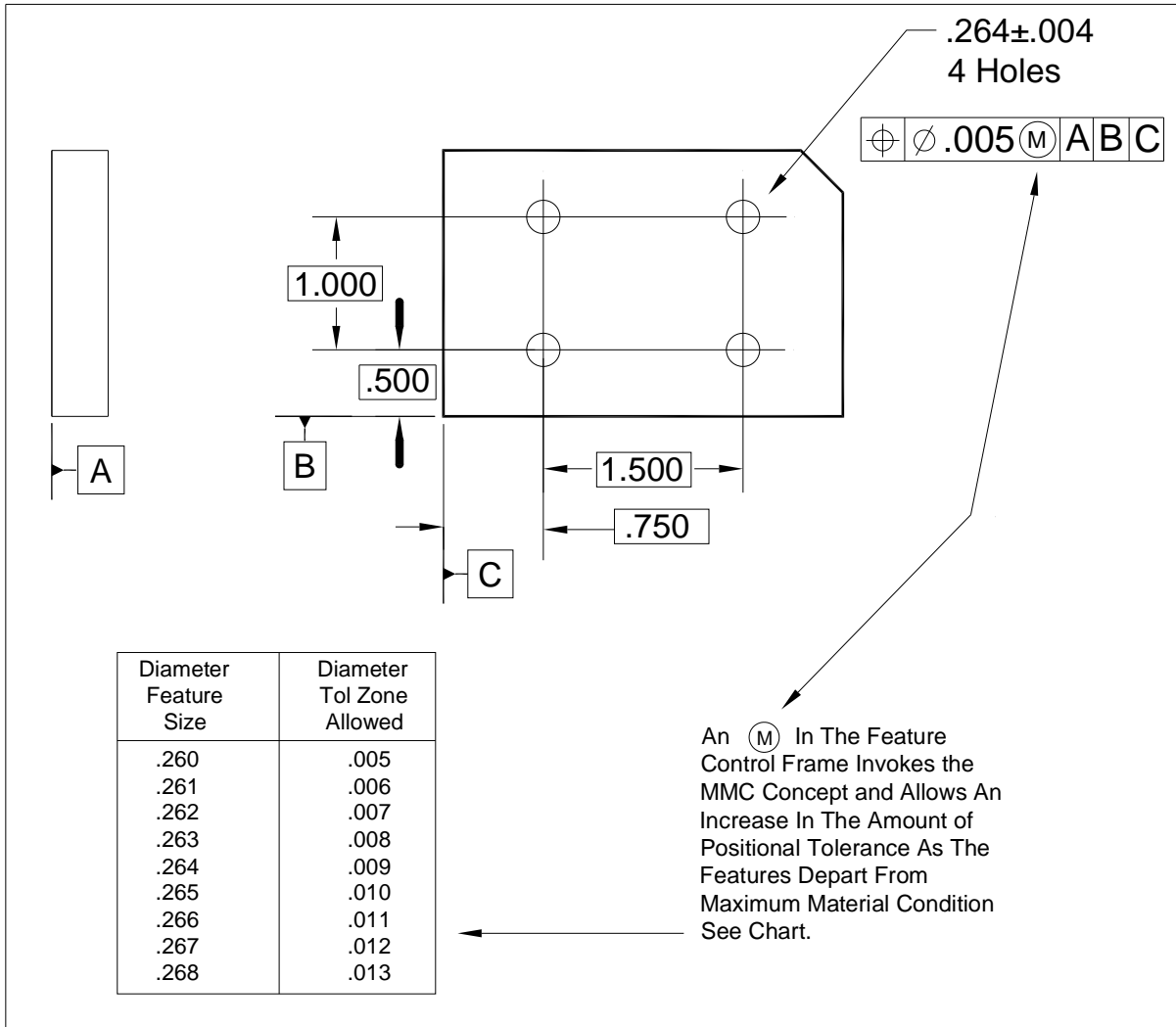
The holes must be positioned at their maximum material condition (MMC) within $\varnothing .005$; what is the maximum material condition (MMC) of these holes? It is the condition where the features contain the most material. Example; (smallest holes or largest pin) see the example below.



The \textcircled{M} modifier in the feature control frame then states that the features must be positioned within $\varnothing .005$ at their maximum material condition (MMC). The maximum material condition (MMC) of the clearance holes is $\varnothing .260$. Thus when these holes are produced at $\varnothing .260$ they have only $\varnothing .005$ for location. If the holes depart from the $\varnothing .260$ (MMC) size, they can have additional location tolerance equal to the amount the hole



size departed from MMC. This condition is illustrated in the example chart in the next graphic.



As can be seen the MMC concept can give more tolerance while still ensuring a functional fit between parts. This concept is not available for use in the plus/minus system. Of course many technicians in manufacturing knew the concept worked despite the lack of a formal name. If drilled holes were a little out of location and did not match the mating part, just opening up the hole a little and they would work. The holes could not be opened past their limits of size, otherwise they would be rejected because they were too large.

With geometric tolerancing the designer/engineer now has a method to specify this condition on the drawing. The MMC condition is usually used when features are designed from a line to a loose condition. Most of the parts worked with will fall into this category. The MMC condition is used about 90% of the time, because most



parts are loose fitting, depending on the industry. The maximum material condition also allows functional type gauging.

MMC can be used with location tolerances, form tolerances, orientation tolerances and datums. The only restriction is that the feature or datums modified must have size. In other words, it is not possible to modify a surface because the surface has no size. “At its maximum material condition” would have no meaning because a surface has no maximum material condition; it is only a surface.

There are certain conditions that allow modifiers to be either specified or implied in the feature control frame. More detailed information on these conditions will be covered in general rules number 2 and 3.

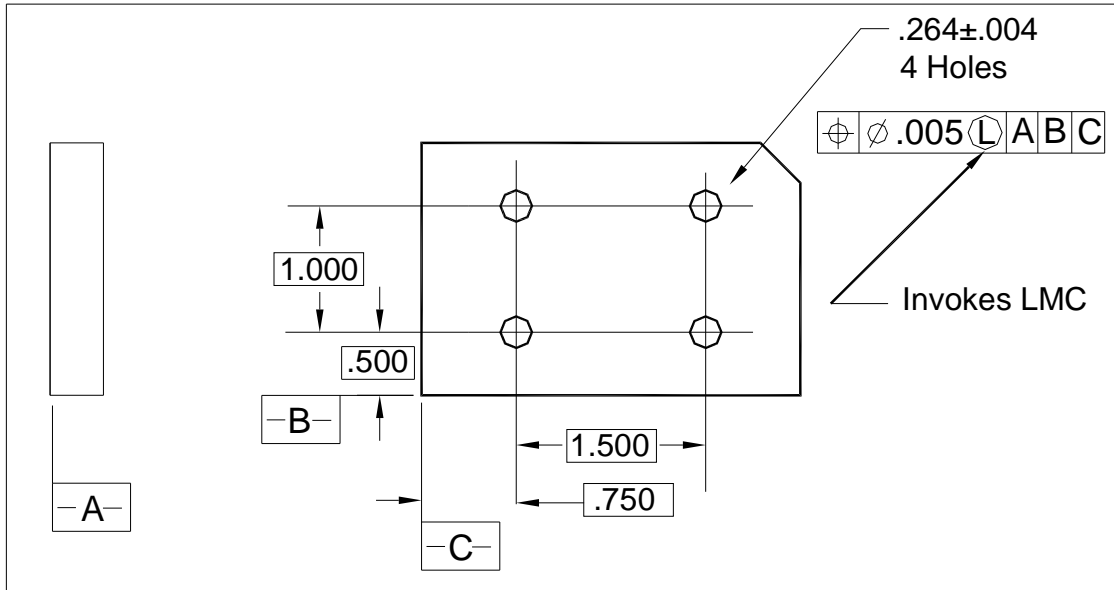


LEAST MATERIAL CONDITION CONCEPT

Abbreviation = LMC

Symbol = \textcircled{L}

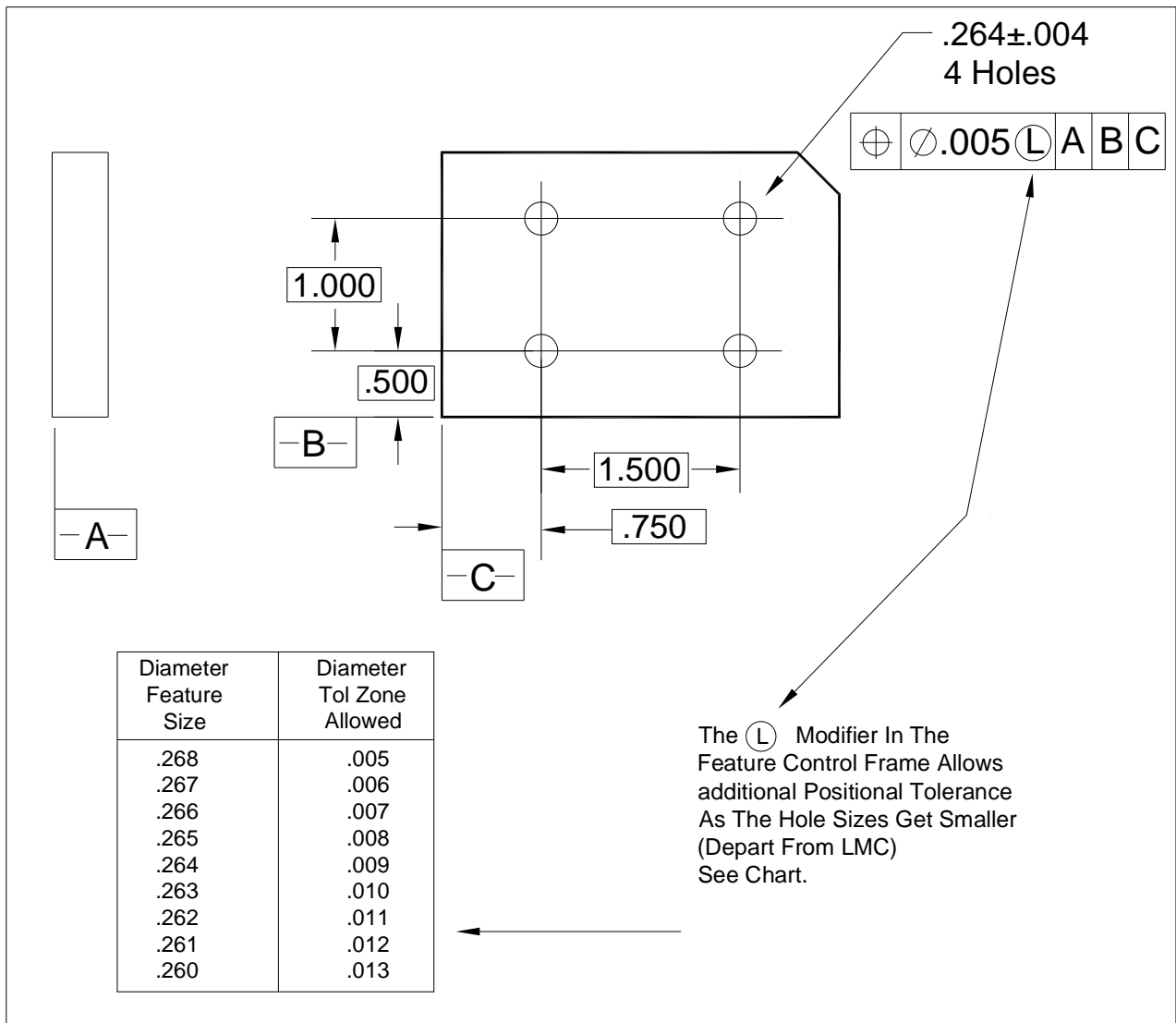
The least material condition concept works just like the maximum material concept, except in reverse. In the graphic below is an example of 4 holes located in a plate at their least material condition (LMC). Notice the \textcircled{L} modifier in the feature control frame.



The feature control frame would read; these features (the .260/.268 holes) must be positioned within a .005 tolerance zone at their least material condition (LMC) in relationship to datum's A, B, and C.

The \textcircled{L} modifier in the feature control frame then states that the features must be positioned within $\text{Ø}.005$ at their least material condition (LMC). The least material condition of the clearance holes is $\text{Ø}.268$. Thus when these holes are produced at $\text{Ø}.268$ they have only $\text{Ø}.005$ for location. If the holes depart from the $\text{Ø}.268$ (LMC) size they can have additional location tolerance equal to the amount the hole size has departed from LMC. This condition is illustrated in the next graphic.

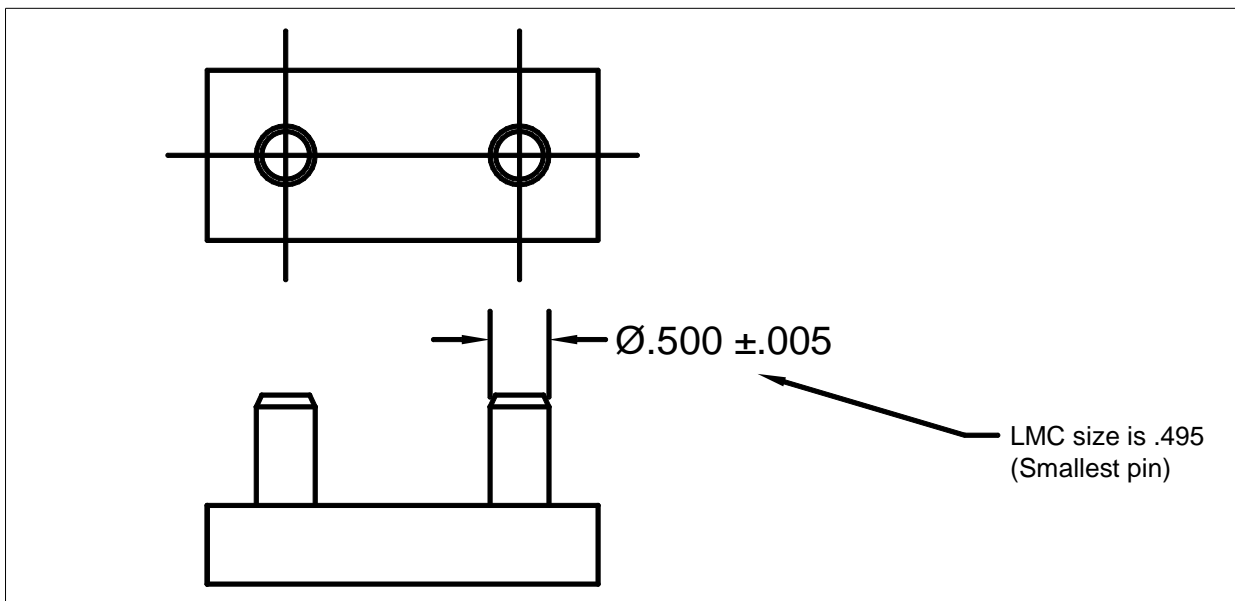
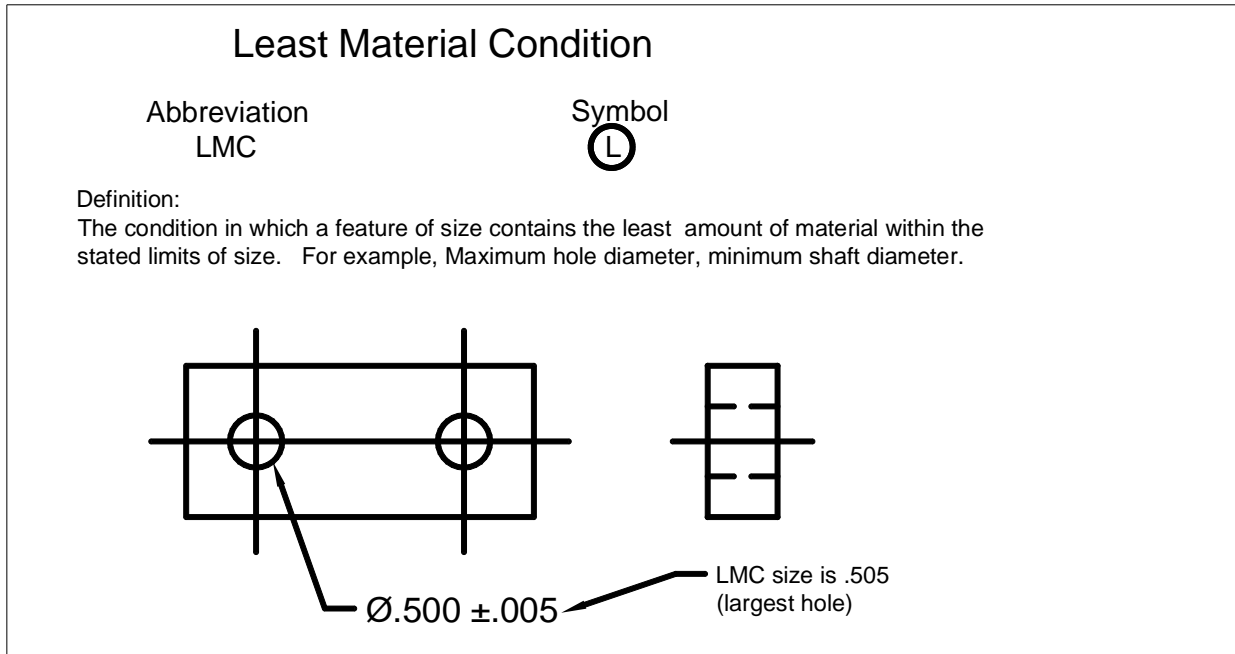




As shown above, with LMC, as the holes get smaller they have more tolerance for location. This is the opposite from MMC. With MMC as the holes get larger they have more location tolerance.



The features must be positioned within $\varnothing.005$ at their least material condition (LMC). What is the least material condition of the features? It is the condition that contains the least material. Or, it may be helpful to think of it in terms of the condition where the features weight the least. An example, the largest hole or smallest pin. Below is a definition in graphic form.



Without a doubt, the MMC is the most common type of application. LMC is used in Unique situations where control of wall thickness, requirements on castings, or gauging and tooling. There are more applications for least material conditions, but it is wise to fully understand the full scope of geometric tolerancing before attempting the unique situations. For most engineering applications the LMC is used about 5% of the time. It is usually difficult to verify this requirement with functional gauging. LMC applications can sometimes be verified with foot gages or wall thickness gages.

LMC can be used with location tolerances, orientation tolerances, form tolerances, and datums. The only restriction is that, just as in MMC, the feature or the datums that are modified must have a size. In other words, a surface cannot be modified because a surface has no size. "At its least material condition" has no meaning because a surface has no least material condition, it is only a surface.

There are certain conditions, which allow modifiers to be either specified or implied in the feature control frame. More detailed information on these conditions will be covered in general rules 2 and 3.



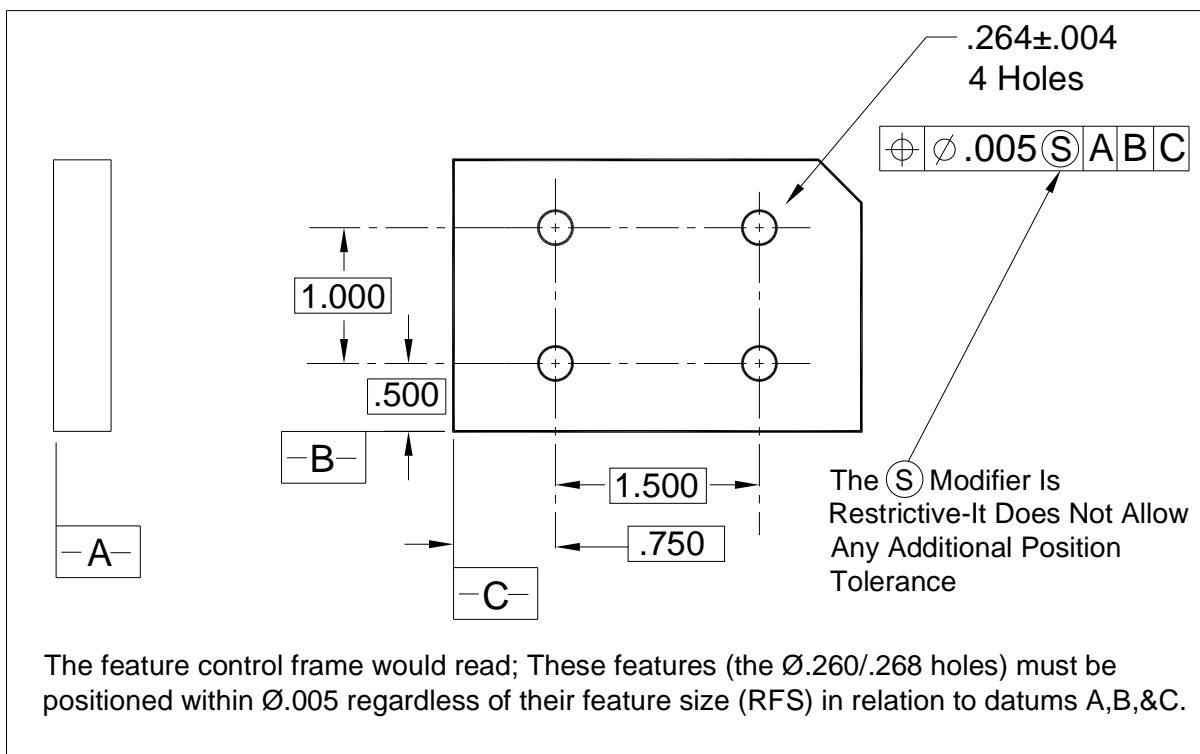
REGARDLESS OF FEATURE SIZE

Abbreviation = RFS

Symbol = (S)

Regardless of feature size is a difficult concept to explain unless the MMC and LMC concepts have been successfully explored and understood by the student. To begin this discourse on RFS the assumption will be made the student has a working understanding of MMC and LMC concepts.

It is known from the MMC and LMC concepts that size can be directly related to how much location, orientation, or form tolerance is available. With the (M) modifier more tolerance is allowed as the feature departs from MMC. With the (L) modifier more tolerance is allowed when the feature departs from LMC. There are some cases in design where regardless of the size of the feature, it is desired that the tolerance be only what is indicated in the feature control frame and no more. In these cases a (S) modifier is placed in the feature control frame as shown in the following graphic.



The features must be positioned within $\varnothing .005$ regardless of their feature size. This means that despite the size of the features being positioned they have $\varnothing .005$ tolerance and no more. If the size of the hole gets large or small the tolerance is the same $\varnothing .005$



The RFS condition is a very restrictive requirement. It is used when location or orientation is very important on the part such as a line to line to tight condition. Examples are dowel pin holes, locating gear centers, or press fits. Consider the example of dowel pin holes, if the size of the hole is at its large or small size, no additional tolerance would be available for location. Because it is a press fit the dowel will always find the center of the hole. In manufacture of most common products there is not an abundance of press or tight fits. In most cases, parts are loose fitting, depending of course on the nature of the business. In the average setting it would be unlikely to see RFS used more than 5% to 10% of the time for feature tolerance modifications.

Regardless of feature size is difficult, if not impossible, to verify with functional gauging techniques. Each part must usually be laid out on a surface plate and checked with dial indicator type equipment. Regardless of feature size is usually associated with tapered or expandable type pins in a gauging situation.

RFS can be used with location tolerances, orientation tolerances, form tolerances, and with datums. The only restriction is, just as in MMC and LMC; the features that are being modified must have size. In other words it is not possible to modify a surface because by definition a surface has no size. "Regardless of feature size" has no meaning when applied to a surface.

There are certain conditions, which allow modifiers to be either specified or implied in the feature control frame. For more detail information see rules # 2 and # 3.



SYMBOLS AND TERMS

Geometric Characteristics Symbols

These are the geometric characteristics and their symbols. The symbols are placed in the first compartment of the feature control frame. They will describe what is required of the feature that the frame is attached or associated with. Observe that the characteristics are categorized into five types of tolerances; Form, Profile, Orientation, Location, and Runout.

For Individual Features	Type of Tolerance	Characteristics	Symbol
For Individual Features	Form	Straightness	
		Flatness	
		Circularity (Roundness)	
		Cylindricity	
For Individual Or Related Features	Profile	Profile of a line	
		Profile of a Surface	
For Related Features	Orientation	Angularity	
		Perpendicularity	
		Parallelism	
	Location	Position	
		Concentricity	
	Runout	Circular Runout	
Total Runout			



FEATURE CONTROL FRAME

A feature control frame carries the instructions for the feature to which it is associated or attached. As its name implies it is called a feature control frame because it controls features. In the geometric system rather than calling a hole, slot or tab, they are lumped together in one category and simply called features. Thus the name feature control frame.

In the first compartment, there will always be one of the geometric characteristic symbols. These symbols describe what is required. It may be flatness or position, or other symbol depending on the requirement, but it will always be a characteristic symbol in the first compartment. Only one geometric symbol is allowed per feature control frame. If two requirements are needed for a feature, there must be two feature control frames.

The tolerance for the feature is in the second compartment. This tolerance is always a total tolerance and is never plus/minus. Therefore, you can be always assured that when you read a feature control frame the tolerance is always the total tolerance zone allowed for that feature.

It should by now be apparent that in geometrics, the tolerance is specified in zones. If the tolerance is a diameter zone as in position of a hole, the diameter symbol (\varnothing) would precede the tolerance stated. If no symbol is placed before the tolerance, the zone is implied as lines or planes of tolerances as in the position of a slot or the perpendicularity of a surface.

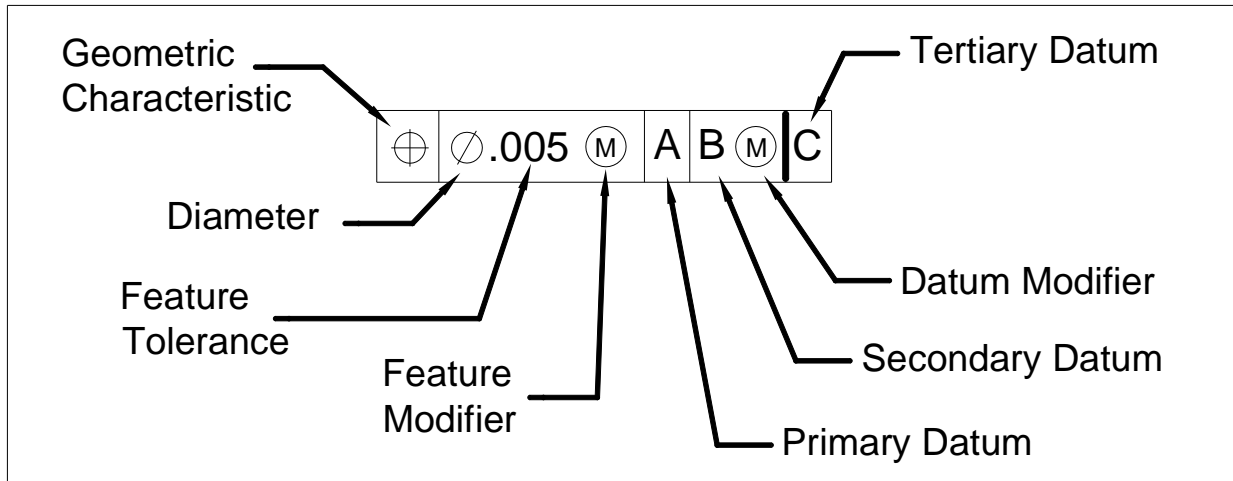
The feature modifier \textcircled{M} , \textcircled{S} , or \textcircled{L} is in the same compartment and follows the tolerance. These modifiers will tell if the MMC, LMC, or RFS concept is invoked. If the feature being controlled has no size, then the feature modifier is not applicable. For more information or a refresher on this subject refer back to the information provided earlier in this paper on MMC, LMC, or RFS.

Datums are in the next compartments if they are applicable. For example, in the case of flatness, a datum is not applicable. In the case of position, a datum or datums are usually required.



FEATURE CONTROL FRAME

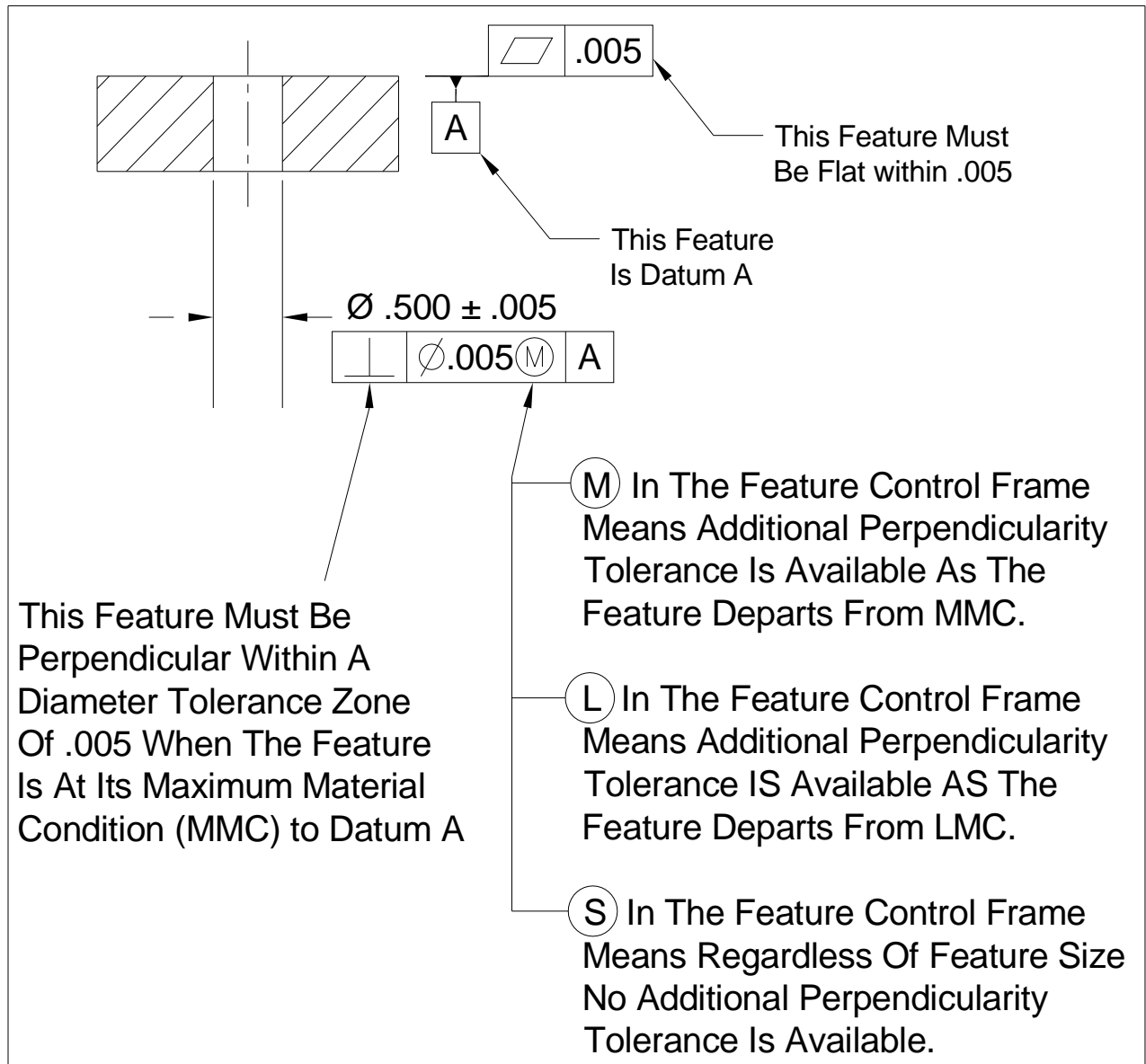
The feature control frame contains instructions for a feature. It is considered an instruction box. It specifies what the requirements are for the feature to which it is attached. Every feature control frame gives only one instruction. This means on one inspection set-up or one gage per feature control frame.



Supplementary Symbols To Modify Feature	Maximum Material Condition	Ⓜ
	Least Material Condition	Ⓛ
	Regardless of Feature Size	Ⓢ



A feature control frame is divided into compartments containing the geometric characteristic symbol followed by the tolerance. Where applicable, the tolerance is preceded by the diameter symbol (\varnothing), and may be followed by a material condition symbol (modifier). When necessary other compartments are added to contain datum references.



The datums are listed from left to right in their order of importance. The datum in the first compartment is called the primary datum. The datum in the second compartment is referred to as the secondary datum, and the datum in the third compartment is of course called the tertiary datum. In the same compartment with the datums you may find a feature modifier (M) (L) (S) if the features have size.

The placement of feature control frames is very important in communicating the designer's intent. If the feature control frame is attached to a surface, it controls that surface. An example of a feature control symbol attached to a surface is the flatness requirement in the preceding graphic. If the feature control frame is under or associated with a size tolerance, it controls the axis or median plane of that feature. An example of a feature control frame associated with size tolerance is the perpendicularity requirement of the graphic on the page preceding this one.

Note: In the 1973, ANSI Y14.5 standard (which appears on older prints) the feature control frames were written with datums first and tolerances last. In 1982, the tolerance standard was changed to show tolerances first and datums last. This change was enacted to bring the United States Standards closer in line with international practices.

In the 1994, revision of ANSI Y14.5, which changed the standard to ASME Y14.5, there is a minor adjustment on placing modifying symbols in the feature control frame. The projected tolerance zone symbol was placed outside and below the associated feature control frame. This symbol is now placed inside the feature control frame. As of 1994 no modifying symbols are to be placed outside of the frame.

Pre - 1994:

\oplus	\oslash	.005	(M)	A
		.010	(P)	

New In 1994:

\oplus	\oslash	.005	(M)	.010	(P)	A
----------	-----------	------	-----	------	-----	---



Rules for:

GEOMETRIC DIMENSIONING and TOLERANCING

There are five different rules that it is necessary to be aware of that pertain to the use of GD&T. Some apply all the time and others only in certain circumstances, so it is imperative to become familiar with and understand all the rules.

Rule #1: When a feature of size has not been toleranced with a geometric tolerance, the size tolerance governs the geometric form as well as the size of the feature. This translates to “**PERFECT FORM**” at **MMC** (Maximum Material Condition), meaning when the feature is at MMC there is no allowance for errors in roundness, straightness, parallelism, perpendicularity, etc. When a geometric tolerance is given, no element of the feature is allowed to extend beyond the **MMC** boundary minus the geometric tolerance.

Like most things there is an exception to this rule (you guessed this part). When there are two or more features interrelated such as a hole and its referenced datum, then rule #1 does not apply. These interrelated features happen when a datum is used in conjunction with the following tolerances: perpendicularity, angularity, concentricity, runout, position, and profile tolerances. If an interrelated feature is required to be within the **MMC** boundary, it should be specified as “perfect form at **MMC** required.” It could also be given zero geometrical tolerance at **MMC**.

To further elaborate on rule number one the graphic on the following page visually defines external and internal features. Pay close attention to this example. Rules are the building blocks of any system and Geometric Dimensioning and Tolerances are nothing if not a system.

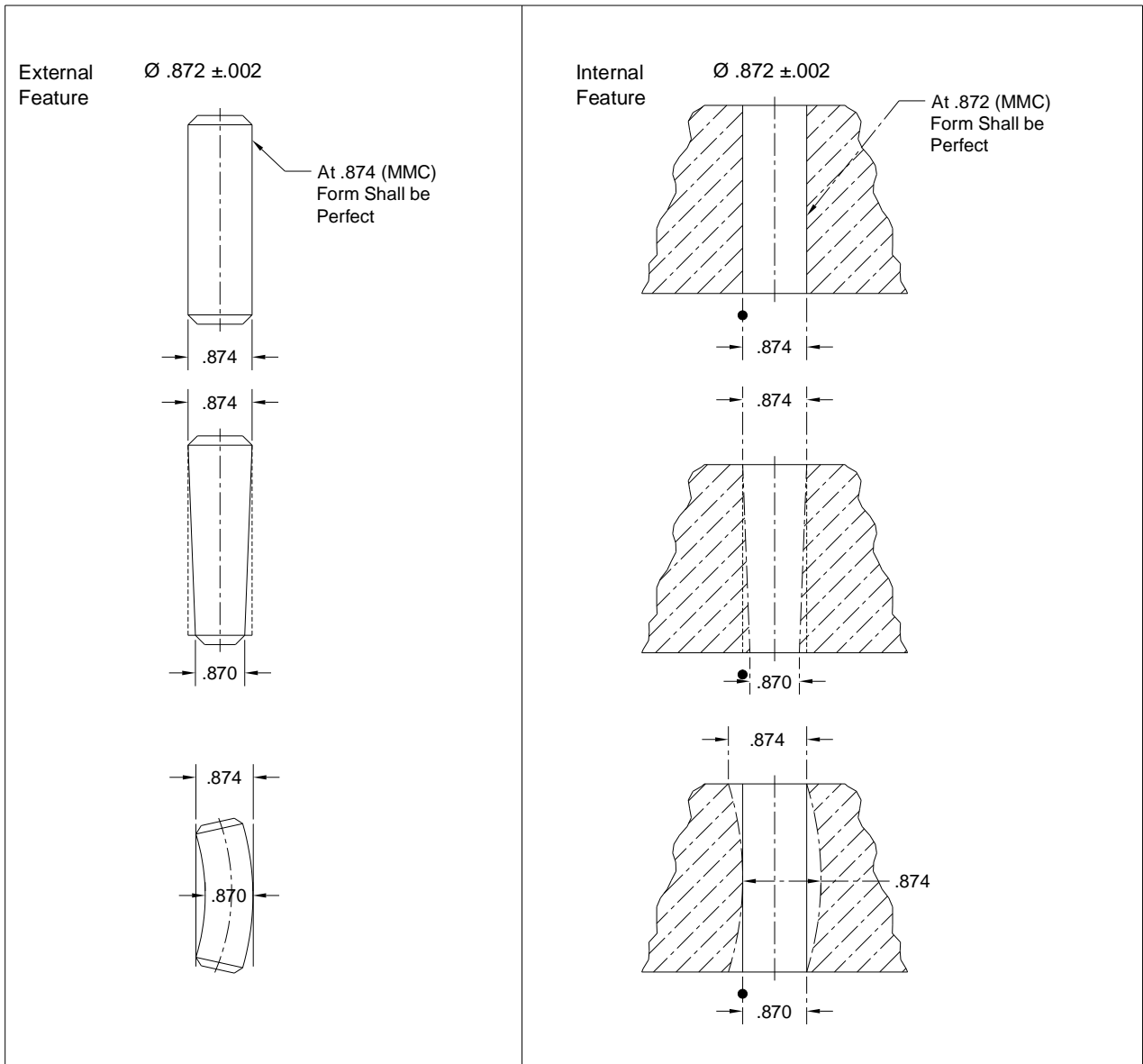


RULE #1 (INDIVIDUAL FEATURE OF SIZE)

Where only a general tolerance 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.

Variations of size: The actual size of an individual feature at any cross section shall be within specified limits of size.

Variation of form: The form of an individual feature is controlled by its limits of size to the extent prescribed in paragraphs A, B, C, and the illustration below.



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

Rule #2: RFS - (S) - (Regardless of Feature Size), MMC - (M) - (Maximum Material Condition), and LMC - (L) - (Least Material Condition) must be specified for positional tolerances, both for the feature being toleranced and for the referenced datums. ANSI (American National Standards Institute) says this must be done in every case prior to 1994. With the 1994 geometrics standard release - (S) - is no longer used in the feature control frame. If (M) or (L) is not specified in the feature control frame, there is no bonus tolerance available.

Rule #3 RFS automatically applies for all other tolerances. Except for position tolerances, it does not need to be specified. MMC on the other hand, does need to be specified.

Rule #4 When a geometric tolerance is specified for a threaded hole it applies to the axis derived from the pitch diameter. If it is otherwise, it must be specified with a note saying it is to the “major diameter” (for O.D. threads) or “minor diameter (for I.D. threads). If a geometric tolerance is called for a gear or a spline, it must specify the exact feature of the gear or spline to which it applies.

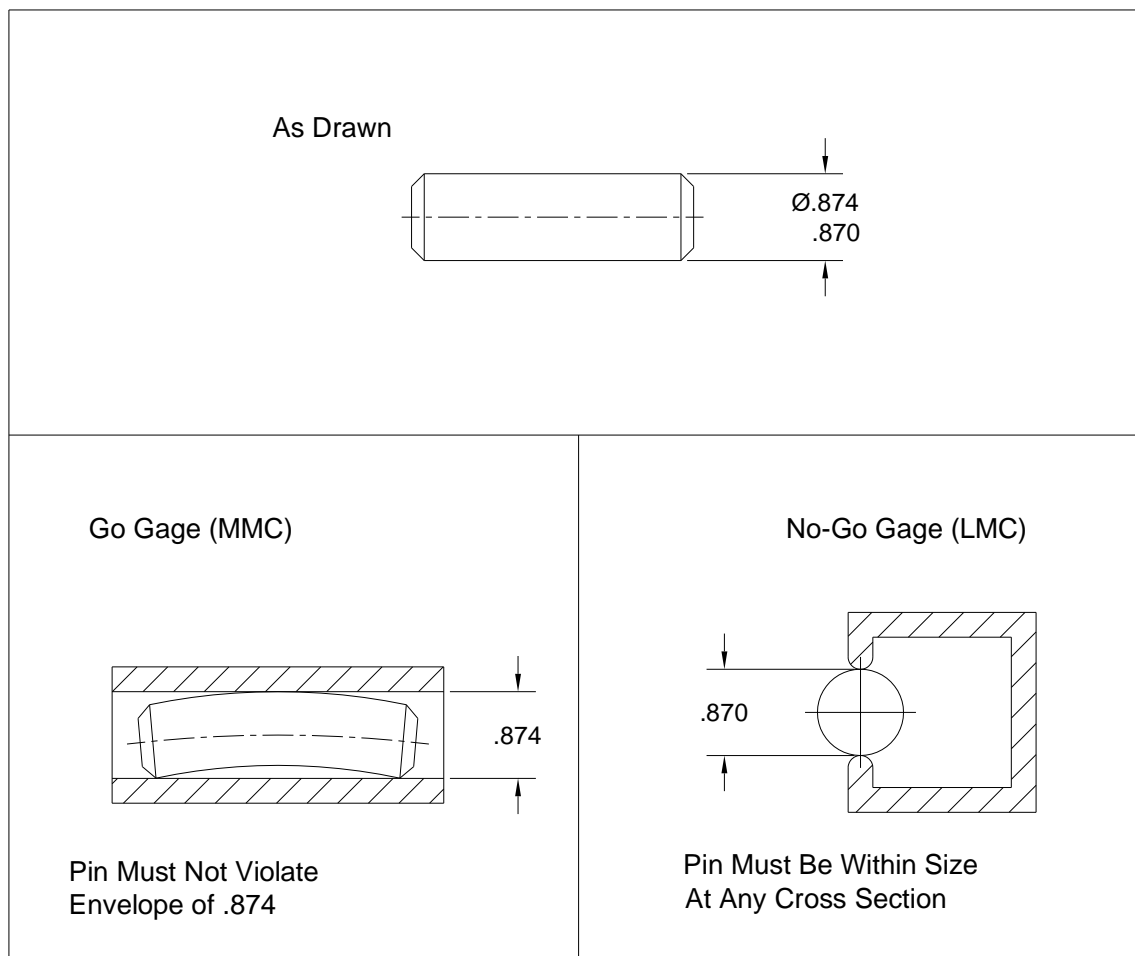
Rule #5 A feature of size that is used for a datum and is referenced MMC in a feature control frame of a related feature, applies at “Virtual Condition”. This means that if any extra tolerance gained from the datum feature must be determined at its virtual condition. Any variation from MMC to the virtual condition can be used for bonus tolerance.



LIMITS OF SIZE

The limits of size for an individual feature are described in Rule #1 in ASME Y14.5M, standard. A copy of this rule is shown in the general rules section. This concept of size limits is very important in the overall scheme of Geometric Tolerancing.

Rule #1 specifies that the size tolerance of a feature will control the form of that feature in addition to its size. This is based on the Taylor Principle (William Taylor first obtained a patent on the full form GO gage in 1905) which is the same principle as our Go and NO-Go gages that are used to check size today. An example of a pin with a size tolerance of $\varnothing .870/.874$ and the respective GO, NO-GO gages are shown in the graphic below.



Note: In reality, gages can not be made perfect. Gage maker tolerance would be applied to the above gages. Gage tolerances are usually 10% of the product tolerance with another 5% allowable wear. The gage tolerances are arranged in such a manner that a good part may be rejected but bad parts will never be accepted. To avoid complexity (and headaches) with these examples gage tolerance will not be considered at this time.



The GO gage always verifies the maximum material condition. Note that the entire pin length must be fully accepted into the $\varnothing .874$ gage. If the pin is fully accepted in the gage, both size and form are controlled within a maximum envelope of $\varnothing .874$

The NO-GO gage always verifies the least material condition. Note that according to rule #1, the gage is required to only check cross sectional points on the pin to ensure it does not violate the least material condition.

In theory rule #1 is automatically invoked when ASME Y14.5 is specified. As a practical matter in industry it is found that in many cases a relaxation or a more stringent enforcement of rule #1 is left to the discretion of inspection or quality function. An example of relaxation is using a micrometer to verify size. A micrometer check will not verify the perfect form at MMC requirement. In theory if it is desired to permit a feature to exceed the boundary of perfect form at MMC, a note such as PERFECT FORM AT MMC NOT REQD is specified exempting the pertinent size dimension from the variations of form requirements.

The control of geometric form prescribed by the limits of size does not apply to the following:

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

 - B. Parts subject to free-state variation in the unrestrained condition.
-

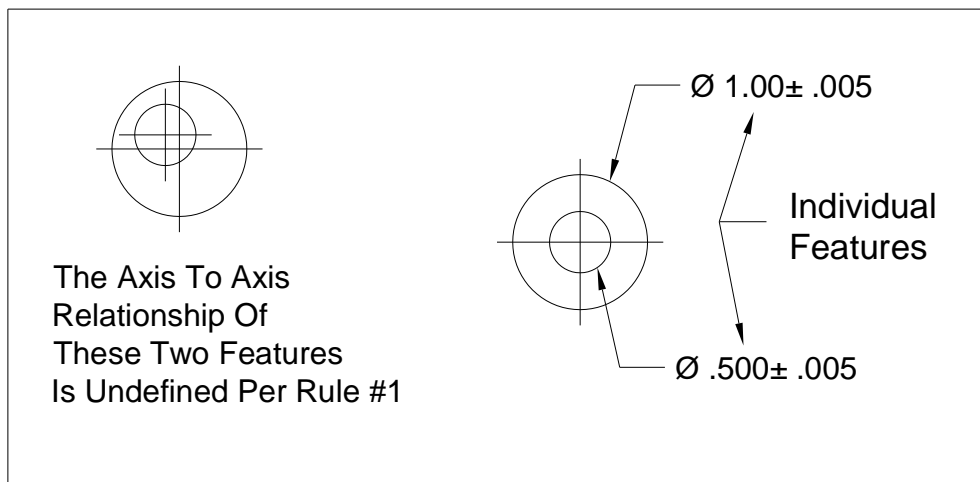
Special note: The Taylor Principle is widely accepted and recognized in the United States and elsewhere. In some countries, Rule #1 is not followed. This occurs in countries that subscribe to the International Standards Organization (ISO). ISO follows a “Principle of Independency”. Size is verified by two point measurements. Form is not included in size. If company drawings are exchanged with other countries (think Boeing outsourcing), consult the current ISO standards.



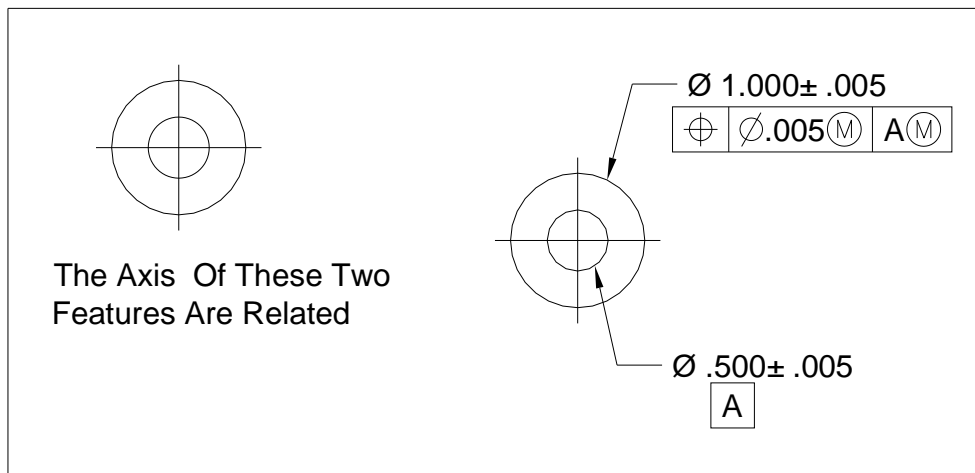
RELATIONSHIP OF INDIVIDUAL FEATURES

An important point to note is that Rule #1 only covers individual features and not the interrelationship of features. An individual feature is defined as one cylindrical or spherical surface, or a set of two parallel surfaces, each of which is associated with a size dimension. Examples of these are holes, slots, tabs, etc.

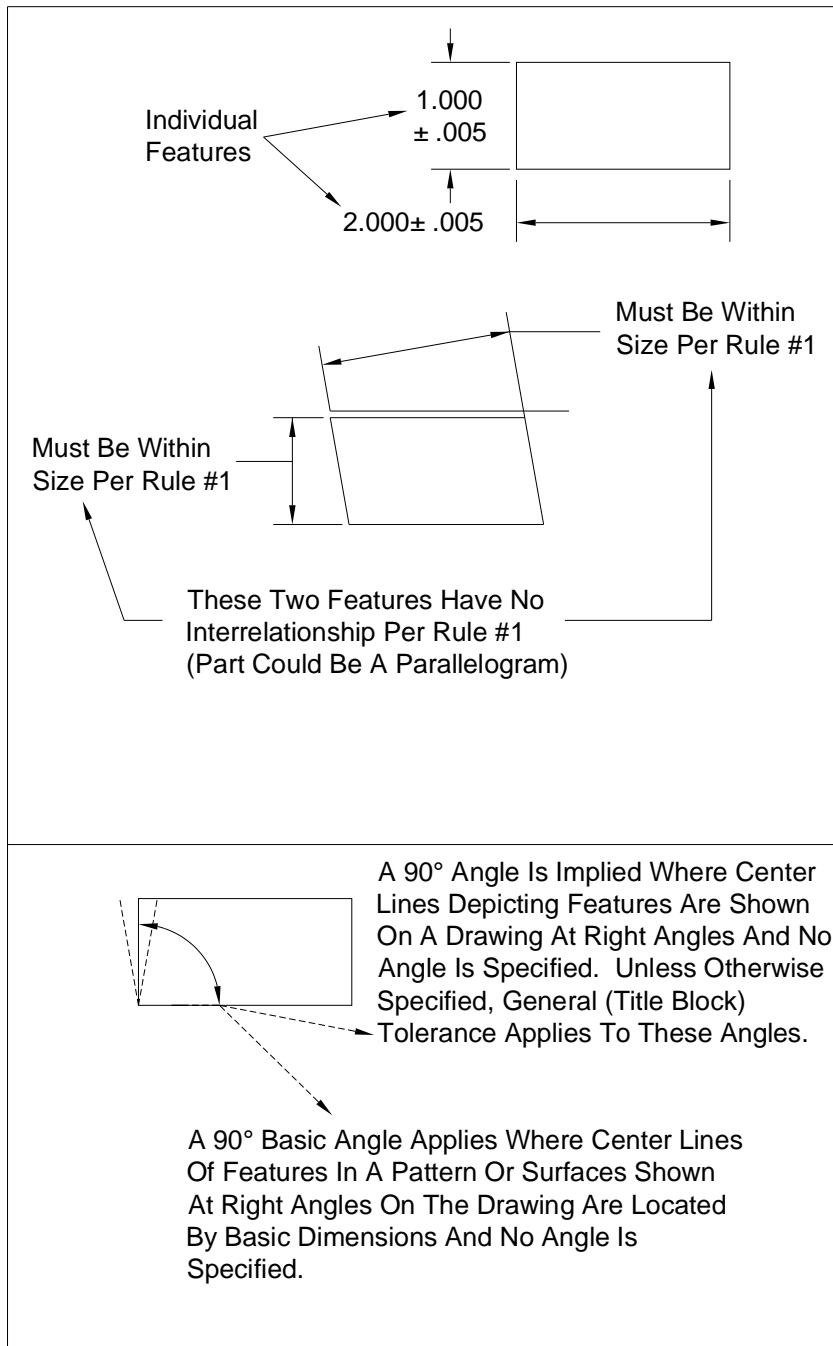
To further clarify the above statement on individual features vs. the interrelationship of features consider the example in the graphic below. A washer is shown with two individual features of size. One individual feature of size is the ID and the other individual feature of size is the OD. They are shown on the same center and are most likely to be required to be on the same center, but the drawing does not state how much tolerance one can be off to the other.



Rule #1 only covers the requirements of each individual feature and not the relationship between features. If it was desired to control the relationship of these features, then the use of datums and a positional or in some cases a form tolerance would be called out. An example of such a control is shown in the graphic below. If nothing is called out, it probably means that it is not very important and the inherent accuracy of manufacturing processes will apply. In any case it is not a clear requirement. Preferably the relationship should be specified if only in a general note or general specification.



A further clarification of individual features vs. the interrelationship of features can be found in the next graphic below. The rectangular plate has two individual features of size. One individual feature of size is the length and the other is the height. Each individual feature is covered under Rule #1, but the interrelationship between these features is not covered. The corners are shown to be square and they are implied to be 90°. The tolerance on these corners, unless otherwise specified, is whatever is specified in the general tolerance note for angles. In many cases this angle control is sufficient for part function if it is not very important to part function. If it is desired to control the relationship of the two features more closely, datums and an orientation or profile specification such as perpendicularity or profile tolerance is used.

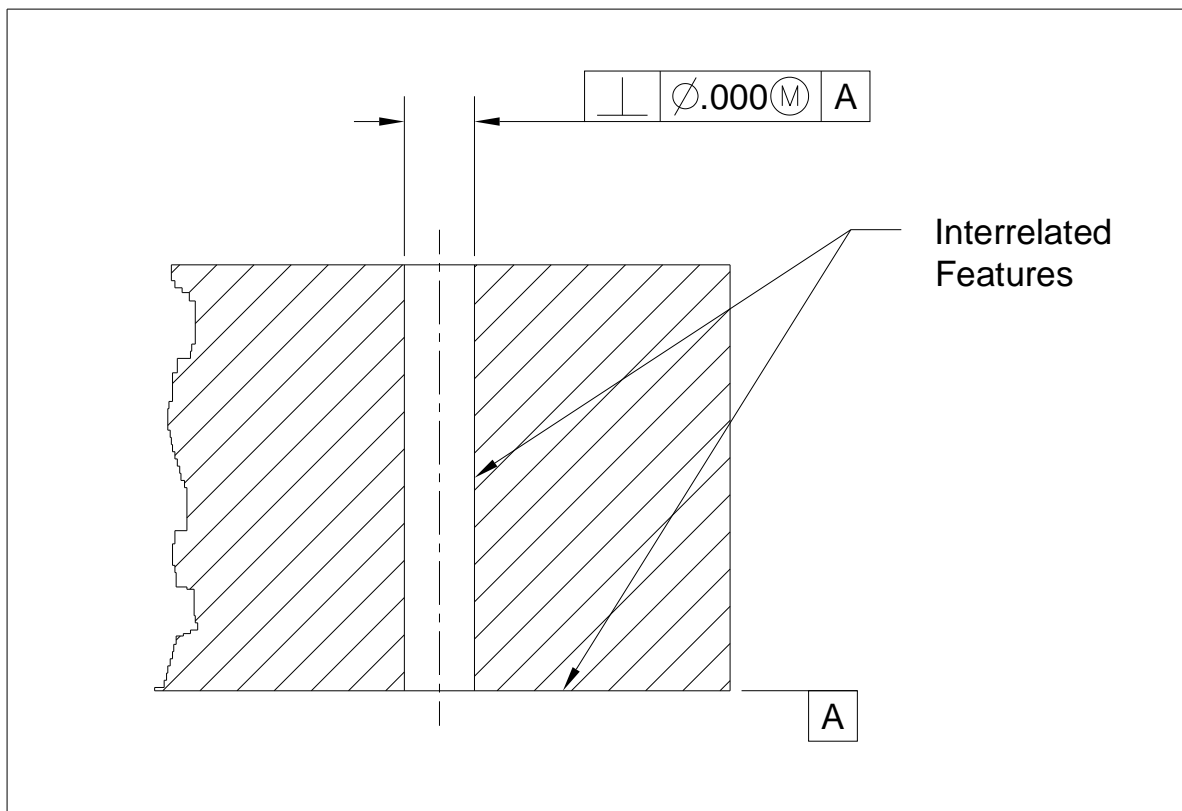


Take notice that Rule #1 prescribes requirements for individual features, but does nothing at all for the interrelationship of features. Features shown perpendicular, coaxial, or symmetrical to each other must be controlled for location or orientation to avoid incomplete communication of drawing requirements. When connecting or “hooking together” a series of individual features, it is practice to use datums and the 3 plane datum framework (Cartesian coordinates).

RELATIONSHIPS BETWEEN INDIVIDUAL FEATURES

The limits of size *do not* control the runout, orientation, or location relationship *between* individual features.

Features shown perpendicular, coaxial, or otherwise geometrically related to each other must be controlled to avoid incomplete drawing requirements. These controls for location, orientation, etc. may be specified by the use of appropriate geometric tolerances.

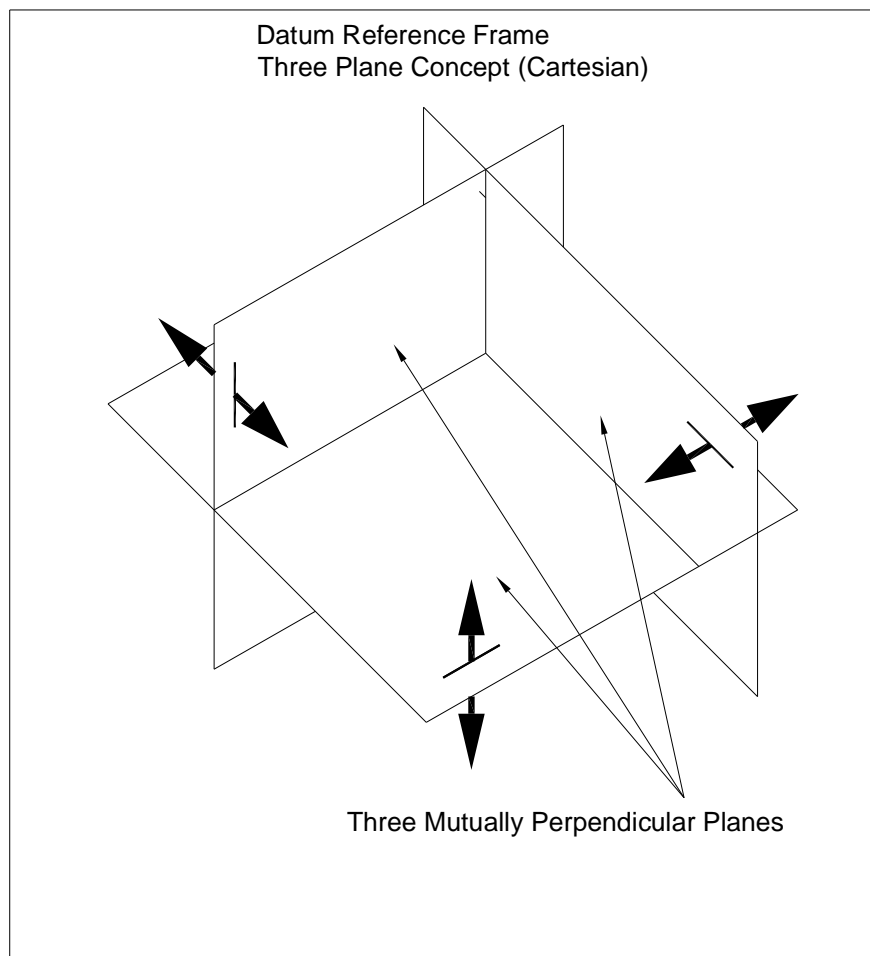


DATUM REFERENCE FRAME

(THREE PLANE CONCEPT)

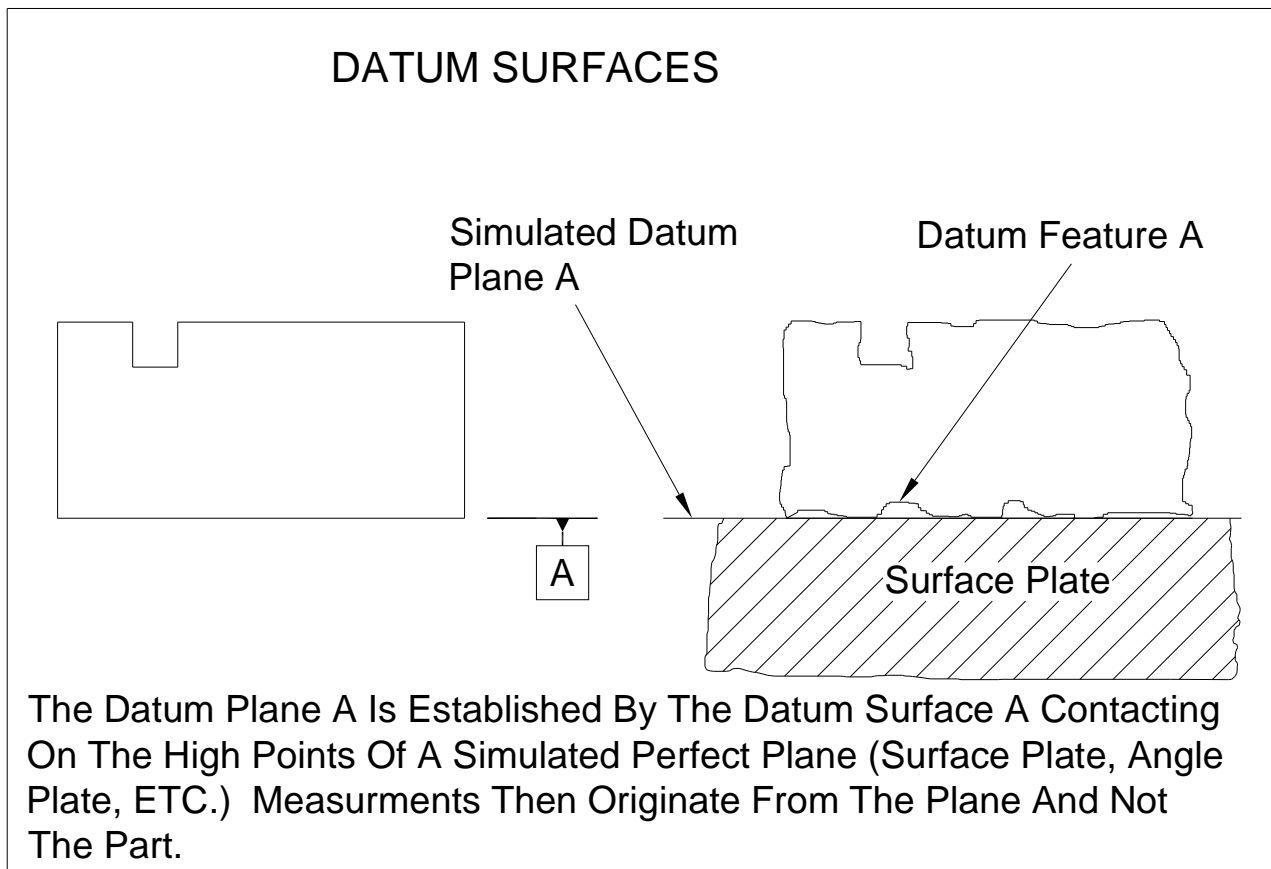
Engineering, manufacturing, and inspection all share a common three plane concept. These three mutually perpendicular planes are perfect, as perfect can be. The three mutually perpendicular perfect planes can be defined in inspection as a surface plate and two angle plates. In manufacturing these three perfect planes are the bed on a machine, a fence and a stop. In engineering the three perfect planes are the surface of the drafting table, the straight edge, and the right triangle, or more to present times the three perfect planes defined by the three dimension modeling software on a computer screen.

Engineering, manufacturing, and inspection each consider their respective planes as perfect for purposes of calculation and reference. In reality these planes are not perfect, but it is necessary to call something perfect (or treat it as if it were true) so these three mutually perpendicular planes that are simulated by all of the above equipment are considered “perfect”. In geometric tolerancing we can relate engineering, manufacturing, and inspection together by using the datum referenced frame (three plane concept) and all speak the same language. The three planes are named; primary, secondary, and tertiary planes.



ESTABLISHING DATUMS

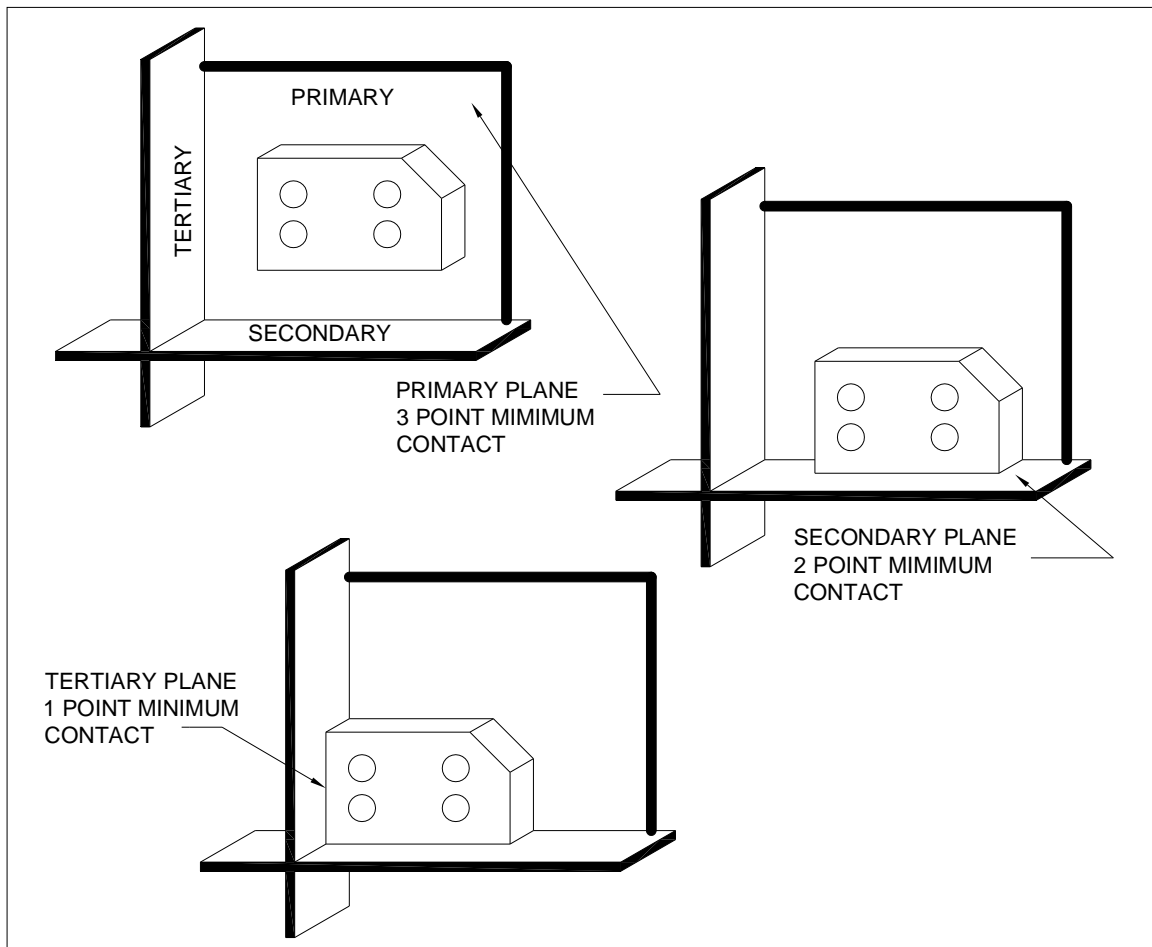
Datums indicate the origin of a dimensional relationship between a tolerated feature and a designated feature or features on a part. When a feature serves as a datum feature, its true geometric counterpart actually establishes the datums. Since measurements cannot be made from a true geometric counterpart, which is theoretical, datums are assumed to exist and be simulated with our manufacturing, processing and inspection equipment, such as the bed on machine, a collet or chuck, gage pin, surface plate or axis that the manufacturing or inspection equipment simulates and not the features themselves. These simulated planes or axis are considered perfect. It is known that our manufacturing and inspection equipment is not perfect, but they are usually created ten times better than the parts worked on so they are considered perfect or exact for purposes of calculations and reference.



POSITIONING PART IN DATUM REFERENCE FRAME

To understand geometric tolerancing and its practical application it is necessary to accept the premise that planes are perfect (logically as they are defined as perfect), and accept the concept that parts are not perfect. In fact they are never perfect, and it is not possible to build a perfect part. In order to engineer, manufacture, or inspect this part it is necessary to load an imperfect part into perfect planes to do the required work.

An illustration for orienting this imperfect part into our perfect, primary, secondary, and tertiary plane is shown in the graphic below. Each unsupported object or part has six degrees of freedom. The part must be fixed in relationship to the datum framework in order to restrict this freedom. The part is related to the primary plane by contacting a minimum of three points. It is related to the secondary datum by contacting a minimum of two points. It is finally related to the tertiary datum by contacting a minimum of one point.



All measurements are now made from the planes and not from the part. This orientation of the part onto the datum reference frame ensures a common base of measurement between engineering, manufacturing and inspection.



IMPLIED DATUMS – ORDER OF PRECEDENCE

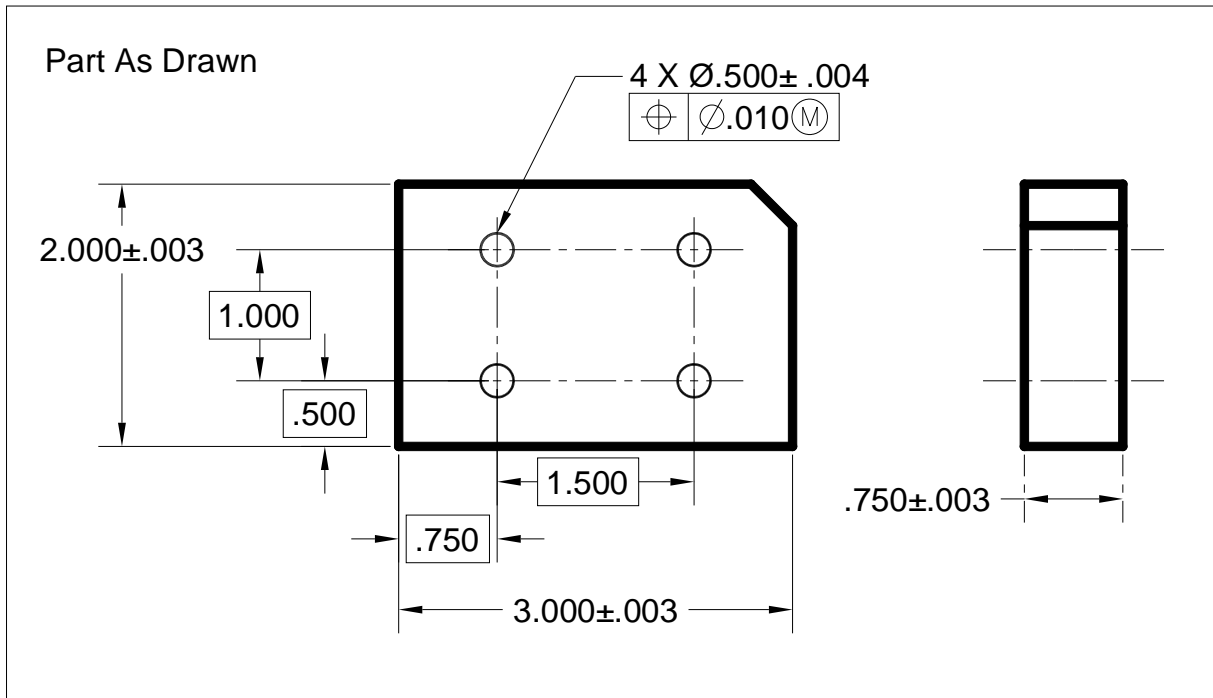
The order of precedence in the selection and establishment of datums is extremely important. Without this concept the geometric datum framework has only a very shallow meaning. Notice in the figure we have a part that has four holes. The four holes are located from the edges with basic dimensions. The datums are not called out in the feature control frame. Rather they are what are called implied datums. They are called implied because the dimensions originate from the bottom edge and the left hand edge. Thus, it is implied that these edges are datums. It is necessary that an agreement be reached by all stakeholders in the part that dimensions that originate from these edges must serve as part datums.

The problem with implied datums is that it is not possible to know the order in which the datums are to be used. We assume the datum reference frame is perfect (at least you should be convinced), but the parts are not perfect. The part is shown with square corners, but it is not possible to build the part perfect. Besides, because there is a tolerance allowed on the size, it is not required to build a perfect part according to Rule #1. Even if a tolerance of perpendicularity of .0001" is put on the corners, the part will still in theory "rock" back and forth in the datum reference frame.

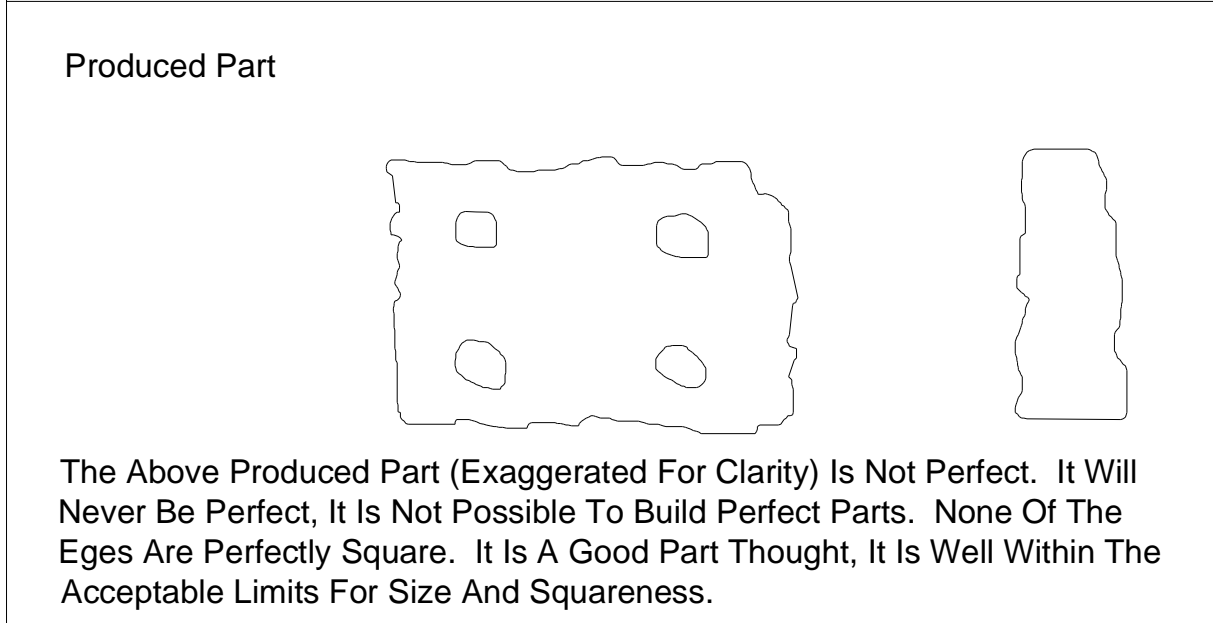
So the question becomes; how is the part loaded into the datum reference frame? Is the bottom edge supposed to be our secondary datum with two points contact or is the left hand edge supposed to be our secondary datum with two points contact? Is the large surface really the primary datum? Which datum is the tertiary? It is truly not clear. There are a number of different combinations this part can be oriented into the datum reference frame. Engineering, manufacturing, and inspection may all have different ideas as to the order that this part is loaded into the datum reference frame. This will result into different ideas as to where the holes must lie in relation to the part. It is absolutely imperative that all parties agree where the holes must lie and how the part is loaded into the reference frame. If all stakeholders agree on how the part is loaded then design (engineering), manufacturing and inspection will all make and inspect the part from the same reference ensuring good parts that meet design requirements are accepted and bad parts are always rejected. The four hole part explained above is a very simple example. As parts become more complex, with some datums being shorter or longer in length, this phenomenon of order becomes more important. If datums are not specified, it is possible to misinterpret drawing intent and reject good parts and accept bad parts.



IMPLIED DATUMS ARE NOT CLEAR

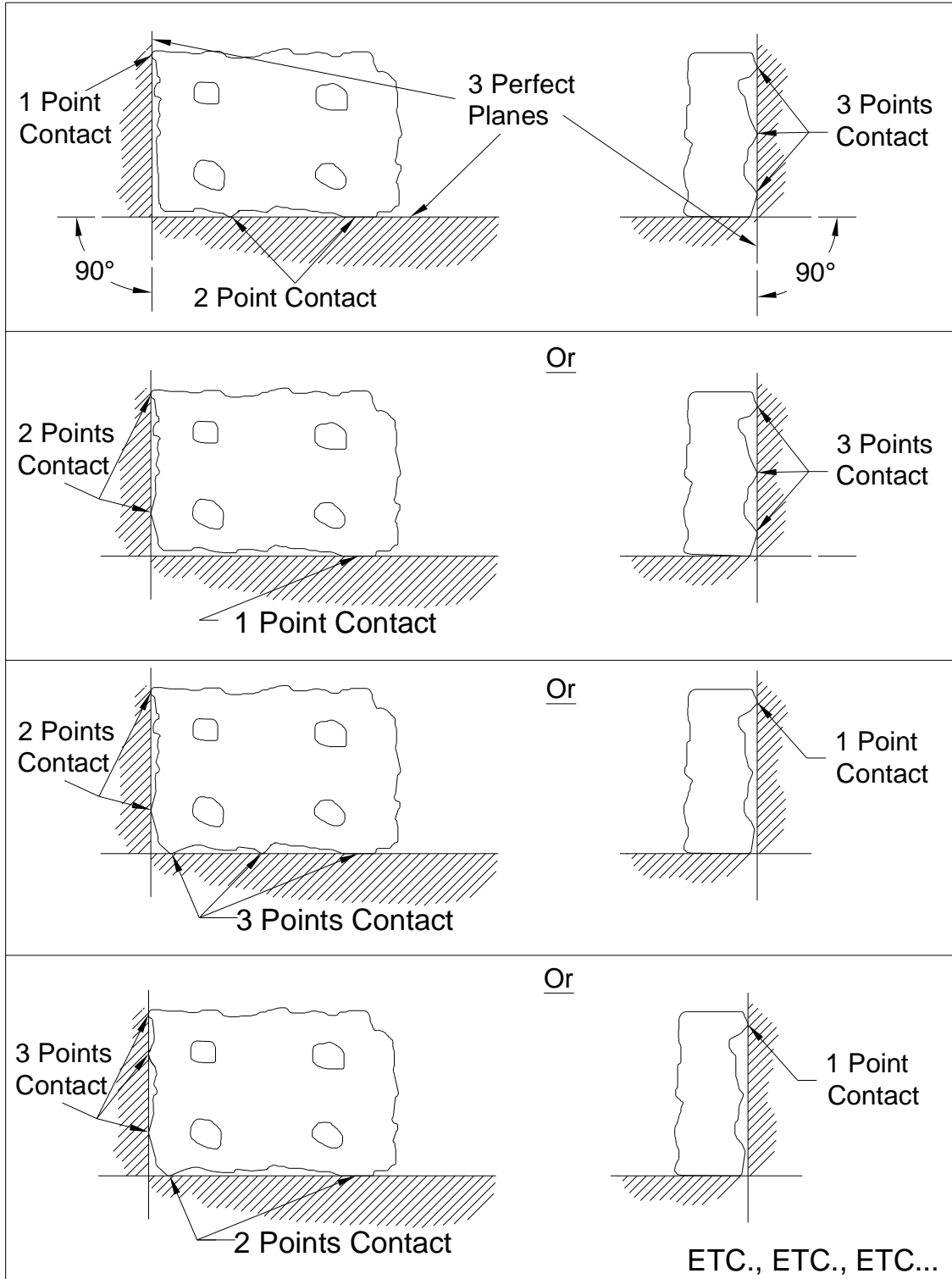


In The Part Above, The Four Holes Are Located From The Edges With Basic Dimensions. There Are No Datums Called Out In The Feature Control Frame. Rather, They Are What Is Called Implied Datums. They Are Called Implied Datums Because The Dimensions Originate From The Bottom Edge And The Left Hand Edge. Thus It Is Implied That These Edges Are The Datums.



IMPLIED DATUMS ARE NOT CLEAR

In what order is this imperfect part loaded onto the perfect datum reference frames? Remember that when hole location is measured it is done from the planes and not the part.



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

In the first datum compartment of the feature control frame, the first or primary datum will be found (three highest points of contact on face). In the second datum compartment is the secondary datum (two highest points of contact on bottom edge). In the third datum compartment is found the tertiary datum (one high point contact on left hand edge). The alphabetical order in the compartments is not important; rather it is the order the datums are placed in the compartments that is important, (first, second, third). All measurements for the hole locations would then originate from the datum planes and not the part.

If the order for the datums is changed in the feature control frame, the part is loaded into the datum framework in order to correspond with the order of the feature control frame. All measurements for the hole location are dictated by the datum planes which are communicated by the order in the feature control frame specified.

An important point to note is that, when datums are specified with a datum feature symbol, these 3, 2, 1 contact points are not actually identifiable points on that surface, but rather the highest points on that surface which come in contact with our simulated datum planes. If there are situations where it is desirable to identify particular contact points, datum targets are used.

If the part surface contacts the planes highest points, how far are the lowest points of the surface away from the plane? If nothing is stated or specified it is whatever variations in part geometry are allowed by the size tolerance and general drawing tolerance. If it is important to part function, a flatness, perpendicularity, parallelism etc, could be specified to ensure these requirements are met.

