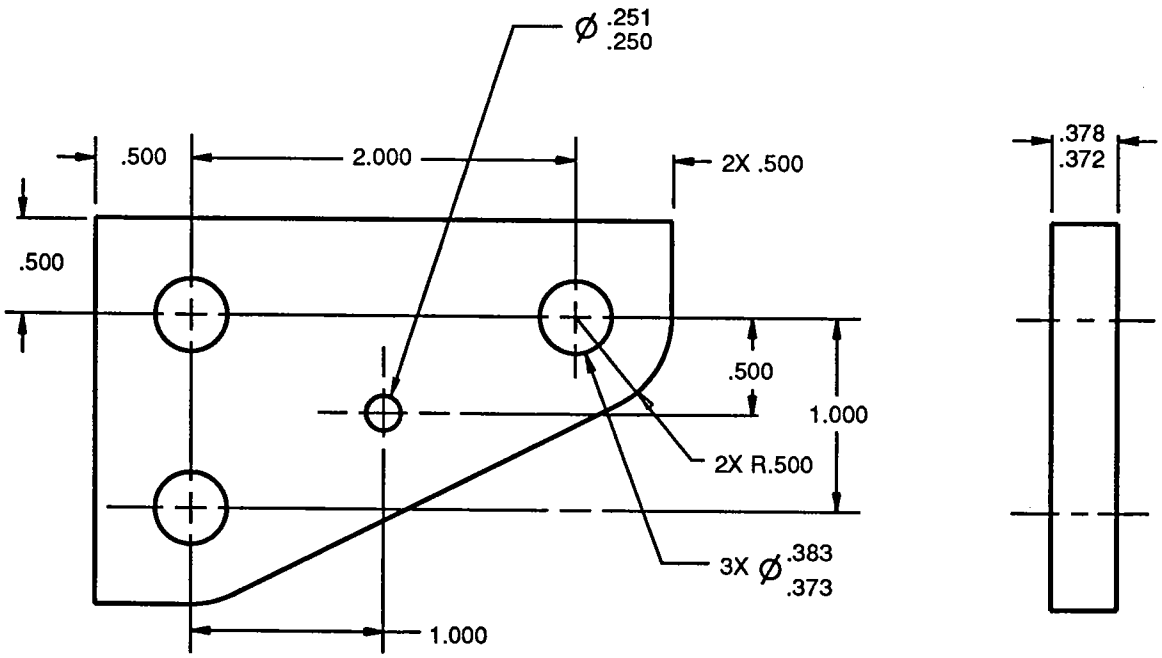
WORKSHOP EXERCISES 2.1

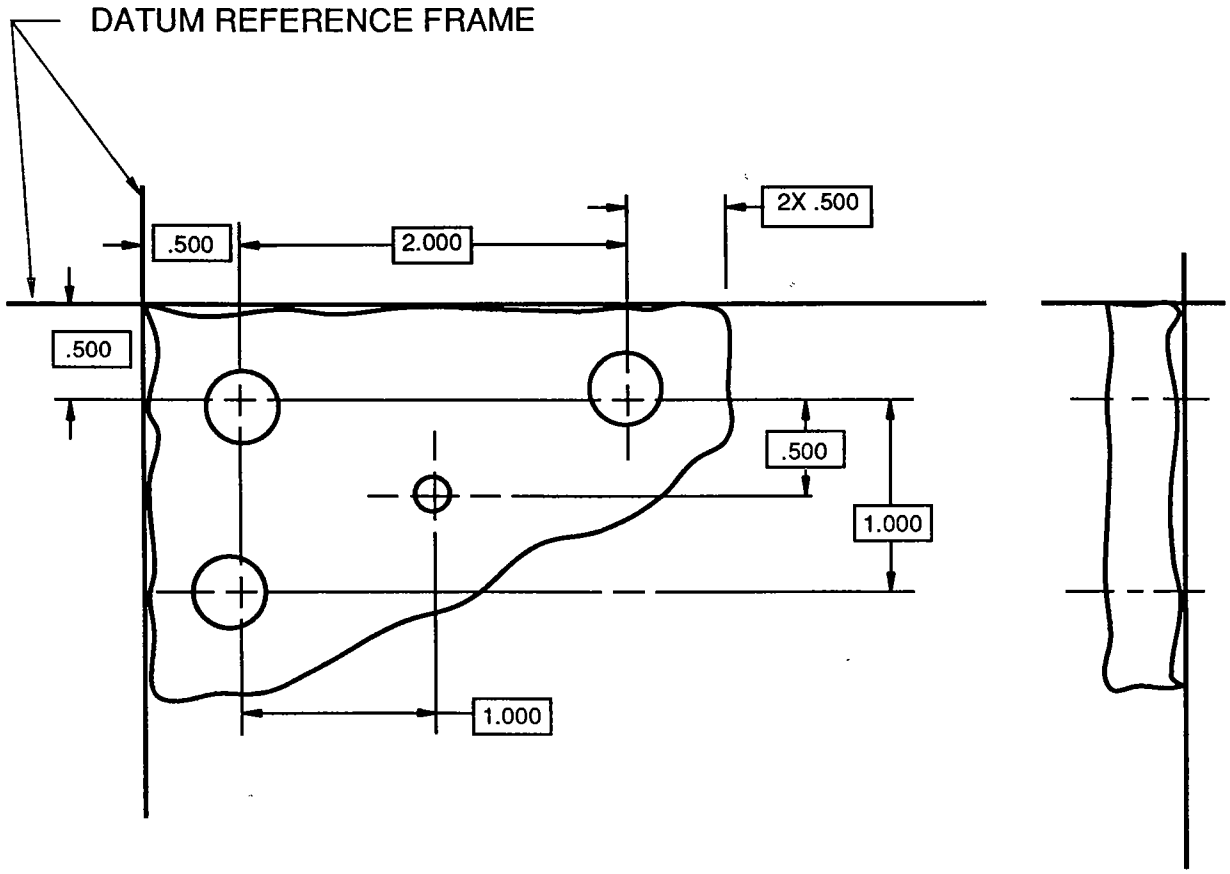
This exercise is designed to give you practical experience in correctly applying and interpreting geometric tolerancing symbology. If you have trouble applying the symbols per the following directions, page through the workbook looking for similar examples. The geometric tolerancing applied to an angle block example earlier in this unit is a good reference. Write clearly and neatly. The following directions apply to the angle plate drawing below.

1. Make the right hand face in the side view flat within .005. Also identify this surface as datum feature A.
2. Make the top surface in the plan view perpendicular within .005 to datum A. Also identify this surface as datum feature B.
3. Make the left hand surface in the plan view perpendicular within .005 to datum A (primary) and B (secondary). Also identify this surface as datum feature C.
4. Make all the dimensions basic, except of course, the size tolerances.
5. Position the three holes within a diameter zone of .010 at MMC to datums A (primary) B (secondary) and C (tertiary).
6. Position the .250 diameter hole within a diameter zone of .008 RFS in relation to datum A (primary), datum B (secondary) and datum C (tertiary).
7. In the plan view, identify the far right hand corner on the top surface as point A. Also identify the lower corner on the left hand surface as point B. On the contoured surface between A and B, apply a profile tolerance of .020 total referenced to datum A (primary), B (secondary) and C (tertiary). Under the profile feature control frame state that the tolerance applies between A and B.



WORKSHOP EXERCISE 2.1

8. The produced part (angle plate) is shown below loaded in the datum reference frame. Shade in and label the geometric tolerance zones that were applied in the previous problem. Illustrate the tolerance zones very neatly and clearly. If you need help look at other examples in the workbook.



9. In the two feature control frames below there is no feature modifier present. In the current ASME Y14.5M, 1994 standard is the tolerance for the features implied to be MMC, LMC or RFS? What is the rule number?

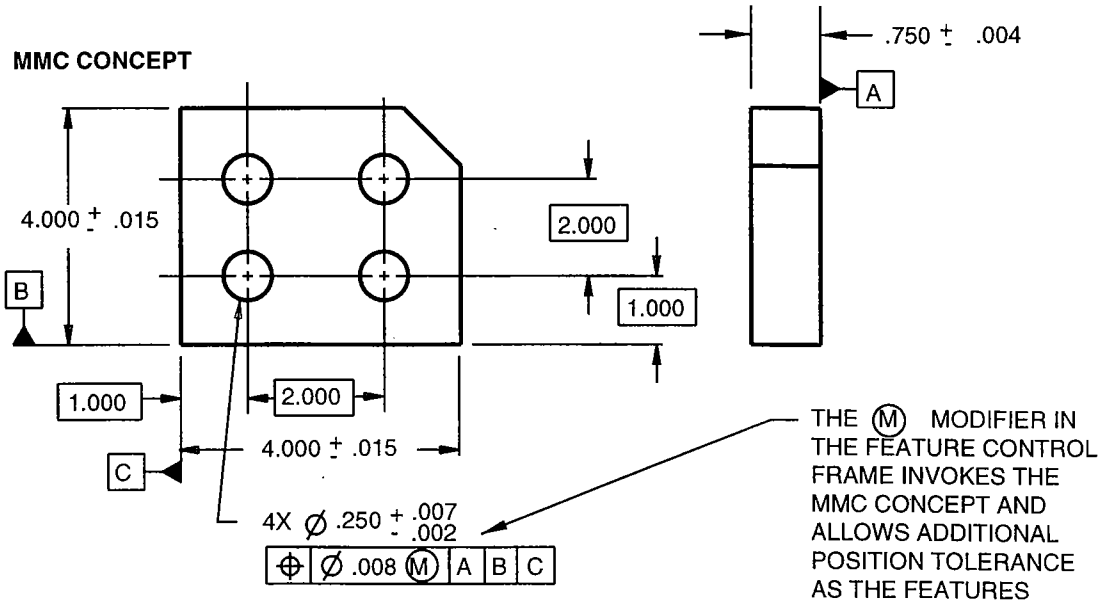
⌖	Ø.030	A	B Ⓜ

⊥	Ø.005	A

10. In interpreting older drawings, the former ANSI Y14.5M, 1982 standard rules for the applicability of modifiers was different. What was the rule for position? What was implied for all other geometric characteristics?

WORKSHOP EXERCISE 2.1

11. On the drawing below, a position tolerance has been applied to the four holes in the plate. The MMC modifier has been placed in the feature control frame. The MMC modifier allows additional positional tolerance as the features depart from MMC. In the chart below, fill in the missing blanks to show the additional position tolerance as the features depart from MMC.



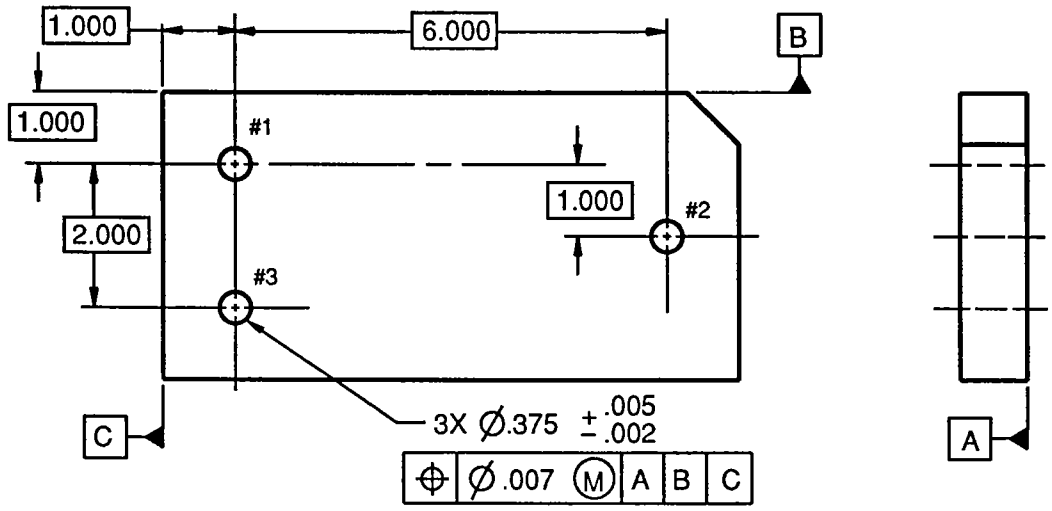
DIAMETER FEATURE SIZE	DIAMETER TOL ZONE ALLOWED
.248	.008
.249	
.250	
.251	.011
.252	
.253	.013
.254	.014
.255	
.256	
.257	.017

Fill in the missing numbers.

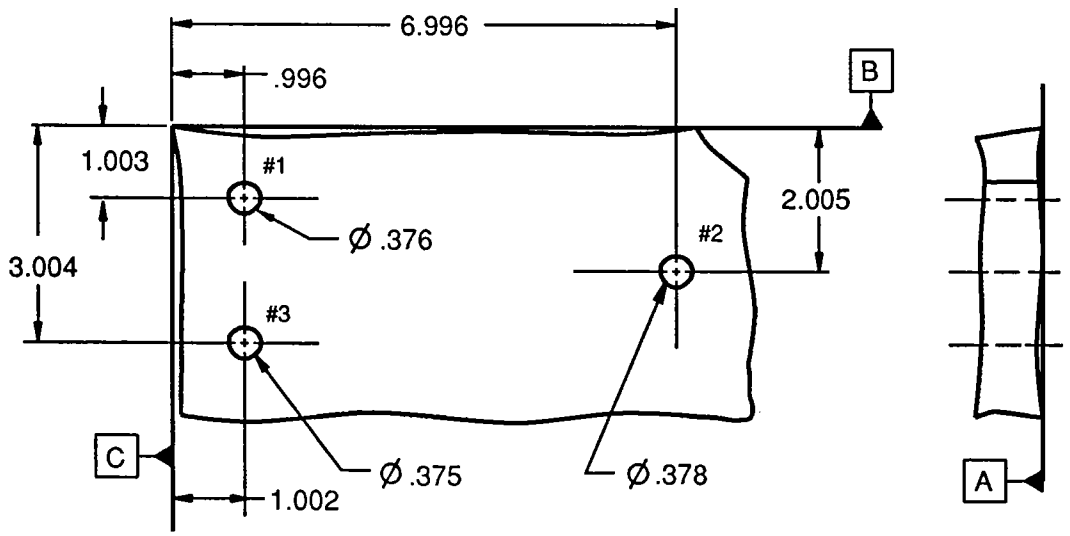
WORKSHOP EXERCISE 3.1 - PROBLEM #1

The top drawing is a plate with position tolerancing applied. The lower drawing is the produced part. Evaluate the dimensions on the produced part to verify conformance to the position tolerances. Use the chart below to record your calculations.

AS DRAWN



PRODUCED PART



FROM CHART

HOLE NO.	HOLE MMC	HOLE ACTUAL SIZE	POSITION TOLERANCE ALLOWED	"X" DIM.	"Y" DIM.	POSITION TOLERANCE ACTUAL	ACC	REJ
1								
2								
3								

GEOMETRIC CHARACTERISTIC SYMBOLS

There are 14 geometric characteristic symbols. These characteristic symbols are placed in the first compartment of a feature control frame. They define the specification that is required.

The characteristics are grouped together in categories. The major categories are form, orientation, location, runout and profile tolerances. They can also be grouped together relative to their use such as: for individual features, for related features and for individual or related features.

GEOMETRIC CHARACTERISTIC SYMBOLS

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

*ARROWHEADS MAY BE FILLED OR NOT FILLED

COMMON SYMBOLS

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

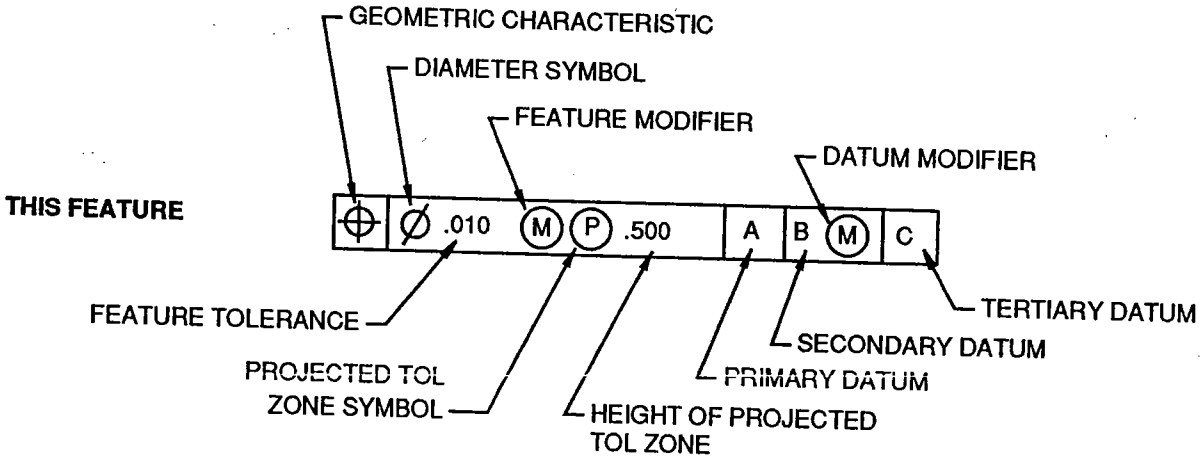
	SYMBOL	ASME Y14.5M	ISO
	FEATURE CONTROL FRAME	⌖ Ø.030 Ⓜ A B C	⌖ Ø.030 Ⓜ A B C
	DIAMETER	Ø	Ø
	SPHERICAL DIAMETER	SØ	SØ
	AT MAXIMUM MATERIAL CONDITION	Ⓜ	Ⓜ
	AT LEAST MATERIAL CONDITION	Ⓛ	Ⓛ
	REGARDLESS OF FEATURE SIZE	NONE	NONE
	PROJECTED TOLERANCE ZONE	Ⓟ	Ⓟ
X	FREE STATE	Ⓕ	Ⓕ
X	TANGENT PLANE	Ⓣ	Ⓣ (proposed)
X	STATISTICAL TOLERANCE	⟨ST⟩	NONE
X	RADIUS	R	R
X	CONTROLLED RADIUS	CR	NONE
	SPHERICAL RADIUS	SR	SR
	BASIC DIMENSION (theoretically exact dimension in ISO)	50	50
X	DATUM FEATURE	* B	* B or *
	DATUM TARGET	Ø8 A1 c A1 Ø8	Ø8 A1 c A1 Ø8
	TARGET POINT	×	×
	DIMENSION ORIGIN	⌖→	⌖→
	REFERENCE DIMENSION (auxiliary dimension in ISO)	(50)	(50)
	NUMBER OF PLACES	8X	8X
	COUNTERBORE/SPOTFACE	⌴	⌴ (proposed)
	COUNTERSINK	⌵	⌵ (proposed)
	DEPTH/DEEP	↧	↧ (proposed)
	SQUARE	□	□
	ALL AROUND	⌖→	NONE
	DIMENSION NOT TO SCALE	150	150
	ARC LENGTH	⌒150	⌒150
X	BETWEEN	↔	NONE
	SLOPE	◺	◺
	CONICAL TAPER	◁	◁
	ENVELOPE PRINCIPLE	NONE (implied)	Ⓔ

*MAY BE FILLED OR NOT FILLED

FEATURE CONTROL FRAME

The feature control frame is probably the most important symbol in the geometric tolerancing system. It states the requirements or instructions for the features to which it is attached. As its name implies, the feature control frame controls features. Each feature control frame will state only one requirement or one message. There is only one set-up or only one gage for each feature control frame. If there are two requirements for a feature, it will require two feature control frames.

To help interpret a feature control frame, it is important to note that there are always two words that are implied to precede a feature control frame. The two words are "This feature" as in "This feature flat" or "These features positioned."



The first compartment of a feature control frame will always contain one of the 14 geometric characteristic symbols. There can never be two geometric characteristic symbols placed in a feature control frame. If there are two requirements, we must use two feature control frames. The geometric characteristic symbol in the first compartment will stipulate the requirement for the feature such as: this feature must be flat, or this feature must be parallel, etc.

The second compartment of a feature control frame will always contain the total tolerance for the feature. The feature tolerance is always specified as a total tolerance. It is never a plus/minus value.

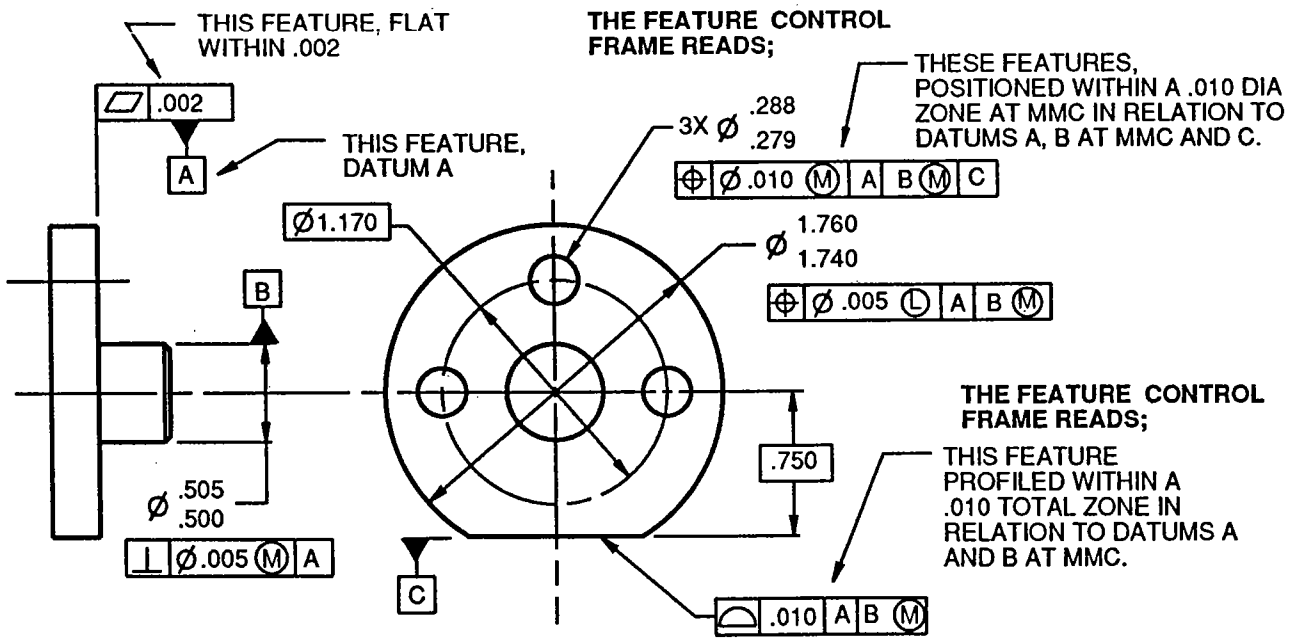
If the tolerance is preceded by a diameter symbol Ø, the tolerance will be a diameter or cylindrical shaped tolerance zone as in the location of a hole. If the tolerance is preceded by a spherical diameter symbol SØ, the tolerance will be a spherical shaped zone as in the location of a ball or sphere. If there is no symbol preceding the tolerance, the default shape tolerance is parallel planes or a total wide zone as in the position of a slot or a profile of a surface.

Following the feature tolerance a feature modifier such as MMC Ⓜ or LMC Ⓛ can be specified. This will occur if the feature being controlled is a feature of size such as a hole, slot, tab or pin. If the feature being controlled is a feature of size and no modifier is specified, the default is RFS. (See modifier rules for more information.) If the feature being controlled has no size, such as a plane surface, then no modifier can be specified.

Also, in the second compartment, following the tolerance and specified modifier, can be found additional symbols, if they are applicable. These symbols can be for the projected tolerance zone, free state, tangent plane and statistical tolerance.

The third and following compartments of a feature control frame contain the specified datums, if datums are applicable. For example, a form tolerance like flatness or straightness will not allow specified datums. The datums are specified in their order of importance such as primary, secondary and tertiary. The alphabetical order of the datums has no significance. The significance is their order of precedence in the feature control frame. (See datum section for more information.)

If any of the datums are features of size, a datum modifier such as MMC Ⓜ or LMC Ⓛ may be specified. If the specified datum feature has size and no modifier is specified, the default is RFS. (See modifier rules for more information.)



The placement of a feature control frame is very important. If the feature control frame is attached or directed to a surface, it controls that surface as in a flatness or profile control. If the feature control frame is attached to or associated with a feature of size, then it controls the axis or median plane of that feature as in a position callout.