Frequently Asked Questions (FAQ)

1) Is Geometric Dimensioning used on just Die Castings and why should it be used? See page 5-2 - Why should GD&T be used?
2) What is a Location Tolerance? See page 5-11, Location Tolerances
3) How do I convert a linear tolerance to true position? See pages 5-32 through 5-34, Conversion of Position.
4) Is a list of GD&T symbols availible? See page 5-8, GD&T Symbols and Meanings.
5) When can I use Profile of a surface instead of flatness? See page 5-14, Profile Tolerances.

1 Introduction

The concept of Geometric Dimensioning and Tolerancing (GD&T) was introduced by Stanley Parker from Scotland in the late 1930's. However, it was not used to any degree until World War II (WW II) because until then the vast majority of products were made in-house. The designer could discuss with the manufacturing personnel (die designer, foundry foreman, machinist, and inspectors) what features were to be contacted to establish the so called "centerlines" that were used on the drawing to locate features such as holes and keyways. Also when two (2) or more features were shown coaxial or symmetrical around these "centerlines", the questions that needed to be answered by the designer was, "how concentric or symmetrical do these features have to be to each other"?. During WW II companies had to "farm out" parts because of the quantities/schedules. This meant the new manufacturer had to interpret the drawing hence the "centerlines" were often established by contacting features that were not functional or important and features produced from these incorrect "centerlines" were not at the location required. The parts did not assemble and/or did not function properly and had to be fixed or scrapped. GD&T was the solution to this major problem. GD&T provides a designer the tools to have clear, concise, and consistent instructions as to what is required. It eliminates ambiguities so that everyone involved with the part will not have to interpret the dimensioning.

This section should be used as a handy reference guide for using GD&T with respect to the die casting process. ASME should be consulted for a more comprehensive guide to GD&T.

2 What is GD&T?

It is compilation of symbols and rules that efficiently describe and control dimensioning & tolerancing for all drawings (castings, machined components,etc.). It is documented in ASME Y14.5M which has the symbols, rules, and simple examples. ASME Y14.8 is a good reference to use for guidance on additional features for casting and forging drawings.

3 Why should GD&T be used?

GD&T is the best effort we have so far to clarify the dimensioning of a part. Die casters should welcome and even insist upon the use of GD&T on any part prints. GD&T requirements should be reviewed and agreed upon between the die caster and designer.

a. It is a simple and efficient method for describing the tolerancing mandated by the designer of the part.

b. It eliminates ambiguities as to what Datum features are to be contacted to establish the Datum planes and/or Datum axis that are to be used for locating other features. All inspection will result in the same result – the dimension is within or out of tolerance. Fig. 5-1 illustrates a simple example of ambiguities associated with the "old" type drawing. Fig. 5-2 illustrates the same example with GD&T.

c. It simplifies inspection because hard gages can often be utilized and inspection fixtures are often mandated which simplifies inspection for production quantities.

d. It forces the designer to totally consider function, manufacturing process, and inspection methods. The result is larger tolerances that guarantee function, but reduce manufacturing amd inspection costs. Also the "bonus" or extra tolerance for certain conditions can result in significant production cost savings. In addition the time to analyze whether a missed dimension is acceptable is dramatically reduced

Note: GD&T convention is to keep units constant between dimensions and tolerance. Although the examples in this section are unitless the numbers correspond to dimensions in inches.

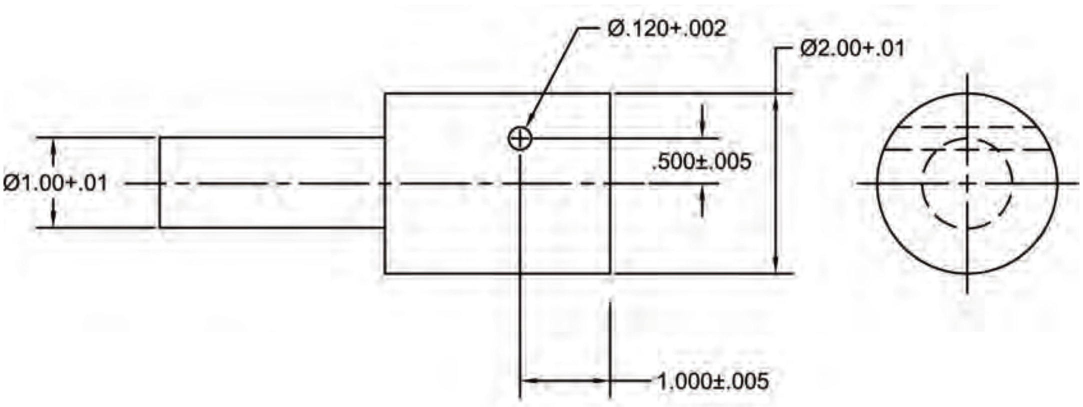

Figure 5-1 "OLD" Drawing without GD&T.

Questions:

1) What is the relationship (coaxiality tolerance) between the ∅1.00 and the ∅2.00?

2) Which feature (∅1.00 or ∅2.00) is to be used for measuring (locating) the .500±.005 dimension for locating the ∅.120 hole?

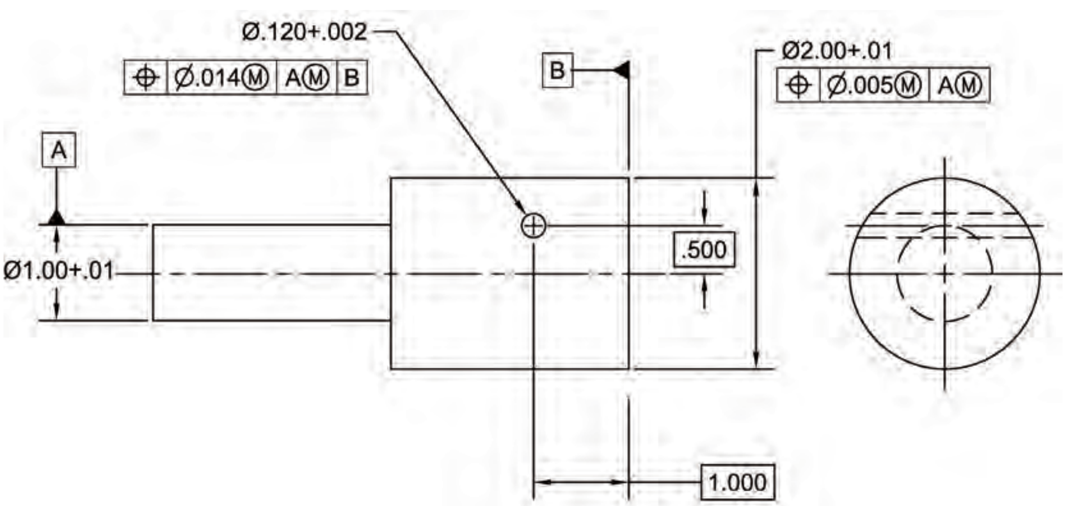

Figure 5-2 "NEW" Drawing with GD&T.

Questions asked in Fig. 5-1 answered:

1) The axis of the ∅2.00 has to be coaxial with the axis of the ∅1.00 within a tolerance zone that is a ∅.005 if the ∅ is 2.01 which is the Maximum Material Condition (MMC).

2) The ∅1.00 is the feature to be used for measuring the .500 dimension for locating the n.120 hole. The tolerance for locating the ∅.120 hole is a ∅ of .014 (the diagonal of the rectangular tolerance zone shown in Fig. 5-1) when the hole is a MMC (∅.120).

4 Datum Reference Frame (DRF)

The DRF is probably the most important concept of GD&T. In order to manufacture and/or inspect a part to a drawing , the three (3) plane concept is necessary. Three (3) mutually perpendicular (exactly 90˚ to each other) and perfect planes need to be created to measure from. In GD&T this is called Datum Reference Frame whereas in mathematics it is the Cartesian coordinate system invented by Rene Descartes in France (1596-1650). Often one would express this concept as the need to establish the X,Y, and Z coordinates. The DRF is created by so-called Datum Simulators which are the manufacturing, processing, and inspection equipment such as surface plate, a collet, a three jaw chuck, a gage pin, etc. The DRF simulators provide the origin of dimensional relationships. They contact the features (named Datum Features) which of course are not perfect hence measurements from simulators (which are nearly perfect) provides accurate values and they stabilize the part so that when the manufacturer inspects the part and the customer inspects the part they both get the same answer. Also if the part is contacted during the initial manufacturing setup in the same manner as when it is inspected, a “layout” for assuring machining stock is not required. The final result (assuming the processing equipment is suitable for the tolerancing specified) will be positive.

4.1 Primary, Secondary, and Tertiary Features & Datums

The primary is the first feature contacted (minimum contact at 3 points), the secondary feature is the second feature contacted (minimum contact at 2 points), and the tertiary is the third feature contacted (minimum contact at 1 point). Contacting the three (3) datum features simultaneously establishes the three (3) mutually perpendicular datum planes or the datum reference frame. If the part has a circular feature that is identified as the primary datum feature then as discussed later a datum axis is obtained which allows two (2) mutually perpendicular planes to intersect the axis which will be the primary and secondary datum planes. Another feature is needed (tertiary) to be contacted in order orientate (fix the two planes that intersect the datum axis) and to establish the datum reference frame. Datum features have to be specified in an order of precedence to properly position a part on the Datum Reference Frame. The desired order of precedence is obtained by entering the appropriate datum feature letter from left to right in the Feature Control Frame (FCF) (see Section 5 for explanation for FCF). The first letter is the primary datum, the second letter is the secondary datum, and the third letter is the tertiary datum. The letter identifies the datum feature that is to be contacted however the letter in the FCF is the datum plane or axis of the datum simulators. Note that there can be multiple datum sets used to reference different features on the casting. See Fig. 5-3 for Datum Features & Planes.

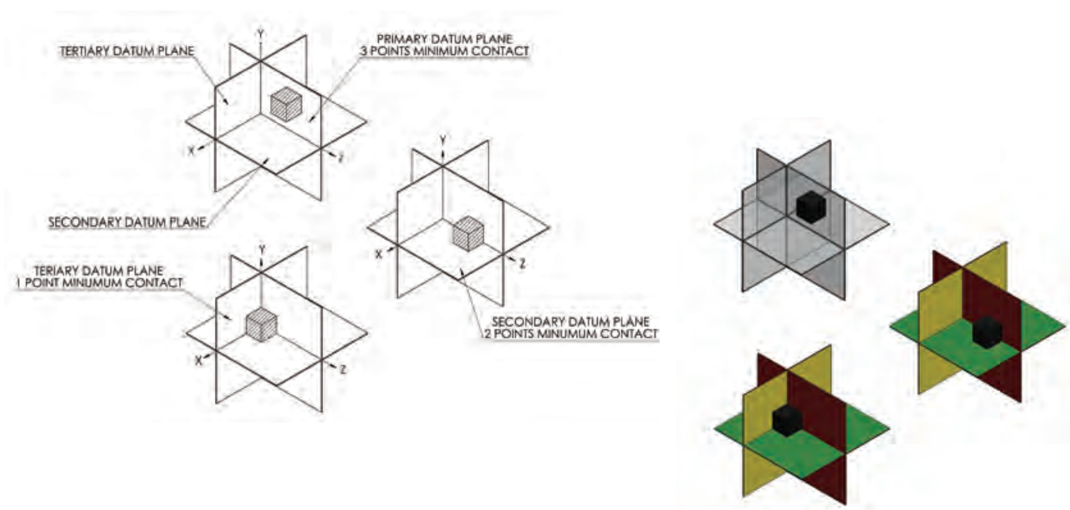

Figure 5-3 Primary, secondary, tertiary features & datum planes.

4.2 Datum Feature vs Datum Plane

The datum features are the features (surfaces) on the part that will be contacted by the datum simulators. The symbol is a capital letter (except I,O, and Q) in a box such as $\boxed{A}$ used in the 1994 ASME Y14.5 or -A- used on drawings made to the Y14.5 before 1994. The features are selected for datums based on their relationship to toleranced features, i.e., function, however they must be accessible, discernible, and of sufficient size to be useful. A datum plane is a datum simulator such as a surface plate. See Fig. 5-4 for a Datum Feature vs a Datum Plane.

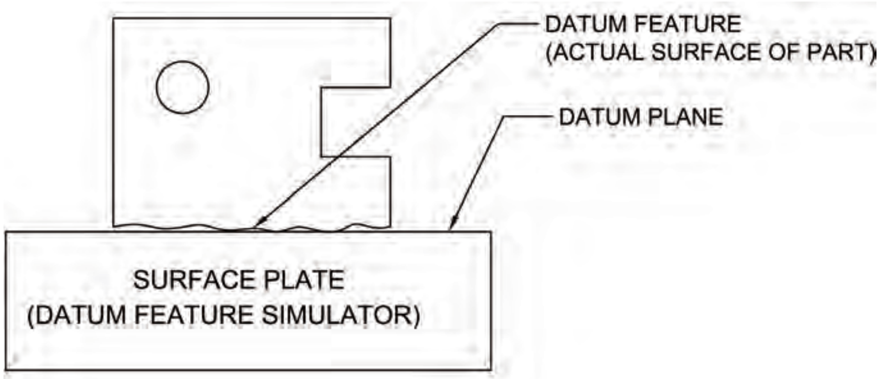

Figure 5-4 Datum feature vs. datum plane.

4.3 Datum Plane vs Datum Axis

A datum plane is the datum simulator such as a surface plate. A datum axis is also the axis of a datum simulator such as a three (3) jaw chuck or an expandable collet (adjustable gage). It is important to note that two (2) mutually perpendicular planes can intersect a datum axis however there are an infinite number of planes that can intersect this axis (straight line). Only one (1) set of mutually perpendicular planes have to be established in order to stabilize the part (everyone has to get the same answer – does the part meet the drawing requirements?) therefore a feature that will orientate or “clock” or “stabilize” has to be contacted. The datum planes and datum axis establish the datum reference frame and are where measurements are made from. See Fig. 5-5 for Datum Feature vs Datum Axis.

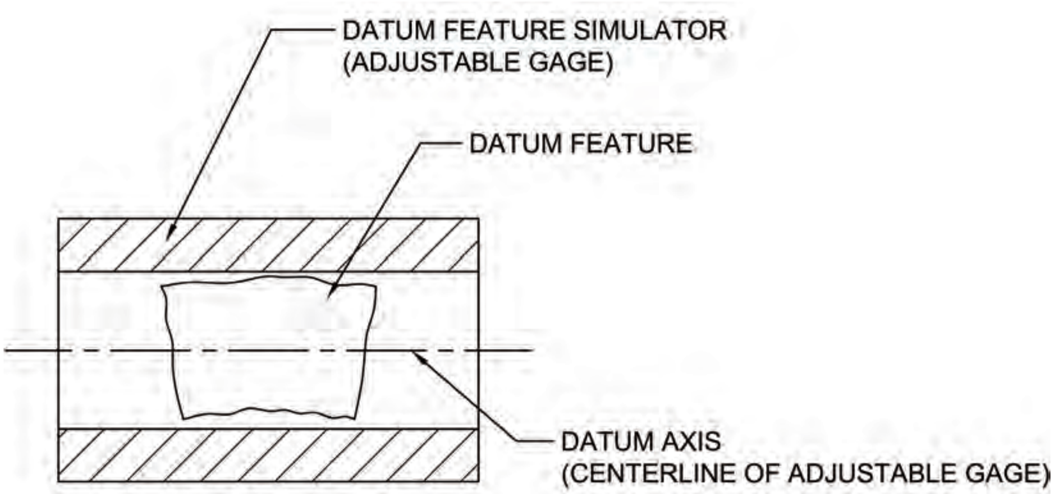

Figure 5-5 Datum feature vs. datum axis.

5

4.4 Datum Target Sizes & Locations

Datum targets are datum simulators such as spherical pins or round flat bottom pins or three (3) jaw chucks or centers that establish datum planes or a datum axis. They contact the datum features and are often specified to be used for inspecting parts that are inherently not round or straight or flat or they are large parts. If targets are not used then the entire datum feature has to contact a datum simulator. An example of what can result is the part could “rock” on a surface plate if the part was not relatively flat which would result in an unstable scenario and conflicting results. If the datum feature is large a datum simulator that contacts the entire feature may not exist or would be extremely expensive to produce. The datum targets are the datum planes and datum axis and often are assembled together to create an inspection fixture and or a manufacturing fixture. See Fig. 5-6 for Datum Target Sizes & Locations.

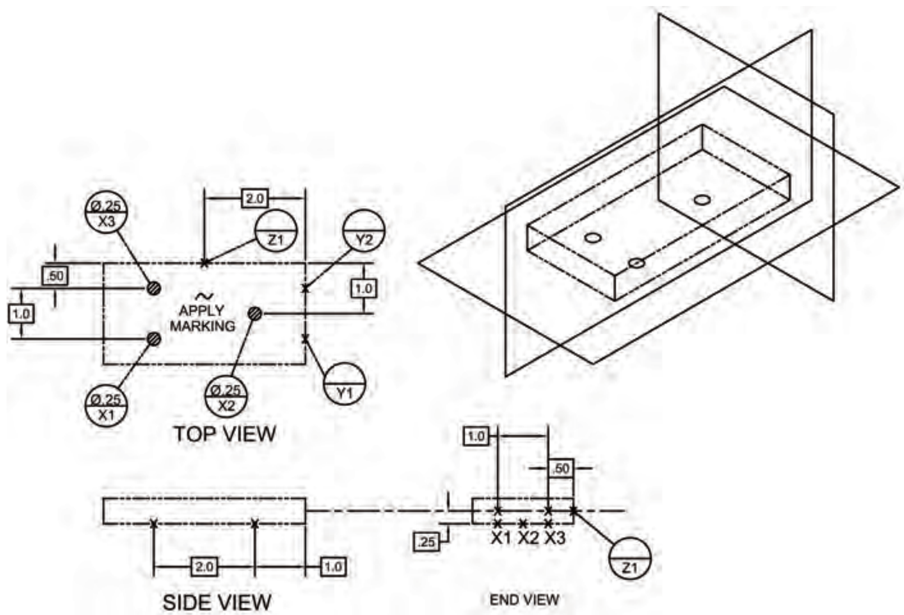

Figure 5-6 Target sizes & locations.

Component configuration shown as phantom lines on separate drawing
• Illustrates orientation when targets contact component
• Illustrates that targets are physically separate from the component
• Apply marking is shown to depict which side is to be contacted by the targets

5 Feature Control Frame

The geometric tolerance for an individual feature is specified in the Feature Control Frame which is divided into compartments – see Fig 5-7. The first compartment contains the type of geometric characteristic such as true position, profile, orientation, etc. The second compartment contains the tolerance (where applicable the tolerance is preceded by a diameter symbol and followed by a material condition symbol). The remaining compartments contain the datum planes or axis in the proper sequence (primary datum is the first letter).

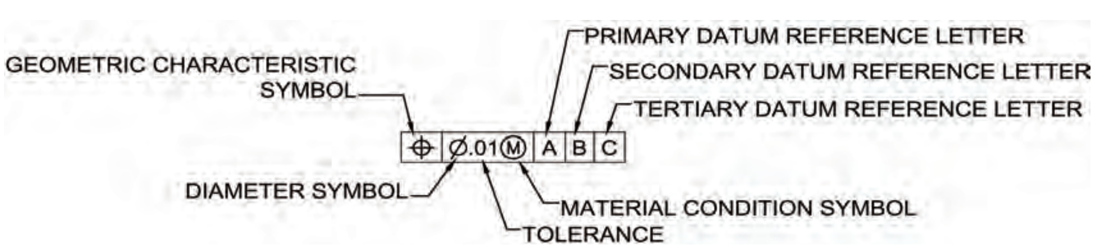

Figure 5-7 Feature control frame.

6 Rule # 1 – Taylor Principle (Envelope Principle)

When only a size tolerance is specified for an individual feature of size the form of this feature shall not extend beyond a boundary (envelope) of perfect form at maximum material condition (MMC). In other words, when the size is at MMC the feature has to be perfectly straight. If the actual size is less than the MMC the variation in form allowed is equal to the difference between the MMC and the actual size. The relationship between individual features is not controlled by size limits. Features shown perpendicular, coaxial or symmetrical to each other must be controlled for location or orientation otherwise the drawing is incomplete. In other words Fig. 5-1 is an incomplete drawing. Fig. 5-8 shows the meaning of Rule #1 for an external cylinder (pin or shaft) and an internal cylinder (hole). Note that a hard gage can be used to inspect this principle or requirement.

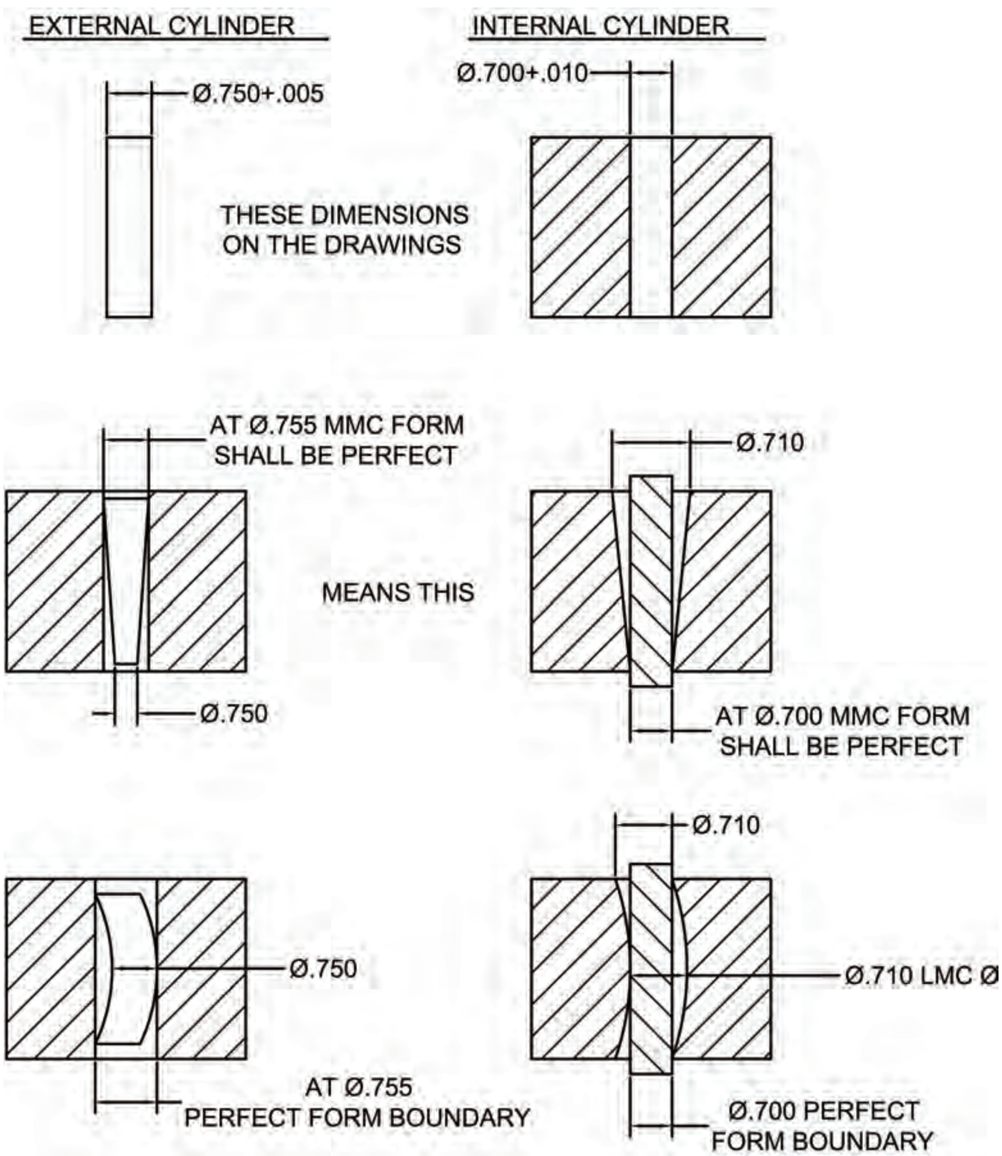

Figure 5-8 Rule #1.

5

7 GD&T Symbols/Meanings

Tolerance Type	Geometric Characteristics	Symbol	Applied To: Feature Surface	Applied To: Feature of Size Dim.	Datum Reference Required	Use Ⓛ or Ⓜ Material Condition	Gages Used
Form	Straightness +	⏤	YES	YES	NO	YES	YES***
	Flatness	⏥		NO		NO	NO
	Circularity	○					
	Cylindricity +	⌭					
Location	Positional Tolerance	⌖	NO	YES	YES	YES	YES***
	Concentricity	◎				NO	NO
	Symmetry +	⌯					
Orientation	Perpendicularity	⊥	YES	YES	YES	YES	YES***
	Parallelism	∥					
	Angularity	∠					
Profile	Profile of a Surface	⌓	YES	NO	YES*	YES**	NO
	Profile of a Line	⌒					
Runout	Circular Runout +	↗	YES	YES	YES	NO	NO
	Total Runout +	⌰					

+ Not typically used for die cast components ** Can be used to control form without a datum reference*
*** Datum reference only* **** – Yes if Ⓜ is specified for the feature of size being controlled*
– No if Ⓢ or Ⓛ are specified for the feature of size being controlled

To improve manufacturability a designer should discuss GD&T with the die caster early in the product design cycle.

8 Material Conditions

Features of size which includes datum features have size tolerances hence the size condition or material (amount of metal) condition can vary from the maximum metal condition (MMC) to the least metal condition (LMC). Consequently if the center planes or axes of a feature of size are controlled by geometric tolerances a modifying symbol can be specified in the feature control frame that applies the tolerance value at either the maximum or the least material condition. It also can be specified for a datum that is a feature of size. If a symbol is not specified the tolerance value applies regardless of material condition which is named regardless of feature size (RFS).

8.1 Maximum Material Condition (MMC)

This is the condition when the actual mating size or envelope size is at the maximum material condition which is maximum size for an external feature such as a cylinder and the minimum size for an internal feature such as a hole. Another way to look at MMC is that it always allows components to be assembled. In die casting MMC is most applicable to hole positions that will be machined. The symbol is Ⓜ. The tolerance value specified for the feature being controlled in the FCF applies only if the actual mating envelope is the MMC size. If the actual mating envelope deviates from MMC an additional tolerance is allowed. The added tolerance is the difference between the actual mating envelope size and the MMC size hence the largest actual mating envelope named virtual condition is equal to the MMC size plus the tolerance specified in the FCF for an external feature and minus for an internal feature. The MMC symbol is used to assure that parts will assemble and it allows the use of so called hard gages (go gages) for quick inspections. An example of position with MMC is shown in Fig. 5-9. It should be noted that actual local size has to meet the size tolerance however the actual local size does not affect the geometric characteristic tolerance.