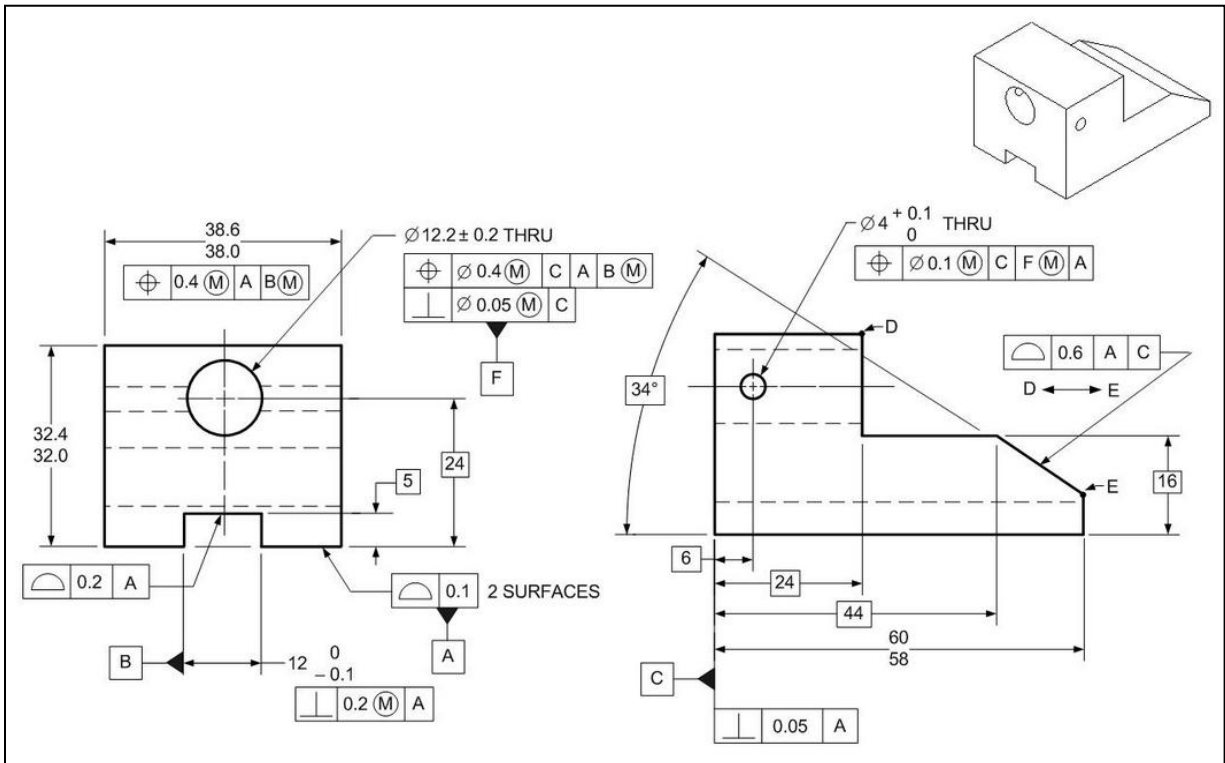


## An Introduction to Geometric Dimensioning and Tolerancing (GD&T)

Michael Yount  
 Proof Engineering Co.

### Table of Contents

1	Introduction.....	2
2	Datum Locating Principles .....	2
3.	Limits of Size.....	6
4	GD&T Tolerances.....	7
5.	Conclusion .....	14



# 1 Introduction

Since the first engineering drawing existed, so have manufacturing tolerances. Tolerances allow parts to deviate from perfection, but only within defined limits. The amount of tolerance allowed is usually based on part function. The limits allow the part to deviate, but a properly applied tolerance will ensure that parts fit properly and function as intended. The goal is to achieve a balance between high cost, narrow tolerances and lower cost, wide tolerances.

When tolerances were first introduced, they were simple; every dimension had a +/- tolerance. If the drawing dimension stated: 2.00" +/- .010" then an acceptable part would measure between 1.990" to 2.010" for that dimension. As engineering progressed and parts became more complicated, a new method of implementing tolerances was created; Geometric Dimensioning and Tolerancing, or GD&T.

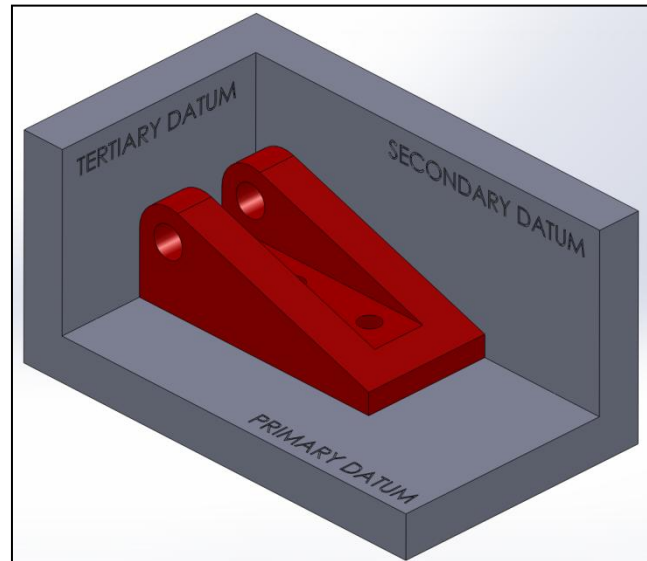
GD&T allows for comprehensive and consistent tolerances with the use of relatively simple tools. A part drawing may include a single GD&T callout, or the drawing may be fully defined using GD&T depending on part requirements. As with all new systems, there is a learning curve with GD&T. For this reason many fabricators and machine shops may not be familiar with GD&T specifics. Therefore it is recommended that you work with your vendors to determine their level of expertise with GD&T prior to adding it to a drawing.

## 2 Datum Locating Principles

In order to provide a foundation for repeatedly locating parts in the most consistent way, a basic understanding of GD&T locating datums is helpful. Here are some of the advantages gained by using GD&T over conventional dimensions/tolerances:

- Provides a concise way to describe a reference coordinate system (datums) of a component or assembly to be used throughout the manufacturing and inspection processes.
- Reduces the amount of notes, dimensions, and tolerances on a drawing; quickly convey design, manufacturing and inspection intent.
- With features like maximum material condition (MMC), more manufacturing tolerance can be allowed while still ensuring proper component function.

Figure 1 - Datums are used to create a reference coordinate system for repeatable part locators



Conventionally, datum A is the primary, B is the secondary, and C is the tertiary datum. In this picture it appears that each datum is a planer surface, but this is not always the case. Technically, it is never the case. This is because a perfectly planer surface is theoretical and can never be achieved in practice. Once this premise is accepted, a deeper understanding of locating principles can be achieved.

Three points define a plane, two points define a line. After the primary datum is located using three points, the secondary datum will typically use two points to define a line. The only degree of freedom remaining can be constrained by a single point on the tertiary (third) datum. Therefore in a standard datuming scheme, 6 points fully locate a part or assembly; 3 for the primary, 2 for the secondary, and 1 for the tertiary datums.

**DATUM A** - Take a look at the primary locating datum which is shown as a plane. Let's imagine this plane as a nice flat granite surface. Once the part is placed on the granite surface, only three points of it will be actually contacting the granite. Remembering that no surface can be perfectly flat (the granite nor the part), means only the lowest three points of the part will be touching the highest three points of the granite where these intersect. Since no two parts are identical these three exact points that touch the granite, and therefore locate the part, will not be the same from part to part.

If necessary, there are ways to control what locations on the parts surface are used to create the reference coordinate system. Instead of using a planer surface as the locating feature, three distinct locators can be used. By using three 'point' locators, the same three areas of every part are used to locate it. This can reduce the amount of locating tolerance, and improve down-stream fit-up or function depending on where the point locators are placed. Three 'point' locating targets are used to create the primary datum for a component Fig.2 & 3.

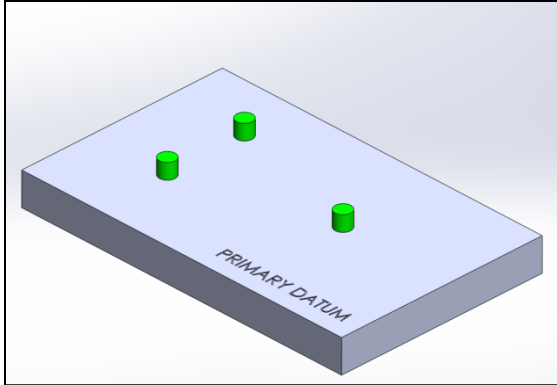


Figure 2 Datum A; Three point locators shown

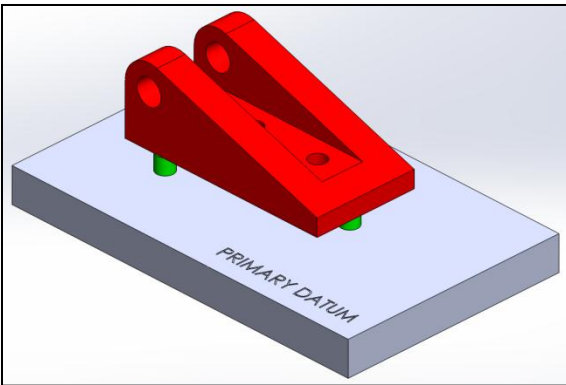


Figure 3 ...with part shown

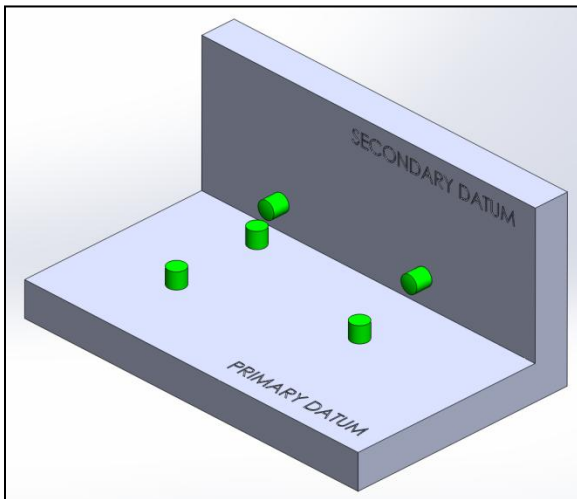


Figure 4 Datum B; Two point locators shown

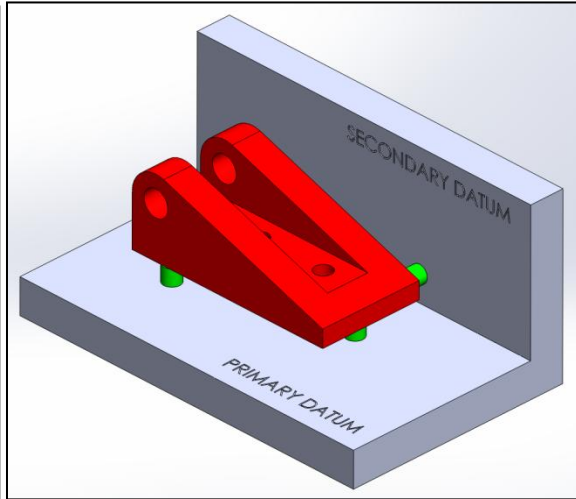


Figure 5 ...with part shown

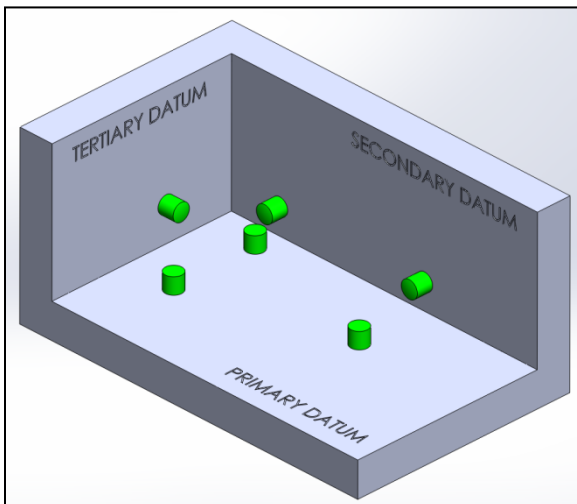


Figure 6 Datum C; One point locator shown

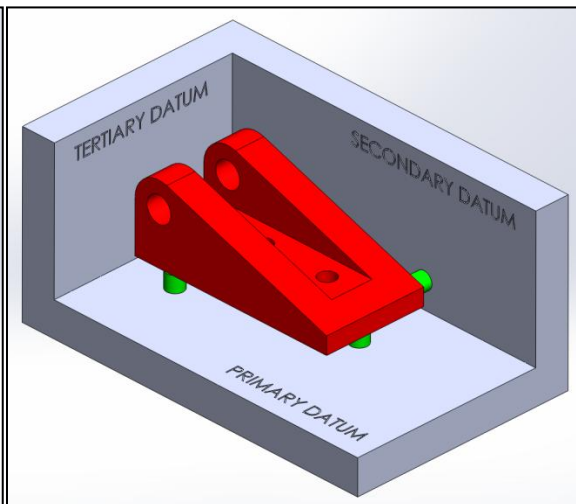


Figure 7 ...with part shown

The locators are considered points because the top of the green locators are spherically shaped, so the center is the highest point. Once the B and C datums are added, the primary datum plane formed by this fixture will repeatedly locate off the same three points on the parts. Instead of relying on the



### 3. Limits of Size





SYMBOL	GEOMETRIC CHARACTERISTIC	TYPE OF TOLERANCE	PRIMARY CONTROL
	FLATNESS	Form	Controls form (shape) of size and non-size features.
	STRAIGHTNESS		Datum reference is not allowed
	CYLINDRICITY	No relation between features	Controls form (shape) of size features
	CIRCULARITY (ROUNDNESS)		Datum reference is not allowed

Figure 9 - GD&T Limits of Size

Many different GD&T tolerances may be used to control a feature's position or location; these will include a reference to a Datum in the feature control frame. **Limits of size** however do not relate the feature's size in question to a location or position, so no Datum reference is needed. This is how GD&T can be used minimally on a drawing. With only limit of size feature control frames on a drawing, no Datums would be required. But having even one positional GD&T tolerance on a drawing will necessitate Datums. Control dimensions in GD&T are slightly different; a rectangular box around the dimension means it is a **basic dimension**. Basic dimensions do not have conventional tolerances, instead they use *feature control frames* (fig.12) to control the tolerance. The box around the basic dimension serves as a visual cue to search for the tolerance in a feature control frame.

#### FEATURE CONTROL FRAMES

A feature control frame (FCF) is the name of a GD&T tolerance symbol used on a drawing. A sample drawing that includes two feature control frames is shown in the figure below. The upper FCF denotes a flatness tolerance of .005". Since it is a limit of size, the FCF does not refer to any Datums. **Important distinction**; all tolerance values shown in FCF's are a total tolerance, not a plus/minus value. So in this example, you could think of it as a +/- .0025" tolerance of flatness. This tolerance only controls how flat this surface is. Tolerances of Location must be used to control where it is located or how much it is tilted relative to the rest of the part.

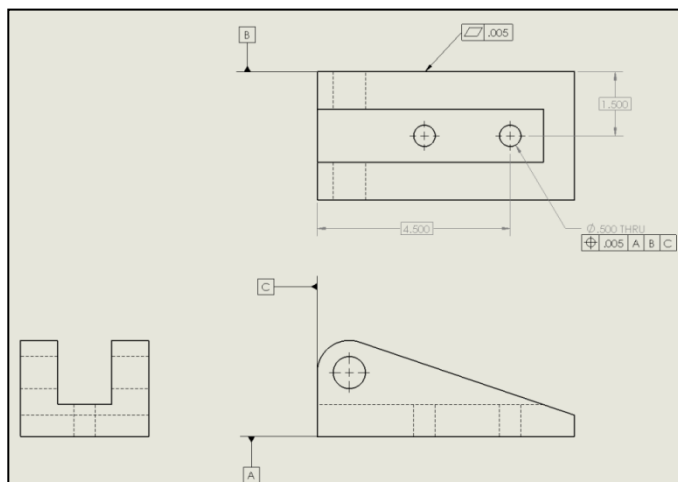


Figure 10 Drawing showing GD&T Feature Control Frames (FCF) and basic dims

## 4 GD&T Tolerances

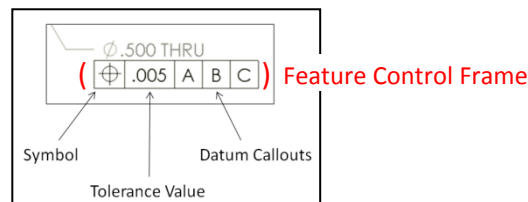
Unlike limits of size, tolerances of location need to reference at least one Datum plane, usually three. An example of this can be seen in Figure 10. The lower FCF includes a reference to three separate Datums. The 'target circle' symbol is named position, and is usually used locating for holes. Figure 11 shows a list of the tolerances of location:

SYMBOL	GEOMETRIC CHARACTERISTIC	TYPE OF TOLERANCE	PRIMARY CONTROL
$\perp$	PERPENDICULARITY	Orientation No relation between features	Controls orientation (tilt) of surfaces, axes, or median planes for size and non-size features Datum reference required <i>Optional: Angularity symbol may be used for all orientation controls</i>
//	PARALLELISM		
$\sphericalangle$	ANGULARITY		
$\oplus$	POSITION	Location	Locates center points, axes and median planes for size features. Can also control orientation. Locates surfaces Can also be used to control size, form, and orientation of surfaces based on datum reference
$\frac{1}{2}$	PROFILE OF A SURFACE		
$\frown$	PROFILE OF A LINE		
$\nearrow$	TOTAL RUNOUT	Runout	Controls surface coaxiality Can also control form and orientation of surfaces.
$\nearrow$	CIRCULAR RUNOUT		
$\odot$	CONCENTRICITY	Location of derived median points.	Locates derived median points of a feature <i>Not common, consider position, runout, or profile.</i>
$\equiv$	SYMMETRY		

Figure 11 GD&T Tolerances of Location

Regarding the position tolerance shown in Fig 10, notice that the two linear dimensions that locate the hole have rectangular boxes around them and are therefore basic dimensions. Basic dimensions are considered theoretically exact dimensions; chained basic dimensions do not create tolerance stacks. Again, basic dimensions have their tolerances in a Feature Control Frame. Let's take a closer look at the position FCF shown in Fig 10.

Figure 12 Position FCF with labels - from Fig 10



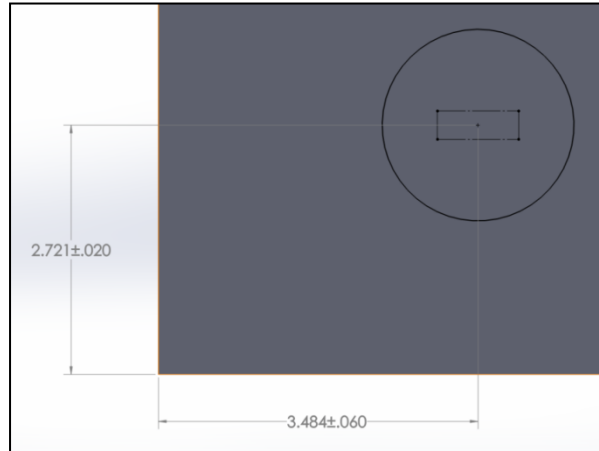
The first box in a Feature Control Frame contains an identifying symbol, position in this case. The second box of a FCF contains the *total* tolerance value. **Important distinction**; all tolerance values shown in FCF's are a **total** tolerance, not a plus/minus value. The Datum callouts start with the third box and continue until there are no more Datums to reference.

Notice that Datum A references a plane that is perpendicular to the hole's axis. This is standard practice and this Datum reference controls the direction of the hole through the material, usually perpendicular. The next two Datum references provide tolerances to the two hole locating dimensions. So to verbalize the Feature Control Frame shown in Fig 12: The hole's position must fall within a total tolerance zone of .005" relative to Datums A, B & C. The Datum order does matter, to inspect properly the part must be located on Datum A first, then Datum B, and finally Datum C.

## TOLERANCE ZONE SHAPE

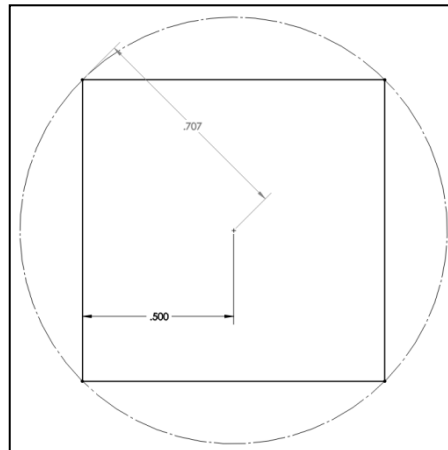
Another difference between standard +/- tolerances and GD&T is the shape of the tolerance zone. In conventional tolerances with a +/- tolerance in two right-angle dimensions (two dimensions to a hole for instance), the tolerance zone is rectangular shaped. This is shown below in Fig 13. If both hole location tolerances are equal, then the zone will be square shaped. The phantom line rectangular box in Fig 13 represents the area that the circle's center may fall, and still remain in tolerance.

Figure 13 Rectangular shaped tolerance zone for conventional



Unlike conventional tolerances with rectangular or square shaped zones, GD&T tolerance zones are circular in shape. When the tolerance value is .005" in a Feature Control Frame, the circle's center may move within a circular area with a **diameter** equal to .005". And the *tolerance* circle's center is at the intersection of the basic dimensions. This is a subtle difference but it should be acknowledged, because the same part that fails with conventional tolerances may pass with GD&T tolerances. The figure below illustrates this issue.

Figure 14 Tolerance zones; Conventional is solid & square vs. GD&T is phantom & round

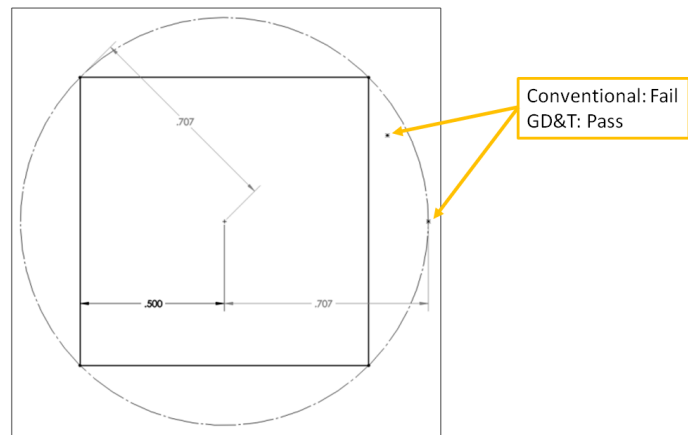


Notice that a square tolerance zone of +/- .500" will allow a hole center location .707" away from the center if it is located in the squares' corner (diagonally). Remembering that a GD&T tolerance value is *total*, using a GD&T tolerance value of 1.414" in this case will allow all hole center locations to pass that



pass the conventional test. But using this GD&T callout will allow some hole center locations to pass that did not pass the conventional test. Below is an illustration of this effect.

Figure 15 Pass/Fail; Hole center locations that pass GD&T but not conventional



To correct this tolerance discrepancy it is better to err on the side of caution. To convert a +/- tolerance to a GD&T tolerance; multiply the +/- tolerance by 2. But to convert a GD&T tolerance to a +/- tolerance; divide the GD&T tolerance by 2 then multiply by .707. Most holes, pins, bolts, etc. are round, therefore the round tolerance zone of GD&T is a more logical shape.

### MODIFYING SYMBOLS

As a reminder; the first FCF box is a symbol, the second contains the tolerance value, and the third through fifth usually contain the Datum callouts. The second box of a Feature Control Frame may also contain a modifying symbol directly after the tolerance value. These are special symbols that can exist in any FCF box except the first. The first FCF box is reserved for *control symbols* already listed in either Fig 9 or Fig 11. Some of the *modifying symbols* are listed here:

- (M) MMC - or Maximum Material Condition
- (L) LMC - or Least Material Condition
- (S) RFS - or Regardless of Feature Size
- (P) Projected Tolerance Zone

MMC and Projected Tolerance Zone will be covered briefly because they are two of the more useful modifying symbols. MMC is valuable for allowing more part variance during fabrication while ensuring that the parts will always assemble properly. Projected Tolerance Zone is used to control the location of a hole *beyond* the parts' physical edges.

## MAX MATERIAL CONDITION (MMC) DEFINITION, PRACTICE

Manufacturing tolerances are used when individual parts are made. When multiple parts must fit together to make an assembly, these individual part tolerances must be properly controlled. If they are not, then the part may not fit into the assembly and will need to be reworked or discarded. In order to maximize the allowable manufacturing variability while ensuring proper fit up, MMC may be used.

Maximum Material Condition, or MMC, is as it sounds - the most material. Consider a bolt going through a hole. The MMC bolt is the largest diameter it can be within its tolerance. However, a MMC hole is the smallest hole, which also represents the most material. By including the MMC modifying symbol in a feature control frame, the size of the hole (diameter) now comes into play when determining *positional* tolerance. Let's take a look at an example.

Figure 16 - Two plate assembly; How to allow more manufacturing tolerance in the TOP RED PLATE

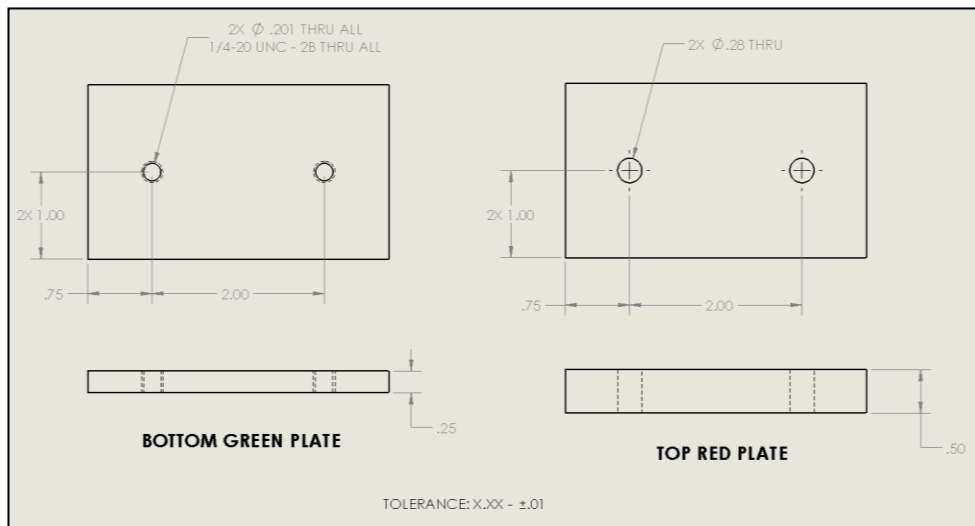
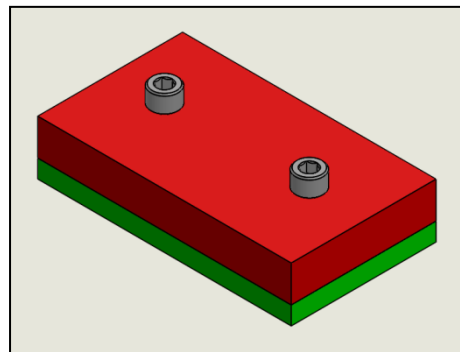


Figure 17 - Conventional Dimensions and Tolerances - No GD&T

The drawing shown above represents a conventional way to dimension and tolerance the parts.  $\pm 0.01$ " for hole positions in the BOTTOM GREEN PLATE, along with  $\pm 0.01$ " for hole positions in the TOP RED PLATE means the clearance hole diameter needs to be minimally 0.02" oversized. A 1/4-20 bolt is .25 max diameter + .02 equals .27. The hole diameter is .28  $\pm .01$ , or .27 at the smallest. If these parts are built within tolerance they will always assemble.

Notice that these parts will assemble even when the holes are at MMC, or the smallest hole. If both of the holes in the TOP RED PLATE were made at the large end of the tolerance,  $\varnothing.29$ , there would then be extra clearance. This is the principle behind allowing extra manufacturing tolerance with MMC, by making the hole larger its position can drift more.

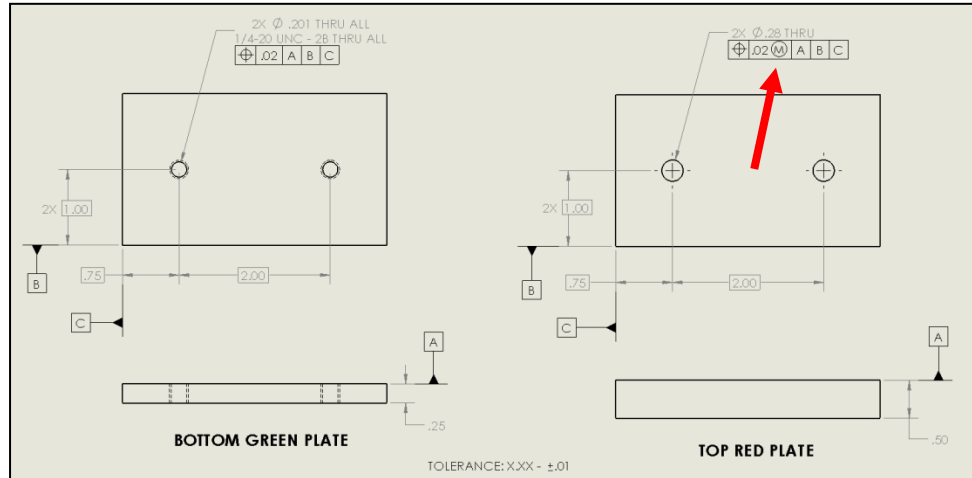


Figure 18 - GD&T VERSION OF FIG 17, NOTE THE CIRCLED M IN THE SECOND FCF

The drawing shown above represents the GD&T version, with MMC for the two holes in the TOP RED PLATE. Similar tolerances are used for the hole's position and diameter as was used in Figure 17, the difference being the circled M in the TOP RED PLATE GD&T Feature Control Frame. This circled M means MMC applies to these two holes.

Since a threaded hole will follow its own strict size and tolerance definitions, MMC is not to be used on threaded or tapped holes, or male threaded parts.

Follow these steps to verify proper MMC usage:

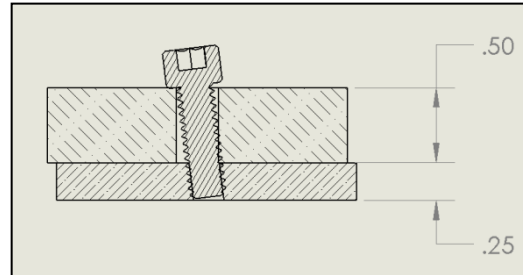
1. Determine all of the holes' diameters at MMC.
2. With all of the holes at MMC (smallest), no *additional* positional tolerance is permitted.
3. For every .001" a hole is over MMC, it is allowed .001" of additional position tolerance.
4. Step #3 is valid up to LMC (Least Material Condition), or the largest hole within tolerance.

With MMC on a drawing the manufacturer has a choice. Make the holes smaller in diameter (within tolerance) and get little-to-no *extra* positional tolerance, or make the holes larger in diameter and get additional position tolerance. If MMC is applied and followed properly then the mating parts will always assemble even though extra positional tolerance may have been allowed during fabrication.

This example uses only two holes, but these same MMC principles apply to any bolt pattern be it square, rectangular, or round.

## PROJECTED TOLERANCE ZONE

Figure 19 - A bolt will not assemble through a thick plate if the threaded hole is on too much of an angle



Fastening two plates together may require that a bolt fit through a clamped plate hole and engage a threaded hole in the base plate. When the clamped plate has significant thickness, the angle of the threaded holes in the base plate should be further limited to make sure assembly is possible. Projected Tolerance Zones were created for this type of situation. Instead of controlling the threaded hole location and angle only *inside* the material, a Projected Tolerance Zone controls the (threaded in this case) hole axis parameters for a given distance *outside* the material. This eliminates manually calculating the required angle and adding the two conventional angle dimensions and tolerances. Let's take a look at an example, starting with a typical GD&T call out *without* the projected tolerance zone modifying symbol.

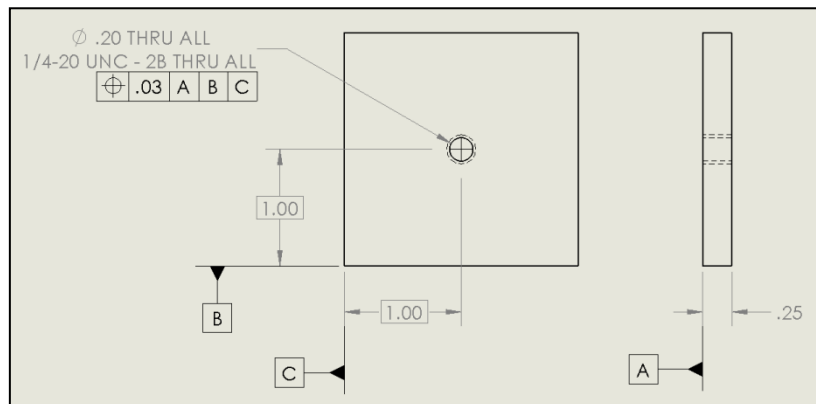


Figure 20 - GD&T used to control the threaded hole

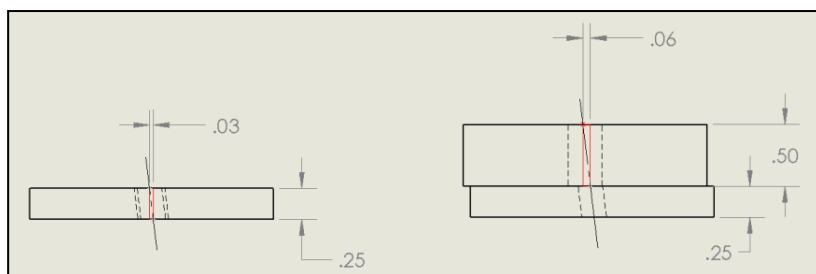


Figure 21 - Visual description for the angular tolerances shown in Fig 20

Above; the 0.03" tolerance applies *within* the material

Above; the same 0.03" tolerance and the same angle, but extended upward through a thicker plate means a larger lineal offset.

By applying a projected tolerance zone modifying symbol to the feature control frame this problem can be easily managed. Below is a copy of the drawing in Fig 20, with an additional tolerance control.

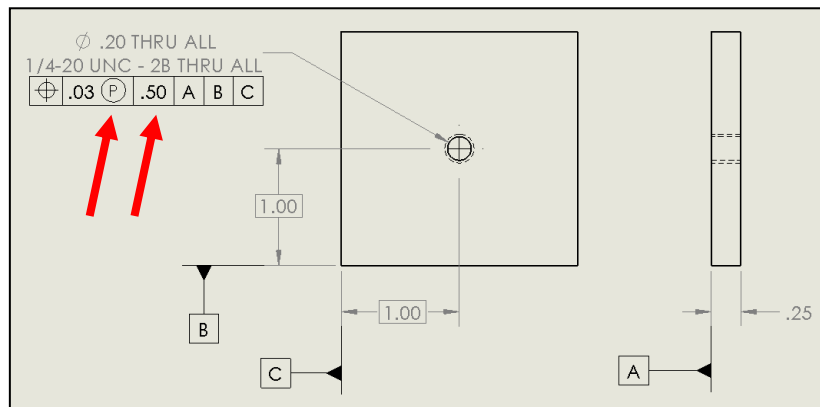


Figure 22 - GD&T used to control the threaded hole using a projected tolerance zone

Notice the circled P in the FCF, followed by the projected distance. The projected distance in this case (.50") matches the clamped plate thickness. This will control the hole's position and angle for .50" outside the actual material.

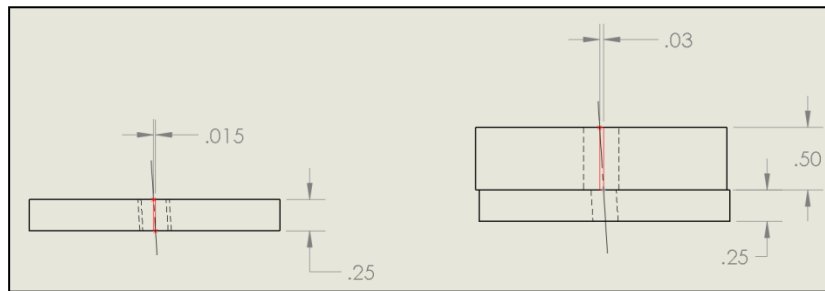


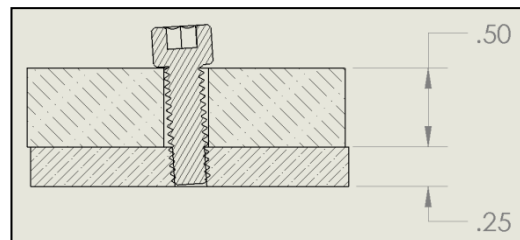
Figure 23 -Visual description for the angular tolerances shown in Fig 22

Above; there is no tolerance to control the angle *inside* the material.

Above; the stated 0.03" tolerance applies *outside* the material projected for a distance of .50".

The projected tolerance zone is not only useful for bolts and tapped holes. It can be used for dowels, cam followers, bearing holes, and other similar situations.

Figure 24 - A bolt will assemble through a thick plate if a projected tolerance zone is properly applied.



## 5. Conclusion

Tolerances are a critical part of manufacturing that require consideration. Conventional tolerances are straightforward, easy to use and understand, and are simple. Simple tolerances work well for simple parts. Other parts can be very challenging to tolerance in a conventional way, for instance, those having complex surfaces or elaborate mating faces. Yet others may have mounting hole patterns or several inter-related features that demand tighter control.

GD&T is the modern language of engineering drawings. Its usefulness goes far beyond the topics discussed in this paper. It is more involved than conventional tolerances but tends to be more concise and powerful. The proper use of GD&T can save money and time in manufacturing while at the same time improving product yield and quality. This win-win scenario is one of the reasons most large companies have shifted toward the use of GD&T in their engineering drawings.

Drawings may use conventional tolerances only, GD&T only, or may use a hybrid of conventional and GD&T tolerances. Employ the simple tolerances where useful, and then control more critical features with GD&T all on the same drawing. As with all things, the more you use it, the better you will know it.

### **For further reading:**

Geometric Dimensioning and Tolerancing, ASME Y14.5-2009  
by James D. Meadows