

Geometric Dimensioning and Tolerancing

Walter M. Stites
Paul Drake

Walter M. Stites
AccraTronics Seals Corp.
Burbank, California

Walter M. Stites is a graduate of California State University, Northridge. His 20-year tenure at AccraTronics Seals Corp began with six years in the machine shop, where he performed every task from operating a hand drill press to making tools and fixtures. Trained in coordinate measuring machine (CMM) programming in 1983, he has since written more than 1,000 CMM programs. He also performs product design, manufacturing engineering, and drafting. In 12 years of computer-assisted drafting, he's generated more than 800 engineering drawings, most employing GD&T. He has written various manuals, technical reports, and articles for journals. Mr. Stites is currently secretary of the ASME Y14.5 subcommittee and a key player in the ongoing development of national drafting standards.

5.1 Introducing Geometric Dimensioning and Tolerancing (GD&T)

When a hobbyist needs a simple part for a project, he might go straight to the little lathe or milling machine in his garage and produce it in a matter of minutes. Since he is designer, manufacturer, and inspector all in one, he doesn't need a drawing. In most commercial manufacturing, however, the designer(s), manufacturer(s), and inspector(s) are rarely the same person, and may even work at different companies, performing their respective tasks weeks or even years apart.

A designer often starts by creating an ideal assembly, where all the parts fit together with optimal tightnesses and clearances. He will have to convey to each part's manufacturer the ideal sizes and shapes, or *nominal dimensions* of all the part's surfaces. If multiple copies of a part will be made, the designer must recognize it's impossible to make them all identical. Every manufacturing process has unavoidable variations that impart corresponding variations to the manufactured parts. The designer must analyze his entire assembly and assess for each surface of each part how much variation can be allowed in size, form,

orientation, and location. Then, in addition to the ideal part geometry, he must communicate to the manufacturer the calculated magnitude of variation or *tolerance* each characteristic can have and still contribute to a workable assembly.

For all this needed communication, words are usually inadequate. For example, a note on the drawing saying, “Make this surface real flat,” only has meaning where all concerned parties can do the following:

- Understand English
- Understand to which surface the note applies, and the extent of the surface
- Agree on what “flat” means
- Agree on exactly how flat is “real flat”

Throughout the twentieth century, a specialized language based on graphical representations and math has evolved to improve communication. In its current form, the language is recognized throughout the world as *Geometric Dimensioning and Tolerancing (GD&T)*.

5.1.1 What Is GD&T?

Geometric Dimensioning and Tolerancing (GD&T) is a language for communicating engineering design specifications. GD&T includes all the symbols, definitions, mathematical formulae, and application rules necessary to embody a viable engineering language. As its name implies, it conveys both the nominal dimensions (ideal geometry), and the tolerances for a part. Since GD&T is expressed using line drawings, symbols, and Arabic numerals, people everywhere can read, write, and understand it regardless of their native tongues. It’s now the predominant language used worldwide as well as the standard language approved by the American Society of Mechanical Engineers (ASME), the American National Standards Institute (ANSI), and the United States Department of Defense (DoD).

It’s equally important to understand what GD&T is not. It is not a creative design tool; it cannot suggest how certain part surfaces should be controlled. It cannot communicate design intent or any information about a part’s intended function. For example, a designer may intend that a particular bore function as a hydraulic cylinder bore. He may intend for a piston to be inserted, sealed with two Buna-N O-rings having .010" squeeze. He may be worried that his cylinder wall is too thin for the 15,000-psi pressure. GD&T conveys none of this. Instead, it’s the designer’s responsibility to translate his hopes and fears for his bore—his intentions—into unambiguous and measurable specifications. Such specifications may address the size, form, orientation, location, and/or smoothness of this cylindrical part surface as he deems necessary, based on stress and fit calculations and his experience. It’s these objective specifications that GD&T codifies. Far from revealing what the designer has in mind, GD&T cannot even convey that the bore is a hydraulic cylinder, which gives rise to the Machinist’s Motto.

*Mine is not to reason why;
Mine is but to tool and die.*

Finally, GD&T can only express what a surface shall be. It’s incapable of specifying manufacturing processes for making it so. Likewise, there is no vocabulary in GD&T for specifying inspection or gaging methods. To summarize, GD&T is the language that designers use to translate design requirements into measurable specifications.

5.1.2 Where Does GD&T Come From?—References

The following American National Standards define GD&T’s vocabulary and provide its grammatical rules.

- ASME Y14.5M-1994, Dimensioning and Tolerancing
- ASME Y14.5.1M-1994, Mathematical Definition of Dimensioning and Tolerancing Principles

Hereafter, to avoid confusion, we'll refer to these as "Y14.5" and "the Math Standard," respectively (and respectfully). The more familiar document, Y14.5, presents the entire GD&T language in relatively plain English with illustrated examples. Throughout this chapter, direct quotations from Y14.5 will appear in boldface. The supplemental Math Standard expresses most of GD&T's principles in more precise math terminology and algebraic notation—a tough read for most laymen. For help with it, see Chapter 7. Internationally, the multiple equivalent ISO standards for GD&T reveal only slight differences between ISO GD&T and the US dialect. For details, see Chapter 6.

Unfortunately, ASME offers no 800 number or hotline for Y14.5 technical assistance. Unlike computer software, the American National and ISO Standards are strictly rulebooks. Thus, in many cases, for ASME to issue an interpretation would be to arbitrate a dispute. This could have far-reaching legal consequences. Your best source for answers and advice are textbooks and handbooks such as this. As members of various ASME and ISO standards committees, the authors of this handbook are brimming with insights, experiences, interpretations, preferences, and opinions. We'll try to sort out the few useful ones and share them with you. In shadowboxes throughout this chapter, we'll concoct FAQs (frequently asked questions) to ourselves. Bear in mind, our answers reflect our own opinions, not necessarily those of ASME or any of its committees.

In this chapter, we've taken a very progressive approach toward restructuring the explanations and even the concepts of GD&T. We have solidified terminology, and stripped away redundancy. We've tried to take each principle to its logical conclusion, filling holes along the way and leaving no ambiguities. As you become more familiar with the standards and this chapter, you'll become more aware of our emphasis on practices and methodologies consistent with state-of-the-art manufacturing and high-resolution metrology.

FAQ: *I notice Y14.5 explains one type of tolerance in a single paragraph, but devotes pages and pages to another type. Does that suggest how frequently each should be used?*

A: No. There are some exotic principles that Y14.5 tries to downplay with scant coverage, but mostly, budgeting is based on a principle's complexity. That's particularly true of this handbook. We couldn't get by with a brief and vague explanation of a difficult concept just because it doesn't come up very often. Other supposed indicators, such as what questions show up on the Certification of GD&T Professionals exam, might be equally unreliable. Throughout this chapter, we'll share our preferences for which types of feature controls to use in various applications.

FAQ: *A drawing checker rejected one of my drawings because I used a composite feature control frame having three stacked segments. Is it OK to create GD&T applications not shown in Y14.5?*

A: Yes. Since the standards can neither discuss nor illustrate every imaginable application of GD&T, questions often arise as to whether or not a particular application, such as that shown in Fig. 5-127, is proper. Just as in matters of law, some of these questions can confound the experts. Clearly, if an illustration in the standard bears an uncanny resemblance to your own part, you'll be on pretty solid ground in copying that application. Just as often, however, the standard makes no mention of your specific application. You are allowed to take the explicit rules and principles and extend them to your application in any way that's consistent with all the rules and principles stated in the standard. Or, more simply, any application that doesn't

violate anything in the standard is acceptable. That's good news for a master practitioner who's familiar with the whole standard. Throughout this chapter we'll try to help novices by including "extension of principle" advice where it's appropriate.

FAQ: *I've found what seem to be discrepancies between Y14.5 and the Math Standard. How can that be? Which standard supersedes?*

A: You're right. There are a couple of direct contradictions between the two standards. Like any contemporary "living" language, GD&T is constantly evolving to keep pace with our modern world and is consequently imperfect. For instance, Y14.5 has 232 pages while the Math Standard has just 82. You could scarcely expect them to cover the same material in perfect harmony. Yet there's no clue in either document as to which one supersedes (they were issued only eight days apart). Where such questions arise, we'll discuss the issues and offer our preference.

5.1.3 Why Do We Use GD&T?

When several people work with a part, it's important they all reckon part dimensions the same. In Fig. 5-1, the designer specifies the distance to a hole's ideal location; the manufacturer measures off this distance and ("X marks the spot") drills a hole; then an inspector measures the actual distance to that hole. All three parties must be in perfect agreement about three things: from where to start the measurement, what direction to go, and where the measurement ends.

As illustrated in Chapter 3, when measurements must be precise to the thousandth of an inch, the slightest difference in the origin or direction can spell the difference between a usable part and an expensive paperweight. Moreover, even if all parties agree to measure to the hole's center, a crooked, bowed, or egg-shaped hole presents a variety of "centers." Each center is defensible based on a different design consideration. GD&T provides the tools and rules to assure that all users will reckon each dimension the same, with perfect agreement as to origin, direction, and destination.

It's customary for GD&T textbooks to spin long-winded yarns explaining how GD&T affords more tolerance for manufacturing. By itself, it doesn't. GD&T affords however much or little tolerance the designer specifies. Just as ubiquitous is the claim that using GD&T saves money, but these claims are never accompanied by cost or Return on Investment (ROI) analyses. A much more fundamental reason for

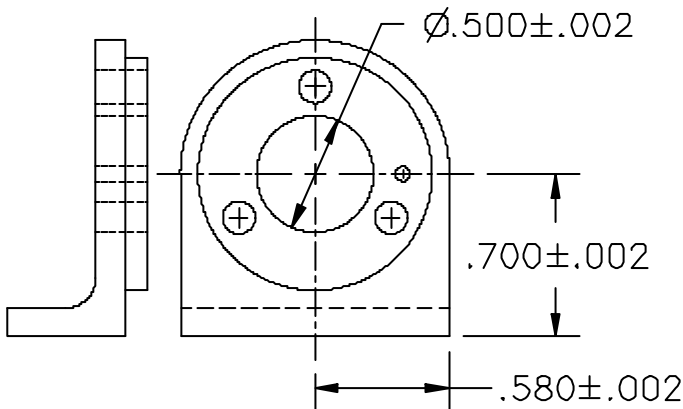


Figure 5-1 Drawing showing distance to ideal hole location

using GD&T is revealed in the following study of how two very different builders approach constructing a house.

A primitive builder might start by walking around the perimeter of the house, dragging a stick in the dirt to mark where walls will be. Next, he'll lay some long boards along the lines on the uneven ground. Then, he'll attach some vertical boards of varying lengths to the foundation. Before long, he'll have a framework erected, but it will be uneven, crooked, and wavy. Next, he'll start tying or tacking palm branches, pieces of corrugated aluminum, or discarded pieces of plywood to the crude frame. He'll overlap the edges of these flexible sidings 1-6 inches and everything will fit just fine. Before long, he'll have the serviceable shanty shown in Fig. 5-2, but with some definite limitations: no amenities such as windows, plumbing, electricity, heating, or air conditioning.

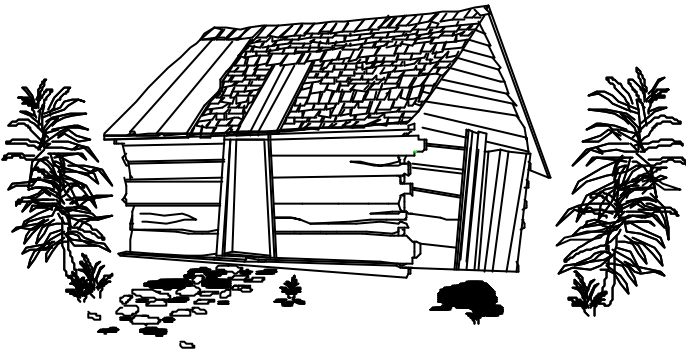


Figure 5-2 House built without all of the appropriate tools

A house having such modern conveniences as glass windows and satisfying safety codes requires more careful planning. Materials will have to be stronger and more rigid. Spaces inside walls will have to be provided to fit structural members, pipes, and ducts.

To build a house like the one shown in Fig. 5-3, a modern contractor begins by leveling the ground where the house will stand. Then a concrete slab or foundation is poured. The contractor will make the slab as level and flat as possible, with straight, parallel sides and square corners. He will select the straightest wooden plates, studs, headers, and joists available for framing and cut them to precisely uniform lengths. Then he'll use a large carpenter's square, level, and plumb bob to make each frame member parallel or perpendicular to the slab.

Why are such precision and squareness so important? Because it allows him to make accurate measurements of his work. Only by making accurate measurements can he assure that such prefabricated

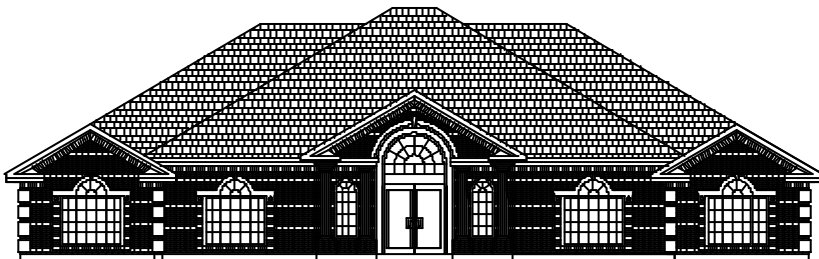


Figure 5-3 House built using the correct tools

items as Sheetrock, windows, bathtubs, and air conditioning ducts will fit in the spaces between his frame members. Good fits are important to conserve space and money. It also means that when electrical outlet boxes are nailed to the studs 12" up from the slab, they will all appear parallel and neatly aligned. Remember that it all derives from the flatness and squareness of the slab.

By now, readers with some prior knowledge of GD&T have made the connection: The house's concrete slab is its "primary datum." The slab's edges complete the "datum reference frame." The wooden framing corresponds to "tolerance zones" and "boundaries" that must contain "features" such as pipes, ducts, and windows.

Clearly, the need for precise form and orientation in the slab and framing of a house is driven by the fixtures to be used and how precisely they must fit into the framing. Likewise, the need for GD&T on a part is driven by the types and functions of its features, and how precisely they must relate to each other and/or fit with mating features of other parts in the assembly. The more complex the assembly and the tighter the fits, the greater are the role and advantages of GD&T.

Fig. 5-4 shows a non-GD&T drawing of an automobile wheel rotor. Despite its neat and uniform appearance, the drawing leaves many relationships between part features totally out of control. For example, what if it were important that the $\varnothing 5.50$ bore be perpendicular to the mounting face? Nothing on the drawing addresses that. What if it were critical that the $\varnothing 5.50$ bore and the $\varnothing 11.00$ OD be on the same axis? Nothing on the drawing requires that either. In fact, Fig. 5-5 shows the "shanty" that could be built. Although all its dimensions are within their tolerances, it seems improbable that any "fixtures" could fit it.

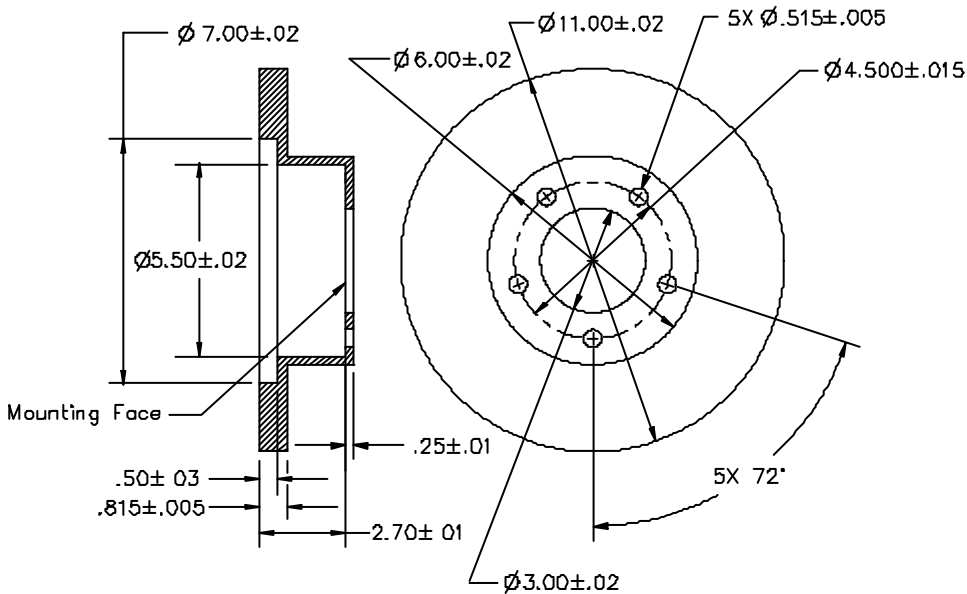


Figure 5-4 Drawing that does not use GD&T

In Fig. 5-6, we've applied GD&T controls to the same design. We've required the mounting face to be flat within $.005$ and then labeled it datum feature A. That makes it an excellent "slab" from which we can launch the rest of the part. Another critical face is explicitly required to be parallel to A within $.003$. The perpendicularity of the $\varnothing 5.50$ bore is directly controlled to our foundation, A. Now the $\varnothing 5.50$ bore can be labeled datum feature B and provide an unambiguous origin—a sturdy "center post"—from which the $\varnothing .515$ bolt holes and other round features are located. Datum features A and B provide a very uniform and well-aligned framework from which a variety of relationships and fits can be precisely controlled. Just as

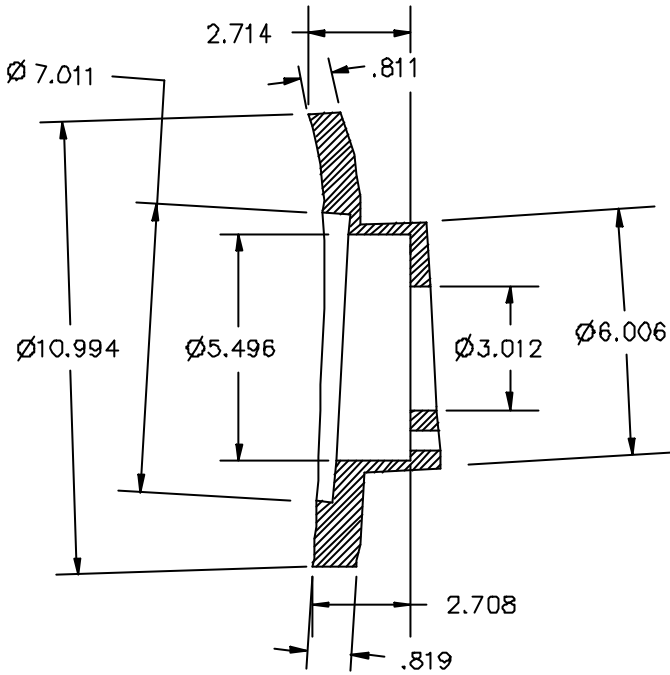


Figure 5-5 Manufactured part that conforms to the drawing without GD&T (Fig. 5-4)

importantly, GD&T provides unique, unambiguous meanings for each control, precluding each person's having his own competing interpretation. GD&T, then, is simply a means of controlling surfaces more precisely and unambiguously.

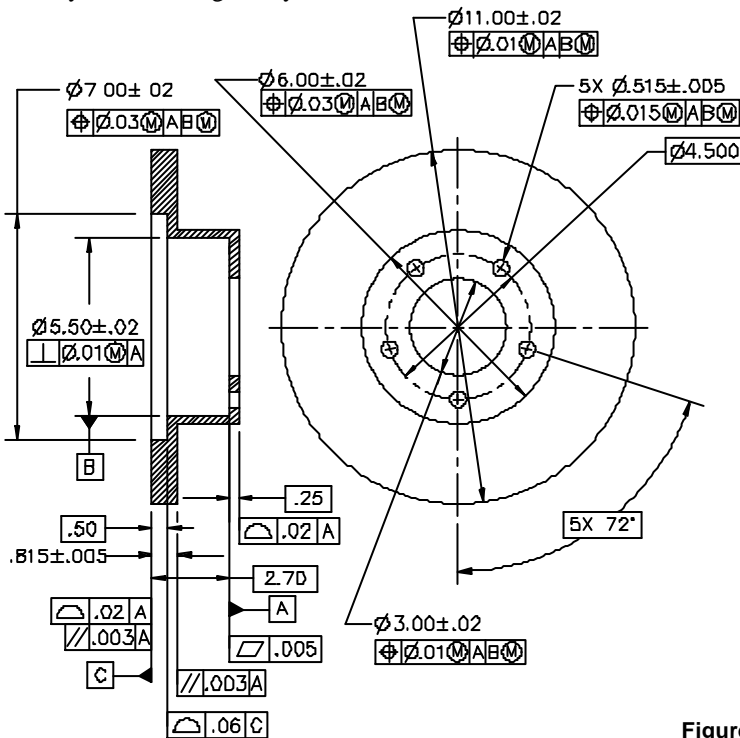


Figure 5-6 Drawing that uses GD&T

5-8 Chapter Five

And that's the fundamental reason for using GD&T. It's the universal language throughout the world for communicating engineering design specifications. Clear communication assures that manufactured parts will function and that functional parts won't later be rejected due to some misunderstanding. Fewer arguments. Less waste.

As far as that ROI analysis, most of the costs GD&T reduces are hidden, including the following:

- Programmers wasting time trying to interpret drawings and questioning the designers
- Rework of manufactured parts due to misunderstandings
- Inspectors spinning their wheels, deriving meaningless data from parts while failing to check critical relationships
- Handling and documentation of functional parts that are rejected
- Sorting, reworking, filing, shimming, etc., of parts in assembly, often in added operations
- Assemblies failing to operate, failure analysis, quality problems, customer complaints, loss of market share and customer loyalty
- The meetings, corrective actions, debates, drawing changes, and interdepartmental vendettas that result from each of the above failures

It all adds up to an enormous, yet unaccounted cost. Bottom line: use GD&T because it's the right thing to do, it's what people all over the world understand, and it saves money.

5.1.4 When Do We Use GD&T?

In the absence of GD&T specifications, a part's ability to satisfy design requirements depends largely on the following four "laws."

1. **Pride in workmanship.** Every industry has unwritten customary standards of product quality, and most workers strive to achieve them. But these standards are mainly minimal requirements, usually pertaining to cosmetic attributes. Further, workmanship customs of precision aerospace machinists are probably not shared by ironworkers.
2. **Common sense.** Experienced manufacturers develop a fairly reliable sense for what a part is supposed to do. Even without adequate specifications, a manufacturer will try to make a bore very straight and smooth, for example, if he suspects it's for a hydraulic cylinder.
3. **Probability.** Sales literature for modern machining centers often specifies repeatability within 2 microns (.00008"). Thus, the running gag in precision manufacturing is that part dimensions should never vary more than that. While the performance of a process can usually be predicted statistically, there are always "special causes" that introduce surprise variations. Further, there's no way to predict what processes might be used, how many, and in what sequence to manufacture a part.
4. **Title block, workmanship, or contractual ("boiler plate") standards.** Sometimes these provide clarification, but often, they're World War II vintage and inadequate for modern high-precision designs. An example is the common title block note, "All diameters to be concentric within .005."

Dependence on these four "laws" carries obvious risks. Where a designer deems the risks too high, specifications should be rigorously spelled out with GD&T.

FAQ: *Should I use GD&T on every drawing?*

A: Some very simple parts, such as a straight dowel, flat washer, or hex nut may not need GD&T. For such simple parts, Rule #1 (explained in section 5.6.3.1), which pertains to size limits, may provide adequate control by itself. However, some practitioners always use GD&T positional tolerancing for holes and width-type features (slots and tabs). It depends primarily on how much risk there is of a part being made, such as that shown in Fig. 5-5, which conforms to all the non-GD&T tolerances but is nevertheless unusable.

FAQ: *Can I use GD&T for just one or two selected surfaces on a drawing, or is it “all or nothing?”*

A: On any single drawing you can mix and match all the dimensioning and tolerancing methods in Y14.5. For example, one pattern of holes may be controlled with composite positional tolerance while other patterns may be shown using coordinate dimensions with plus and minus tolerances. Again, it depends on the level of control needed. But, if you choose GD&T for any individual feature or pattern of features, you must give that feature the full treatment. For example, you shouldn't dimension a hole with positional tolerance in the X-axis, and plus and minus tolerance in the Y-axis. Be consistent. Also, it's a good idea to control the form and orientational relationships of surfaces you're using as datum features.

FAQ: *Could GD&T be used on the drawings for a house?*

A: Hmmm. Which do you need, shanty or chateau?

5.1.5 How Does GD&T Work?—Overview

In the foregoing paragraphs, we alluded to the goal of GD&T: to guide all parties toward reckoning part dimensions the same, including the origin, direction, and destination for each measurement. GD&T achieves this goal through four simple and obvious steps.

1. Identify part surfaces to serve as origins and provide specific rules explaining how these surfaces establish the starting point and direction for measurements.
2. Convey the nominal (ideal) distances and orientations from origins to other surfaces.
3. Establish boundaries and/or tolerance zones for specific attributes of each surface along with specific rules for conformance.
4. Allow dynamic interaction between tolerances (simulating actual assembly possibilities) where appropriate to maximize tolerances.

5.2 Part Features

Up to this point, we've used the terms *surface* and *feature* loosely and almost interchangeably. To speak GD&T, however, we must begin to use the vocabulary as Y14.5 does.

***Feature* is the general term applied to a physical portion of a part, such as a surface, pin, tab, hole, or slot.**

Usually, a part feature is a single surface (or a pair of opposed parallel plane surfaces) having uniform shape. You can establish datums from, and apply GD&T controls to features only. The definition implies that no feature exists until a part is actually produced. There are two general types of features: those that have a built-in dimension of “size,” and those that don't.

FAQ: *Is a center line a feature?*

A: No, since a center line or center plane can never be a physical portion of a part.

FAQ: *Well, what about a nick or a burr? They're "physical portions of a part," right?*

A: True, but Y14.5 doesn't mean to include nicks and burrs as features. That's why we've added "having uniform shape" to our own description.

FAQ: *With transitions at tangent radii or slight angles, how can I tell exactly where one feature ends and the adjacent feature begins?*

A: You can't. The Math Standard points out, "Generally, features are well defined only in drawings and computer models." Therefore, you are free to reckon the border between features at any single location that satisfies all pertinent tolerances.

5.2.1 Nonsize Features

A *nonsize feature* is a surface having no unique or intrinsic size (diameter or width) dimension to measure. Nonsize features include the following:

- A nominally flat planar surface
- An irregular or "warped" planar surface, such as the face of a windshield or airfoil
- A *radius*—a portion of a cylindrical surface encompassing less than 180° of arc length
- A *spherical radius*—a portion of a spherical surface encompassing less than 180° of arc length
- A *revolute*—a surface, such as a cone, generated by revolving a spine about an axis

5.2.2 Features of Size

A feature of size is one cylindrical or spherical surface, or a set of two opposed elements or opposed parallel surfaces, associated with a size dimension.

A feature of size has opposing points that partly or completely enclose a space, giving the feature an intrinsic dimension—size—that can be measured apart from other features. Holes are "internal" features of size and pins are "external" features of size. Features of size are subject to the principles of material condition modifiers, as we'll explain in section 5.6.2.1.

"Opposed parallel surfaces" means the surfaces are designed to be parallel to each other. To qualify as "opposed," it must be possible to construct a perpendicular line intersecting both surfaces. Only then, can we make a meaningful measurement of the size between them. From now on, we'll call this type of feature a *width-type feature*.

FAQ: *Where a bore is bisected by a groove, is the bore still considered a single feature of size, or are there two distinct bores?*

A: A similar question arises wherever a boss, slot, groove, flange, or step separates any two otherwise continuous surfaces. A specification preceded by 2X clearly denotes two distinct features. Conversely, Y14.5 provides no symbol for linking interrupted surfaces. For example, an extension line that connects two surfaces by bridging across an interruption has no standardized meaning. Where a single feature control shall apply to all portions of an interrupted surface, a note, such as TWO SURFACES AS A SINGLE FEATURE , should accompany the specification.

5.2.2.1 Screw Threads

A screw thread is a group of complex helical surfaces that can't directly be reckoned with as a feature of size. However, the abstract *pitch cylinder* derived from the thread's flanks best represents the thread's functional axis in most assemblies. Therefore, by default, the pitch cylinder “stands in” for the thread as a datum feature of size and/or as a feature of size to be controlled with an orientation or positional tolerance. The designer may add a notation specifying a different abstract feature of the thread (such as MAJOR DIA, or MINOR DIA). This notation is placed beneath the feature control frame or beneath or adjacent to the “datum feature” symbol, as applicable.

FAQ: *For a tapped hole, isn't it simpler just to specify the minor diameter?*

A: Simpler, yes. But it's usually a mistake, because the pitch cylinder can be quite skewed to the minor diameter. The fastener, of course, will tend to align itself to the pitch cylinder. We've seen projected tolerance zone applications where parts would not assemble despite the minor diameters easily conforming to the applicable positional tolerances.

5.2.2.2 Gears and Splines

Gears and splines, like screw threads, need a “stand in” feature of size. But because their configurations and applications are so varied, there's no default for gears and splines. In every case, the designer shall add a notation specifying an abstract feature of the gear or spline (such as MAJOR DIA, PITCH DIA, or MINOR DIA). This notation is placed beneath the feature control frame or beneath the “datum feature” symbol, as applicable.

5.2.3 Bounded Features

There is a type of feature that's neither a sphere, cylinder, nor width-type feature, yet clearly has “a set of two opposed elements.” The D-hole shown in Fig. 5-70, for example, is called an “irregular feature of size” by some drafting manuals, while Y14.5's own coverage for this type of feature is very limited. Although the feature has obvious MMC and LMC boundaries, it's arguable whether the feature is “associated with a size dimension.” We'll call this type of feature a *bounded feature*, and consider it a nonsize feature for our purposes. However, like features of size, bounded features are also subject to the principles of material condition modifiers, as we'll explain in section 5.6.2.1.

5.3 Symbols

In section 5.1, we touched on some of the shortcomings of English as a design specification language. Fig. 5-7 shows an attempt to control part features using mostly English. Compare that with Fig. 5-6, where GD&T symbols are used instead. Symbols are better, because of the following reasons:

- Anyone, regardless of his or her native tongue, can read and write symbols.
- Symbols mean exactly the same thing to everyone.
- Symbols are so compact they can be placed close to where they apply, and they reduce clutter.
- Symbols are quicker to draw and easier for computers to draw automatically.
- Symbols are easier to spot visually. For example, in Figs. 5-6 and 5-7, find all the positional callouts.

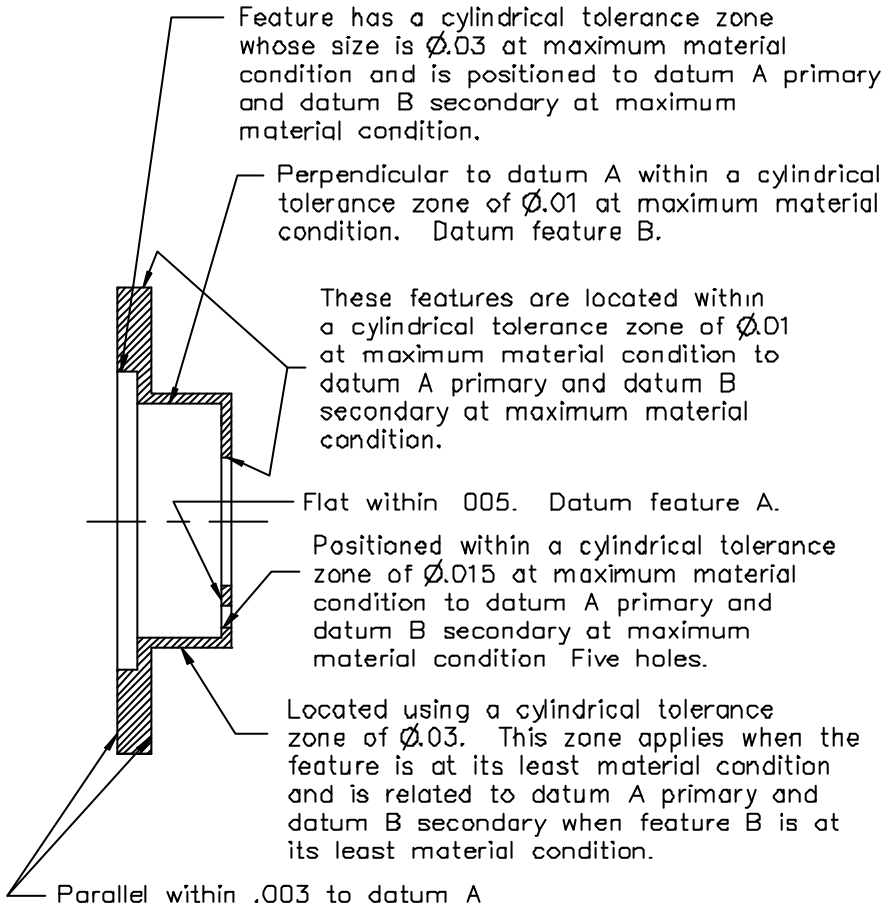


Figure 5-7 Using English to control part features

In the following sections, we'll explain the applications and meanings for each GD&T symbol. Unfortunately, the process of replacing traditional words with symbols is ongoing and complicated, requiring coordination among various national and international committees. In several contexts, Y14.5 suggests adding various English-language notes to a drawing to clarify design requirements. However, a designer should avoid notes specifying methods for manufacture or inspection.

5.3.1 Form and Proportions of Symbols

Fig. 5-8 shows each of the symbols used in dimensioning and tolerancing. We have added dimensions to the symbols themselves, to show how they are properly drawn. Each linear dimension is expressed as a multiple of h , a variable equal to the letter height used on the drawing. For example, if letters are drawn .12" high, then $h = .12$ " and $2h = .24$ ". It's important to draw the symbols correctly, because to many drawing users, that attention to detail indicates the draftsman's (or programmer's) overall command of the language.

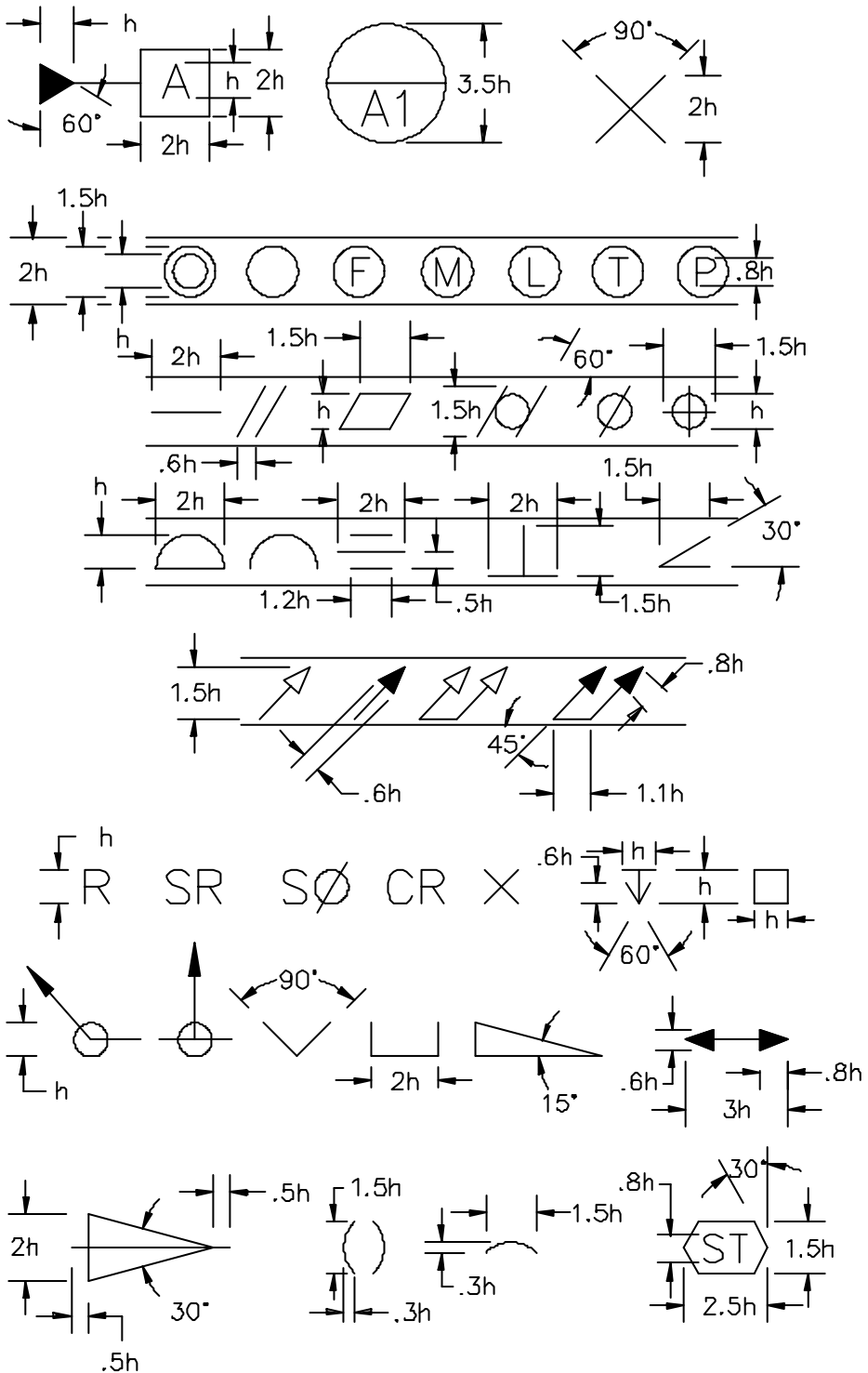


Figure 5-8 Symbols used in dimensioning and tolerancing

5.3.2 Feature Control Frame

Each geometric control for a feature is conveyed on the drawing by a rectangular sign called a *feature control frame*. As Fig. 5-9 shows, the feature control frame is divided into compartments expressing the following, sequentially from left to right.

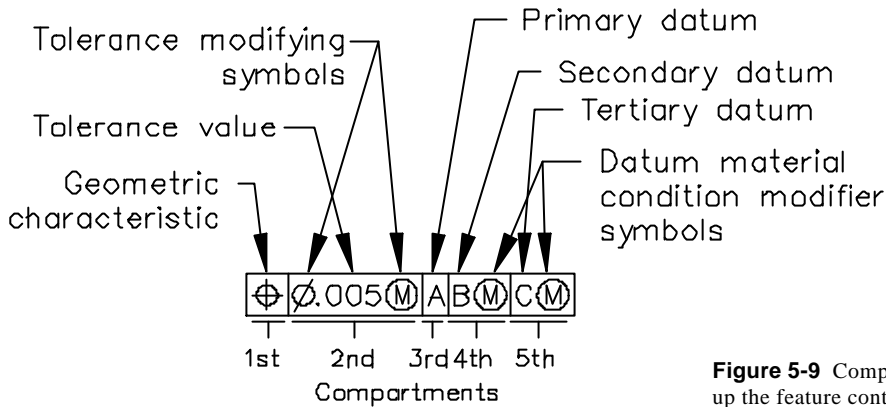


Figure 5-9 Compartments that make up the feature control frame

The **1st** compartment contains a *geometric characteristic symbol* specifying the type of geometric control. Table 5-1 shows the 14 available symbols.

The **2nd** compartment contains the geometric tolerance value. Many of the *modifying symbols* in Table 5-2 can appear in this compartment with the tolerance value, adding special attributes to the geometric control. For instance, where the tolerance boundary or zone is cylindrical, the tolerance value is preceded by the “diameter” symbol, \varnothing . Preceding the tolerance value with the “ $S\varnothing$ ” symbol denotes a spherical boundary or zone. Other optional modifying symbols, such as the “statistical tolerance” symbol, may follow the tolerance value.

The **3rd**, **4th**, and **5th** compartments are each added only as needed to contain (sequentially) the primary, secondary, and tertiary datum references, each of which may be followed by a material condition modifier symbol as appropriate.

Thus, each feature control frame displays most of the information necessary to control a single geometric characteristic of the subject feature. Only basic dimensions (described in section 5.3.3) are left out of the feature control frame.

5.3.2.1 Feature Control Frame Placement

Fig. 5-10(a) through (d) shows four different methods for attaching a feature control frame to its feature.

- (a) Place the frame below or attached to a leader-directed callout or dimension pertaining to the feature.
- (b) Run a leader from the frame to the feature.
- (c) Attach either side or either end of the frame to an extension line from the feature, provided it is a plane surface.
- (d) Attach either side or either end of the frame to an extension of the dimension line pertaining to a feature of size.

Table 5-1 summarizes the application options and rules for each of the 14 types of geometric tolerances. For each type of tolerance applied to each type of feature, the table lists the allowable “feature control frame placement options.” Multiple options, such as “a” and “d,” appearing in the same box yield identical results. Notice, however, that for some tolerances, the type of control depends on the feature control frame placement. For a straightness tolerance applied to a cylindrical feature, for instance, placement “b” controls surface elements, while placements “a” or “d” control the derived median line.

Table 5-1 Geometric characteristics and their attributes

CHARACTERISTIC	SYMBOL	TYPE OF FEATURE CONTROLLED	FEATURE CONTROL FRAME PLACEMENT OPTIONS (SEE LEGEND)					
			BOUNDARY/TOL ZONE SHAPE MODIFIER	TOLERANCE MODIFIABLE TO MMC OR LMC	NUMBER OF DATUM REFERENCES ALLOWED	MVC/LMC ALLOWED FOR DATUM REFERENCE(S)	BASIC DIMENSIONS	REQD
STRAIGHTNESS	—	CYL-SURFACE ELEMENTS	b			0		
		CYL-DERIVED MEDIAN LINE	a, d	∅	✓	0		
		PLANE-LINE ELEMENTS	b, c			0,1		
FLATNESS	▱	PLANE	b, c			0		
		WIDTH-DERIVED MEDIAN PLANE	a, d		✓	0		
CIRCULARITY	○	REVOLUTE, SPHERE	a, b, d			0		
CYLINDRICITY	∅	CYLINDER	a, b, d			0		
PROFILE OF A LINE	⤿	ALL	b			0-3	✓	✓
		REVOLUTE	b			0-3	✓	✓
		OTHER (NON-REVOLUTE)	b			0-3	✓	✓
PROFILE OF A SURFACE	⤿	COPLANARITY OF PLANES	b			0		
		PLANE (INCL LINE ELEMENTS)	b, c			1-3	✓	
		CYLINDER	a, d	∅	✓	1-3	✓	
		WIDTH	a, d		✓	1-3	✓	
PERPENDICULARITY PARALLELISM	⊥ //	REVOLUTE-RADIAL ELEMENT	b, c			1-3	✓	
		PLANE (INCL LINE ELEMENTS)	b, c			1-3	✓	✓
		CYLINDER	a, d	∅	✓	1-3	✓	✓
		WIDTH	a, d		✓	1-3	✓	✓
ANGULARITY	∠	REVOLUTE-RADIAL ELEMENT	b, c			1-3	✓	✓
		PLANE (INCL LINE ELEMENTS)	b, c			1-3	✓	✓
		CYLINDER	a, d	∅	✓	1-3	✓	✓
		WIDTH	a, d		✓	1-3	✓	✓
POSITION	⊕	REVOLUTE-RADIAL ELEMENT	b, c			1-3	✓	✓
		CYLINDER	a, d	∅	✓	1-3	✓	✓
		WIDTH	a, d		✓	1-3	✓	✓
CONCENTRICITY	⊙	SPHERE	a, d	S∅	✓	1-3	✓	✓
		ALL NON-SPHERICAL	a, b, d	∅		1-3		
		SPHERE	a, b, d	S∅		1-3		
SYMMETRY	≡	OPPOSED POINTS	a, d			1-3		
CIRCULAR RUNOUT	↗	REVOLUTE	a, b, d			1-2		
		CYLINDER	a, b, d			1-2		
TOTAL RUNOUT	↗	PLANE PERP TO AXIS	b, c			1-2		

FEATURE CONTROL FRAME PLACEMENT OPTIONS (LEGEND)

- (a) Place the frame below or attached to a leader-directed callout or dimension pertaining to the feature.
- (b) Run a leader from the frame to the feature.
- (c) Attach either side or either end of the frame to an extension line from the feature, provided it is a plane surface.
- (d) Attach either side or either end of the frame to an extension of the dimension line pertaining to a feature of size.

Table 5-2 Modifying symbols

Characteristic	Symbol
At maximum material condition	Ⓜ
At least material condition	Ⓛ
Projected tolerance zone	Ⓟ
Free state	ⓕ
Tangent plane	Ⓣ
Diameter	∅
Spherical diameter	S∅
Radius	R
Spherical radius	SR
Controlled radius	CR
Reference	()
Arc length	⌒
Statistical tolerance	ⓈⓉ
Between	↔

5.3.2.2 Reading a Feature Control Frame

It's easy to translate a feature control frame into English and read it aloud from left to right. Tables 5-1 and 5-2 show equivalent English words to the left of each symbol. Then, we just add the following English-language preface for each compartment:

- 1st compartment—"The..."
- 2nd compartment—"...of this feature shall be within..."
- 3rd compartment—"...to primary datum..."
- 4th compartment—"...and to secondary datum..."
- 5th compartment—"...and to tertiary datum..."

Now, read along with us Fig. 5-9's feature control frame. "The position of this feature shall be within diameter .005 at maximum material condition to primary datum A and to secondary datum B at maximum material condition and to tertiary datum C at maximum material condition." Easy.

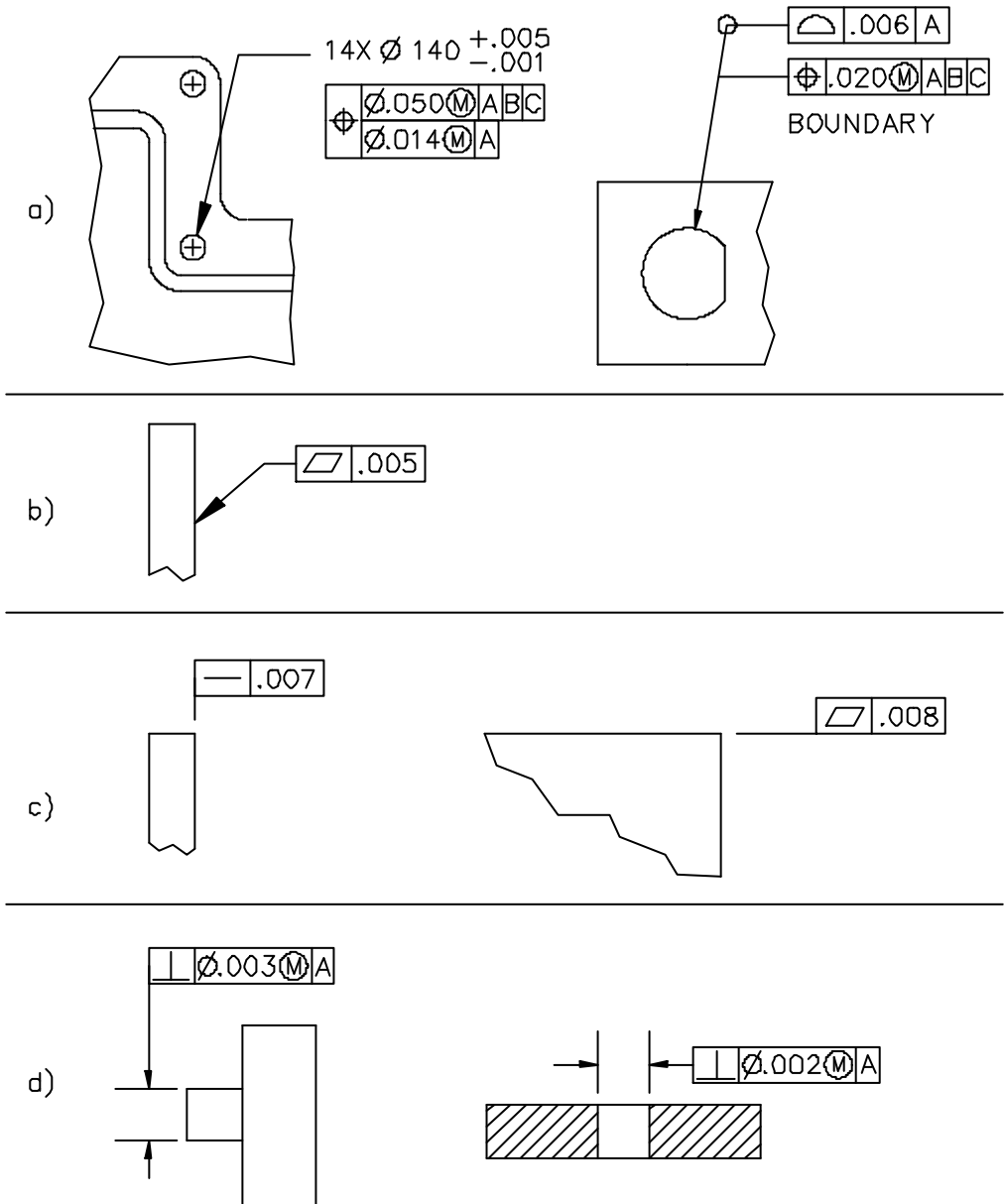


Figure 5-10 Methods of attaching feature control frames

5.3.3 Basic Dimensions

A *basic dimension* is a numerical value used to describe the theoretically exact size, profile, orientation, or location of a feature or datum target. The value is usually enclosed in a rectangular frame, as shown in

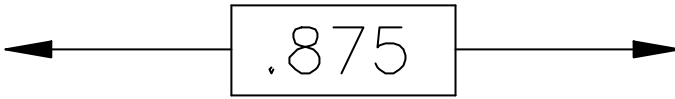


Figure 5-11 Method of identifying a basic .875 dimension

Fig. 5-11. Permissible variation from the basic value is specified in feature control frames, notes, or in other toleranced dimensions.

5.3.4 Reference Dimensions and Data

A *reference dimension* is a dimension, usually without tolerance, used for information only. On a drawing, a dimension (or other data) is designated as “reference” by enclosing it in parentheses. In written notes, however, parentheses retain their more common grammatical interpretation unless otherwise specified. Where a basic dimension is shown as a reference, enclosure in the “basic dimension frame” is optional. Although superfluous data and advice should be minimized on a drawing, a well-placed reference dimension can prevent confusion and time wasted by a user trying to decipher a relationship between features. Reference data shall either repeat or derive from specifications expressed elsewhere on the drawing or in a related document. However, the reference data itself shall have no bearing on part conformance.

5.3.5 “Square” Symbol

A square shape can be dimensioned using a single dimension preceded (with no space) by the “square” symbol shown in Fig. 5-47. The symbol imposes size limits and Rule #1 between each pair of opposite sides. (See section 5.6.3.1.) However, perpendicularity between adjacent sides is merely implied. Thus, the “square” symbol yields no more constraint than if 2X preceded the dimension.

5.3.6 Tabulated Tolerances

Where the tolerance in a feature control frame is tabulated either elsewhere on the drawing or in a related document, a representative letter is substituted in the feature control frame, preceded by the abbreviation TOL. See Figs. 5-116 and 5-117.

5.3.7 “Statistical Tolerance” Symbol

Chapters 8 and 10 explain how a *statistical tolerance* can be calculated using statistical process control (SPC) methods. Each tolerance value so calculated shall be followed by the “statistical tolerance” symbol shown in Fig. 5-12. In a feature control frame, the symbol follows the tolerance value and any applicable modifier(s). In addition, a note shall be placed on the drawing requiring statistical control of all such tolerances. Chapter 11 explains the note in greater detail and Chapter 24 shows several applications.



Figure 5-12 “Statistical tolerance” symbol

5.4 Fundamental Rules

Before we delve into the detailed applications and meanings for geometric tolerances, we need to understand a few fundamental ground rules that apply to every engineering drawing, regardless of the types of tolerances used.

(a) Each dimension shall have a tolerance, except for those dimensions specifically identified as reference, maximum, minimum, or stock (commercial stock size). The tolerance may be applied directly to the dimension (or indirectly in the case of basic dimensions), indicated by a general note, or located in a supplementary block of the drawing format. See ANSI Y14.1.

(b) Dimensioning and tolerancing shall be complete so there is full understanding of the characteristics of each feature. Neither scaling (measuring the size of a feature directly from an engineering drawing) nor assumption of a distance or size is permitted, except as follows: Undimensioned drawings, such as loft, printed wiring, templates, and master layouts prepared on stable material, are excluded provided the necessary control dimensions are specified.

(c) Each necessary dimension of an end product shall be shown. No more dimensions than those necessary for complete definition shall be given. The use of reference dimensions on a drawing should be minimized.

(d) Dimensions shall be selected and arranged to suit the function and mating relationship of a part and shall not be subject to more than one interpretation.

(e) The drawing should define a part without specifying manufacturing methods. Thus, only the diameter of a hole is given without indicating whether it is to be drilled, reamed, punched, or made by any other operation. However, in those instances where manufacturing, processing, quality assurance, or environmental information is essential to the definition of engineering requirements, it shall be specified on the drawing or in a document referenced on the drawing.

(f) It is permissible to identify as nonmandatory certain processing dimensions that provide for finish allowance, shrink allowance, and other requirements, provided the final dimensions are given on the drawing. Nonmandatory processing dimensions shall be identified by an appropriate note, such as **NONMANDATORY (MFG DATA)**.

(g) Dimensions should be arranged to provide required information for optimum readability. Dimensions should be shown in true profile views and refer to visible outlines.

(h) Wires, cables, sheets, rods, and other materials manufactured to gage or code numbers shall be specified by linear dimensions indicating the diameter or thickness. Gage or code numbers may be shown in parentheses following the dimension.

(i) A 90° angle applies where center lines and lines depicting features are shown on a drawing at right angles and no angle is specified.

(j) A 90° basic angle applies where center lines of features in a pattern or surfaces shown at right angles on the drawing are located or defined by basic dimensions and no angle is specified.

(k) Unless otherwise specified, all dimensions are applicable at 20°C (68°F). Compensation may be made for measurements made at other temperatures.

(l) All dimensions and tolerances apply in a free state condition. This principle does not apply to nonrigid parts as defined in section 5.5.

(m) Unless otherwise specified, all geometric tolerances apply for full depth, length, and width of the feature.

(n) Dimensions and tolerances apply only at the drawing level where they are specified. A dimension specified for a given feature on one level of drawing, (for example, a detail drawing) is not mandatory for that feature at any other level (for example, an assembly drawing).

5.5 Nonrigid Parts

A *nonrigid part* is a part that can have different dimensions while restrained in assembly than while relaxed in its “free state.” Rubber, plastic, or thin-wall parts may be obviously nonrigid. Other parts might reveal themselves as nonrigid only after assembly or functioning forces are applied. That’s why the exemption of “nonrigid parts” from Fundamental Rule (l) is meaningless. Instead, the rule must be inter-

puted as applying to all parts and meaning, “Unless otherwise specified, all dimensions and tolerances apply in a free state condition.” Thus, a designer must take extra care to assure that a suspected nonrigid part will have proper dimensions while assembled and functioning. To do so, one or more tolerances may be designated to apply while the part is restrained in a way that simulates, as closely as practicable, the restraining forces exerted in the part’s assembly and/or functioning.

5.5.1 Specifying Restraint

A nonrigid part might conform to all tolerances only in the free state, only in the restrained state, in both states, or in neither state. Where a part, such as a rubber grommet, may or may not need the help of restraint for conformance, the designer may specify optional restraint. This allows all samples to be inspected in their free states. Parts that pass are accepted. Those that fail may be reinspected—this time, while restrained. Where there is a risk that restraint could introduce unacceptable distortion, the designer should specify mandatory restraint instead.

Restraint may be specified by a note such as UNLESS OTHERWISE SPECIFIED, ALL DIMENSIONS AND TOLERANCES MAY (or SHALL) APPLY IN A RESTRAINED CONDITION. Alternatively, the note may be directed only to certain dimensions with flags and modified accordingly. The note shall always include (or reference a document that includes) detailed instructions for restraining the part. A typical note, like that shown in Fig. 5-134, identifies one or two functional datum features (themselves nonrigid) to be clamped into some type of gage or fixture. The note should spell out any specific clamps, fasteners, torques, and other forces deemed necessary to simulate expected assembly conditions.

5.5.2 Singling Out a Free State Tolerance

Even where restraint is specified globally on a drawing, a geometric tolerance can be singled out to apply only in the free state. Where the “free state” symbol follows a tolerance (and its modifiers), the tolerance shall be verified with no external restraining forces applied. See section 5.8.7 and Fig. 5-45 for an example.

5.6 Features of Size—The Four Fundamental Levels of Control

Four different levels of GD&T control can apply to a feature of size. Each higher-level tolerance adds a degree of constraint demanded by the feature’s functional requirements. However, all lower-level controls remain in effect. Thus, a single feature can be subject to many tolerances simultaneously.

Level 1: Controls size and (for cylinders or spheres) circularity at each cross section only.

Level 2: Adds overall form control.

Level 3: Adds orientation control.

Level 4: Adds location control.

5.6.1 Level 1—Size Limit Boundaries

For every feature of size, the designer shall specify the largest and the smallest the feature can be. In section 5.7, we discuss three different ways the designer can express these *size limits* (also called “limits of size”) on the drawing. Here, we’re concerned with the exact requirements these size limits impose on a feature. The Math Standard explains how specified size limits establish small and large *size limit boundaries* for the feature. The method may seem complicated at first, but it’s really very simple.

It starts with a geometric element called a *spine*. The spine for a cylindrical feature is a simple (nonself-intersecting) curve in space. Think of it as a line that may be straight or wavy. Next, we take an imaginary solid ball whose diameter equals the small size limit of the cylindrical feature, and sweep its center along the spine. This generates a “wormlike” 3-dimensional (3-D) boundary for the feature’s smallest size.

Fig. 5-13 illustrates the spine, the ball, and the 3-D boundary. Likewise, we may create a second spine, and sweep another ball whose diameter equals the large size limit of the cylindrical feature. This generates a second 3-D boundary, this time for the feature's largest size.

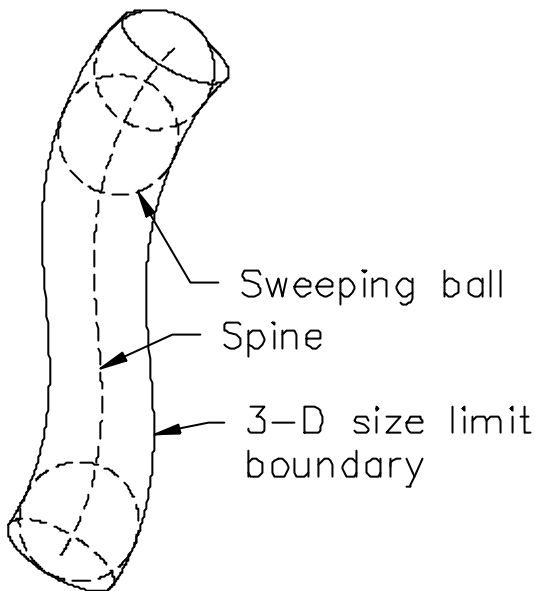


Figure 5-13 Generating a size limit boundary

As Fig. 5-14 shows, a cylindrical feature of size conforms to its size limits when its surface can contain the smaller boundary and be contained within the larger boundary. (The figure shows a hole, but the requirement applies to external features as well.) Under Level 1 control, the curvatures and relative locations of each spine may be adjusted as necessary to achieve the hierarchy of containments, except that the small size limit boundary shall be entirely contained within the large size limit boundary.

For a width-type feature (slot or tab), a spine is a simple (nonself-intersecting) surface. Think of it as a plane that may be flat or warped. The appropriate size ball shall be swept all over the spine, generating a 3-D boundary resembling a thick blanket. Fig. 5-15 illustrates the spines, balls, and 3-D boundaries for both size limits. Again, whether an internal or external feature, both feature surfaces shall contain the smaller boundary and be contained within the larger boundary.

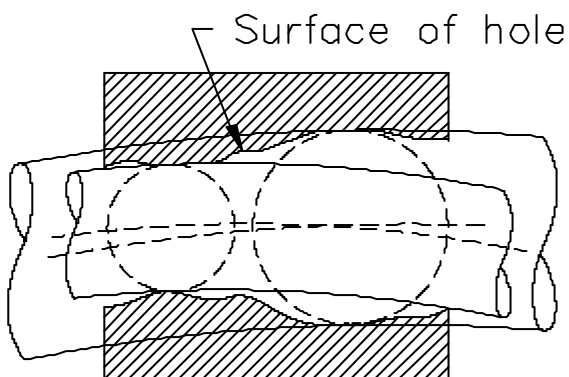


Figure 5-14 Conformance to limits of size for a cylindrical feature

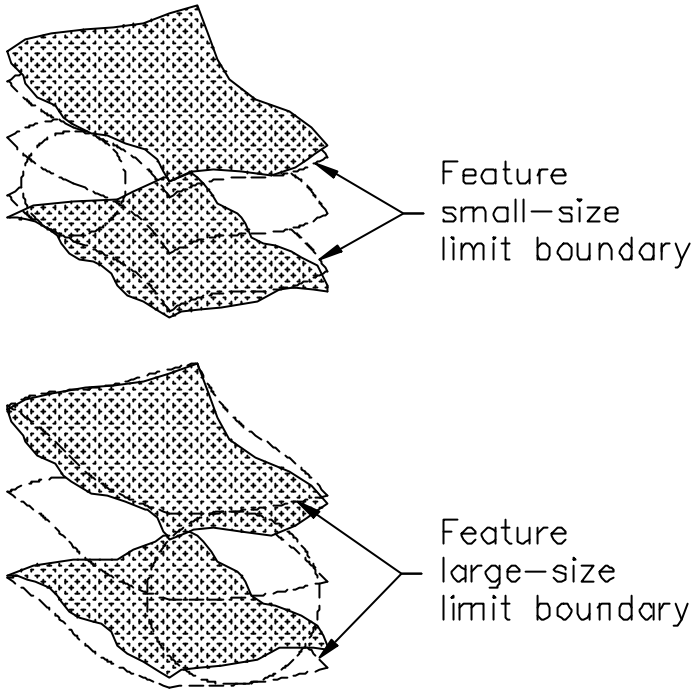


Figure 5-15 Conformance to limits of size for a width-type feature

The boundaries for a spherical feature of size are simply a small size limit sphere and a large size limit sphere. The rules for containment are the same and the boundaries need not be concentric.

In addition to limiting the largest and smallest a feature can be at any cross section, the two size limit boundaries control the *circularity* (roundness) at each cross section of a cylindrical or spherical feature of size. Fig. 5-16 shows a single cross section through a cylindrical feature and its small and large size limit boundaries. Notice that even though the small boundary is offset within the large boundary, the difference between the feature's widest and narrowest diameters cannot exceed the total size tolerance without violating a boundary. This Level 1 control of size and circularity at each cross section is adequate for most nonmating features of size. If necessary, circularity may be further refined with a separate circularity tolerance as described in section 5.8.5.

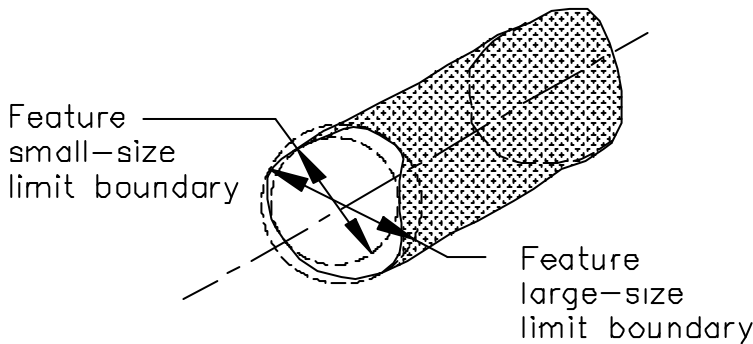


Figure 5-16 Size limit boundaries control circularity at each cross section

Obviously, the sweeping ball method is an ideal that cannot be realized with hard gages, but can be modeled by a computer to varying degrees of accuracy approaching the ideal. Since metrology (measuring) will always be an inexact science, inspectors are obliged to use the available tools to try to approximate the ideals. If the tool at hand is a pair of dial calipers or a micrometer, the inspector can only make “two-point” measurements across the width or diameter of a feature. But the inspector should make many such measurements and every measured value shall be between the low and high size limits. The inspector should also visually inspect the surface(s) for high or low regions that might violate a size limit boundary without being detected by the two-point measurements.

Before publication of the Math Standard, size limits were interpreted as applying to the smallest and largest two-point measurements obtainable at any cross section. However, with no spine linking the cross sections, there’s no requirement for continuity. A cylindrical boss could resemble coins carelessly stacked. It was agreed that such abrupt offsets in a feature are unsatisfactory for most applications. The new “sweeping ball” method expands GD&T beyond the confines of customary gaging methods, creating a mathematically perfect requirement equal to any technology that might evolve.

5.6.2 Material Condition

Material condition is another way of thinking about the size of an object taking into account the object’s nature. For example, the nature of a mountain is that it’s a pile of rock material. If you pile on more material, its “material condition” increases and the mountain gets bigger. The nature of a canyon is that it’s a void. As erosion decreases its “material condition,” the canyon gets bigger.

If a mating feature of size is as small as it can be, will it fit tighter or sloppier? Of course, you can’t answer until you know whether we’re talking about an internal feature of size, such as a hole, or an external feature of size, such as a pin. But, if we tell you a feature of size has less material, you know it will fit more loosely regardless of its type. *Material condition*, then, is simply a shorthand description of a feature’s size in the context of its intended function.

Maximum material condition (abbreviated MMC) is **the condition in which a feature of size contains the maximum amount of material within the stated limits of size.**

You can think of MMC as the condition where the most part material is present at the surface of a feature, or where the part weighs the most (all else being equal). This equates to the smallest allowable hole or the largest allowable pin, relative to the stated size limits.

Least material condition (abbreviated LMC) is **the condition in which a feature of size contains the least amount of material within the stated limits of size.**

You can think of LMC as the condition where the least part material is present at the surface of a feature, or where the part weighs the least (all else being equal). This equates to the largest allowable hole or the smallest allowable pin, relative to the stated size limits.

It follows then, that for every feature of size, one of the size limit boundaries is an *MMC boundary* corresponding to an *MMC limit*, and the other is an *LMC boundary* corresponding to an *LMC limit*. Depending on the type of feature and its function, the MMC boundary might ensure matability or removal of enough stock in a manufacturing process; the LMC boundary may ensure structural integrity and strength or ensure that the feature has enough stock for removal in a subsequent manufacturing process.

5.6.2.1 Modifier Symbols

Each geometric tolerance for a feature of size applies in one of the following three contexts:

- Regardless of Feature Size (RFS), the default
- modified to Maximum Material Condition (MMC)
- modified to Least Material Condition (LMC)

Table 5-1 shows which types of tolerances may be optionally “modified” to MMC or LMC. As we’ll detail in the following paragraphs, such modification causes a tolerance to establish a new and useful fixed-size boundary based on the geometric tolerance and the corresponding size limit boundary. Placing a material condition modifier symbol, either a circled M or a circled L, immediately following the tolerance value in the feature control frame modifies a tolerance. As we’ll explain in section 5.9.8.4, either symbol may also appear following the datum reference letter for each datum feature of size. In notes outside a feature control frame, use the abbreviation “MMC” or “LMC.”

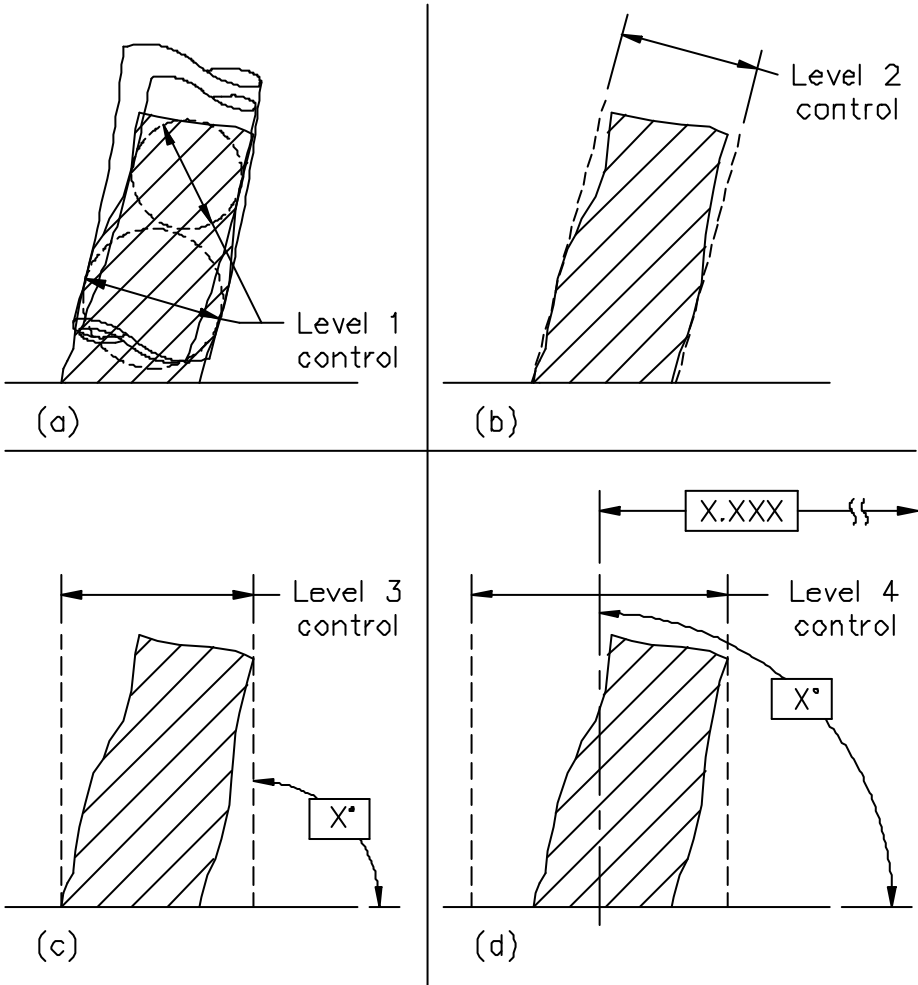


Figure 5-17 Levels of control for geometric tolerances modified to MMC

A geometric tolerance applied to a feature of size with no modifying symbol applies RFS. A few types of tolerances can only apply in an RFS context. As we'll explain in section 5.6.4, a Level 2, 3, or 4 tolerance works differently in an RFS context. Rather than a fixed-size boundary, the tolerance establishes a central tolerance zone.

5.6.3 Method for MMC or LMC

Geometric tolerances modified to MMC or LMC extend the system of boundaries for direct control of the feature surface(s). At each level of control, the applied tolerances establish a unique boundary, shown in Fig. 5-17(a) through (d) and Fig. 5-18(a) through (d), beyond which the feature surface(s) shall not encroach. Each higher-level tolerance creates a new boundary with an added constraint demanded by the feature's functional (usually mating) requirements. However, all lower-level controls remain in effect, regardless of their material condition contexts. Thus, a single feature can be subject to many boundaries simultaneously. The various boundaries are used in establishing datums (see Section 9), calculating tolerance stackups (see Chapters 9 and 11), and functional gaging (see Chapter 19).

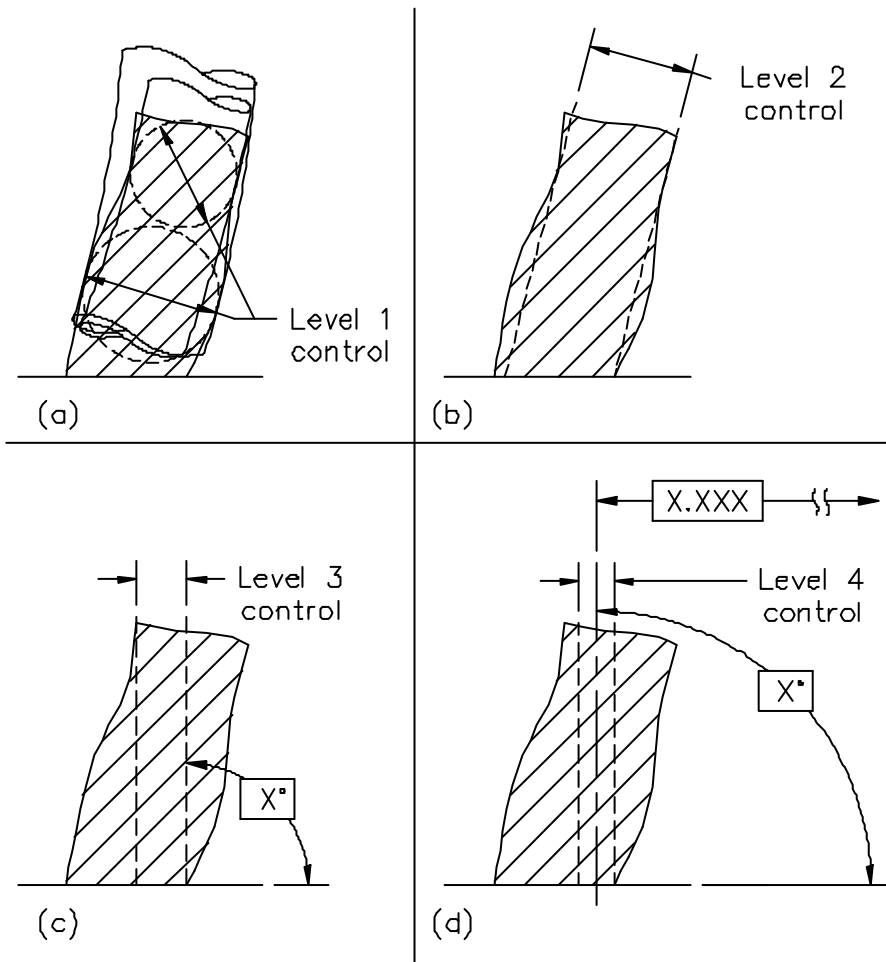


Figure 5-18 Levels of control for geometric tolerances modified to LMC

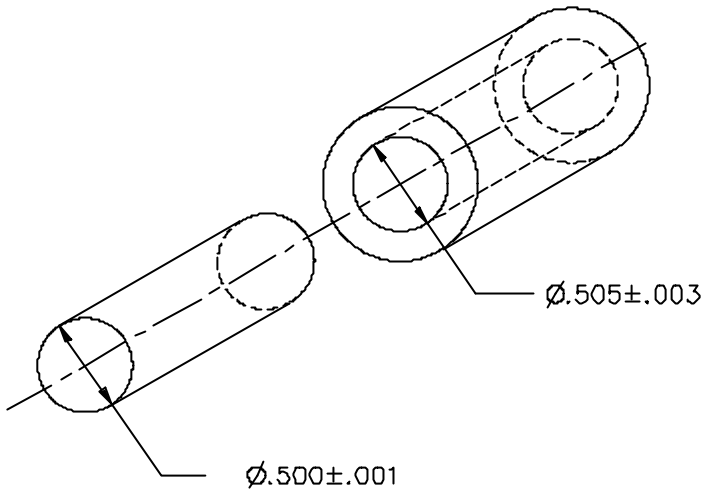


Figure 5-19 Cylindrical features of size that must fit in assembly

5.6.3.1 Level 2—Overall Feature Form

For features of size that must achieve a clearance fit in assembly, such as those shown in Fig. 5-19, the designer calculates the size tolerances based on the assumption that each feature, internal and external, is straight. For example, the designer knows that a $\varnothing.501$ maximum pin will fit in a $\varnothing.502$ minimum hole if both are straight. If one is banana shaped and the other is a lazy “S,” as shown in Fig. 5-20, they usually won’t

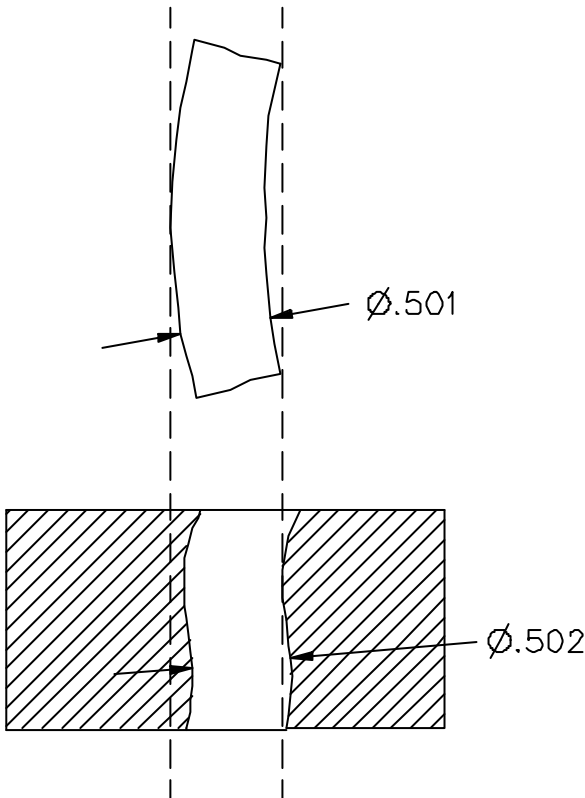


Figure 5-20 Level 1’s size limit boundaries will not assure assemblability

go together. Because Level 1's size limit boundaries can be curved, they can't assure assemblability. Level 2 adds control of the overall geometric shape or *form* of a feature of size by establishing a perfectly formed boundary beyond which the feature's surface(s) shall not encroach.

Boundaries of Perfect Form—A size limit spine can be required to be perfectly formed (straight or flat, depending on its type). Then, the sweeping ball generates a *boundary of perfect form*, either a perfect cylinder or pair of parallel planes. The feature surface(s) must then achieve some degree of straightness or flatness to avoid violating the boundary of perfect form. Boundaries of perfect form have no bearing on the orientational, locational, or coaxial relationships between features. However, this Level 2 control is usually adequate for a feature of size that relates to another feature in the absence of any orientation or location restraint between the two features—that is, where the features are free-floating relative to each other. Where necessary, overall form control may be adjusted with a separate straightness, flatness, or cylindricity tolerance, described in sections 5.8.2, 5.8.4, and 5.8.6, respectively.

For an individual feature of size, the MMC and LMC size limit boundaries can be required to have perfect form in four possible combinations: MMC only, LMC only, both, or neither. Each combination is invoked by different rules which, unfortunately, are scattered throughout Y14.5. We've brought them together in the following paragraphs. (Only the first rule is numbered.)

At MMC (Only)—Rule #1—Based on the assumption that most features of size must achieve a clearance fit, Y14.5 established a default rule for perfect form. Y14.5's *Rule #1* decrees that, unless otherwise specified or overridden by another rule, a feature's MMC size limit spine shall be perfectly formed (straight or flat, depending on its type). This invokes a boundary of perfect form at MMC (also called an *envelope*). Rule #1 doesn't require the LMC boundary to have perfect form.

In our example, Fig. 5-21 shows how Rule #1 establishes a $\varnothing.501$ boundary of perfect form at MMC (envelope) for the pin. Likewise, Rule #1 mandates a $\varnothing.502$ boundary of perfect form at MMC (envelope)

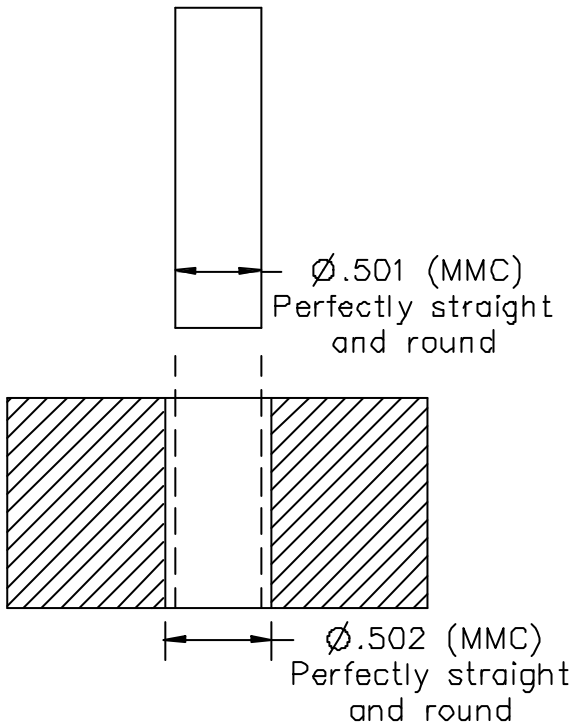


Figure 5-21 Rule #1 specifies a boundary of perfect form at MMC

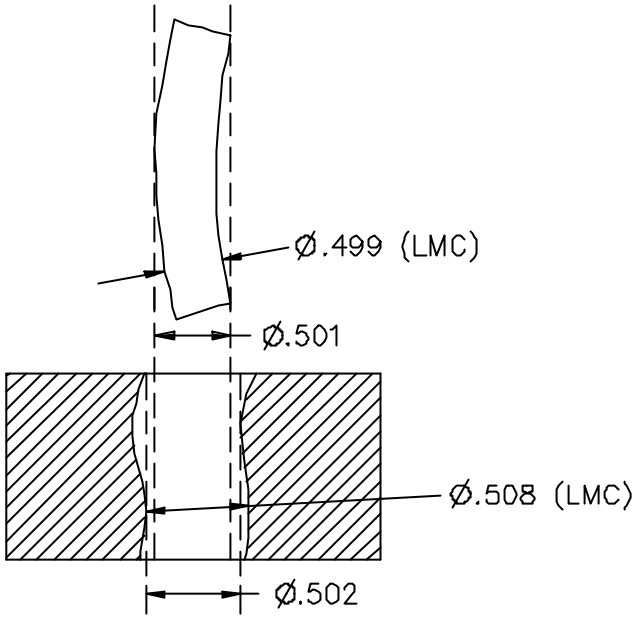


Figure 5-22 Rule #1 assures matability

for the hole. Fig. 5-22 shows how matability is assured for any pin that can fit inside its $\text{Ø}.501$ envelope and any hole that can contain its $\text{Ø}.502$ envelope. This simple hierarchy of fits is called the *envelope principle*.

At LMC (Only)—(Y14.5 section 5.3.5)—Fig. 5-23 illustrates a case where a geometric tolerance is necessary to assure an adequate “skin” of part material in or on a feature of size, rather than a clearance fit. In such an application, the feature of size at LMC represents the worst case. An LMC modifier applied to the geometric tolerance overrides Rule #1 for the controlled feature of size. Instead, the feature’s LMC spine shall be perfectly formed (straight or flat, depending on its type). This invokes a boundary of perfect form at LMC. The MMC boundary need not have perfect form. The same is true for a datum feature of size referenced at LMC.

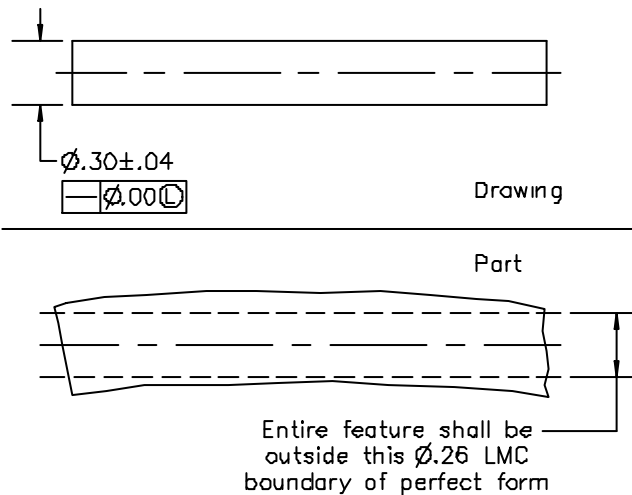


Figure 5-23 Using an LMC modifier to assure adequate part material

At both MMC and LMC—There are rare cases where a feature of size is associated with an MMC modifier in one context, and an LMC modifier in another context. For example, in Fig. 5-24, the datum B bore is controlled with a perpendicularity tolerance at MMC, then referenced as a datum feature at LMC. Each modifier for this feature, MMC and LMC, invokes perfect form for the feature’s corresponding size limit boundary.

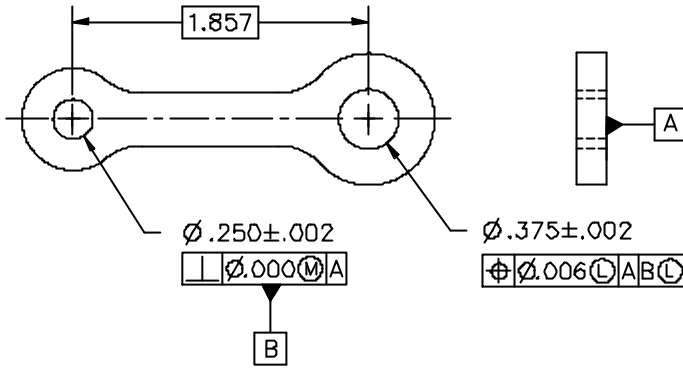


Figure 5-24 Feature of size associated with an MMC modifier and an LMC modifier

At neither MMC nor LMC—the Independency Principle—Y14.5 exempts the following from Rule #1.

- **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-furnished condition on the finished part.**
- Dimensions for which restrained verification is specified in accordance with section 5.5.1
- A cylindrical feature of size having a straightness tolerance associated with its diameter dimension (as described in section 5.8.2)
- A width-type feature of size having a straightness or (by extension of principle) flatness tolerance associated with its width dimension (as described in section 5.8.4)

In these cases, feature form is either noncritical or controlled by a straightness or flatness tolerance separate from the size limits. Since Rule #1 doesn’t apply, the size limits by themselves impose neither an MMC nor an LMC boundary of perfect form.

Fig. 5-25 is a drawing of an electrical bus bar. The cross-sectional dimensions have relatively close tolerances, not because the bar fits closely inside anything, but rather because of a need to assure a

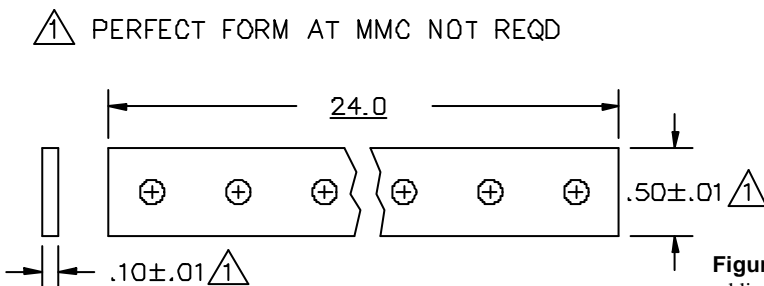


Figure 5-25 Nullifying Rule #1 by adding a note

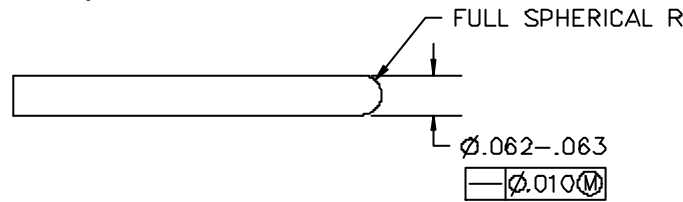
minimum current-carrying capacity without squandering expensive copper. Neither the MMC nor the LMC boundary need be perfectly straight. However, if the bus bar is custom rolled, sliced from a plate, or machined at all, it won't automatically be exempted from Rule #1. In such a case, Rule #1 shall be explicitly nullified by adding the note **PERFECT FORM AT MMC NOT REQD** adjacent to each of the bus bar's size dimensions.

Many experts argue that Rule #1 is actually the "exception," that fewer than half of all features of size need any boundary of perfect form. Thus, for the majority of features of size, Rule #1's perfect form at MMC requirement accomplishes nothing except to drive up costs. The rebuttal is that Y14.5 prescribes the "perfect form not required" note and designers simply fail to apply it often enough. Interestingly, ISO defaults to "perfect form not required" (sometimes called the *independency principle*) and requires application of a special symbol to invoke the "envelope" (boundary) of perfect form at MMC. This is one of the few substantial differences between the US and ISO standards.

Regardless of whether the majority of features of size are mating or nonmating, regardless of which principle, envelope or independency, is the default, every designer should consider for every feature of size whether a boundary of perfect form is a necessity or a waste.

Virtual Condition Boundary for Overall Form—There are cases where a perfect form boundary is needed, but at a different size than MMC. Fig. 5-26 shows a drawn pin that will mate with a very flexible socket in a mating connector. The pin has a high aspect (length-to-diameter) ratio and a close diameter tolerance. It would be extremely difficult to manufacture pins satisfying both Rule #1's boundary of perfect form at MMC ($\varnothing.063$) and the LMC ($\varnothing.062$) size limit. And since the mating socket has a flared lead-in, such near-perfect straightness isn't functionally necessary.

Drawing



Part

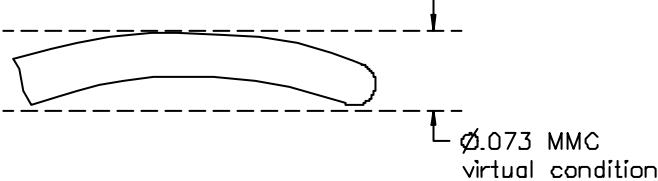


Figure 5-26 MMC virtual condition of a cylindrical feature

Fig. 5-27 shows a flat washer to be stamped out of sheet stock. The thickness (in effect, of the sheet stock) has a close tolerance because excessive variation could cause a motor shaft to be misaligned. Here again, for the tolerance and aspect ratio, Rule #1 would be unnecessarily restrictive. Nevertheless, an envelope is needed to prevent badly warped washers from jamming in automated assembly equipment.

In either example, the note **PERFECT FORM AT MMC NOT REQD** could be added, but would then allow pins as curly as a pig's tail or washers as warped as a potato chip. A better solution is to control the pin's overall form with a separate straightness tolerance modified to MMC. This replaces Rule #1's boundary of perfect form at MMC with a new perfect form boundary, called a *virtual condition boundary*, at some size other than MMC. Likewise, the washer's overall flatness can be controlled with a separate flatness tolerance modified to MMC. For details on how to apply these tolerances, see sections 5.8.2 and 5.8.4.

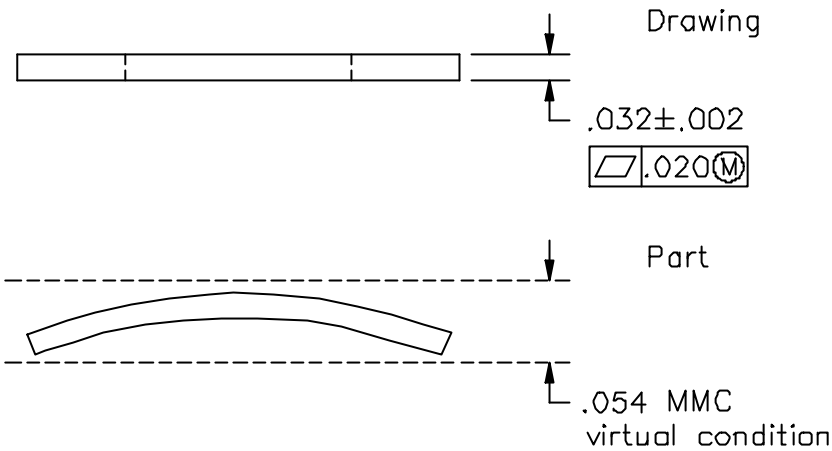


Figure 5-27 MMC virtual condition of a width-type feature

Any geometric tolerance applied to a feature of size and modified to MMC establishes a virtual condition boundary in the air adjacent to the feature surface(s). The boundary constitutes a restricted air space into which the feature shall not encroach. A geometric tolerance applied to a feature of size and modified to LMC likewise establishes a virtual condition boundary. However, in the LMC case, the boundary is embedded in part material, just beneath the feature surface(s). This boundary constitutes a restricted core or shell of part material into which the feature shall not encroach. The perfect geometric shape of any virtual condition boundary is a counterpart to the nominal shape of the controlled feature and is usually expressed with the form tolerance value, as follows.

Straightness Tolerance for a Cylindrical Feature—The “ \varnothing ” symbol precedes the straightness tolerance value. The tolerance specifies a virtual condition boundary that is a cylinder. The boundary cylinder extends over the entire length of the actual feature.

Flatness Tolerance for a Width-Type Feature—No modifying symbol precedes the flatness tolerance value. The tolerance specifies a virtual condition boundary of two parallel planes. The boundary planes extend over the entire length and breadth of the actual feature.

Whether the form tolerance is modified to MMC or LMC determines the size of the virtual condition boundary relative to the feature’s specified size limits.

Modified to MMC—The MMC virtual condition boundary represents a restricted air space reserved for the mating part feature. In such a mating interface, the internal feature’s MMC virtual condition boundary must be at least as large as that for the external feature. *MMC virtual condition* (the boundary’s fixed size) is determined by three factors: 1) the feature’s type (internal or external); 2) the feature’s MMC size limit; and 3) the specified geometric tolerance value.

For an internal feature of size:

$$\text{MMC virtual condition} = \text{MMC size limit} - \text{geometric tolerance}$$

For an external feature of size:

$$\text{MMC virtual condition} = \text{MMC size limit} + \text{geometric tolerance}$$

Four notes regarding these formulae:

1. For the pin in Fig. 5-26, the diameter of the virtual condition boundary equals the pin's MMC size plus the straightness tolerance value: $\varnothing.063 + \varnothing.010 = \varnothing.073$. This boundary can be simulated with a simple $\varnothing.073$ ring gage.
2. A Level 2 (straightness or flatness) tolerance value of zero at MMC is the exact equivalent of Rule #1 and therefore redundant.
3. For an internal feature, a geometric tolerance greater than the MMC size limit yields a negative virtual condition. This is no problem for computerized analysis, but it precludes functional gaging.
4. For a screw thread, an MMC virtual condition can be calculated easily based on the MMC pitch diameter. The boundary, however, has limited usefulness in evaluating an actual thread.

Modified to LMC—The LMC virtual condition boundary assures a protected core of part material within a pin, boss, or tab, or a protected case of part material around a hole or slot. *LMC virtual condition* (the boundary's fixed size) is determined by three factors: 1) the feature's type (internal or external); 2) the feature's LMC size limit; and 3) the specified geometric tolerance value.

For an internal feature of size:

$$\text{LMC virtual condition} = \text{LMC size limit} + \text{geometric tolerance}$$

For an external feature of size:

$$\text{LMC virtual condition} = \text{LMC size limit} - \text{geometric tolerance}$$

Fig. 5-28 shows a part where straightness of datum feature A is necessary to protect the wall thickness. Here, the straightness tolerance modified to LMC supplants the boundary of perfect form at LMC. The tolerance establishes a virtual condition boundary embedded in the part material beyond which the feature surface shall not encroach. For datum feature A in Fig. 5-28, the diameter of this boundary equals the LMC size minus the straightness tolerance value: $\varnothing.247 - \varnothing.005 = \varnothing.242$. Bear in mind the difficulties of verifying conformance where the virtual condition boundary is embedded in part material and can't be simulated with tangible gages.

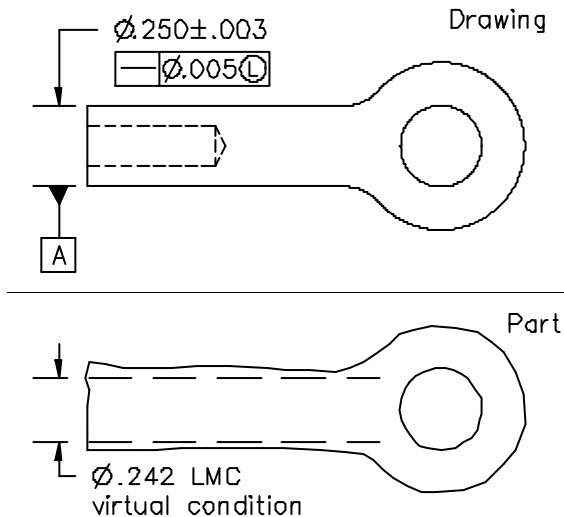


Figure 5-28 LMC virtual condition of a cylindrical feature

5.6.3.2 Level 3—Virtual Condition Boundary for Orientation

For two mating features of size, Level 2’s perfect form boundaries can only assure assemblability in the absence of any orientation or location restraint between the two features—that is, the features are free-floating relative to each other. In Fig. 5-29, we’ve taken our simple example of a pin fitting into a hole, and added a large flange around each part. We’ve also stipulated that the two flanges shall bolt together and make full contact. This introduces an orientation restraint between the two mating features. When the flange faces are bolted together tightly, the pin and the hole must each be very square to their respective flange faces. Though the pin and the hole might each respect their MMC boundaries of perfect form, nothing prevents those boundaries from being badly skewed to each other.

We can solve that by taking the envelope principle one step further to Level 3. An orientation tolerance applied to a feature of size, modified to MMC or LMC, establishes a virtual condition boundary beyond which the feature’s surface(s) shall not encroach. For details on how to apply an orientation tolerance, see section 5.10.1. In addition to perfect form, this new boundary has perfect orientation in all applicable degrees of freedom relative to any datum feature(s) we select (see section 5.9.7). The shape and size of the virtual condition boundary for orientation are governed by the same rules as for form at Level 2. A single feature of size can be subject to multiple virtual condition boundaries.

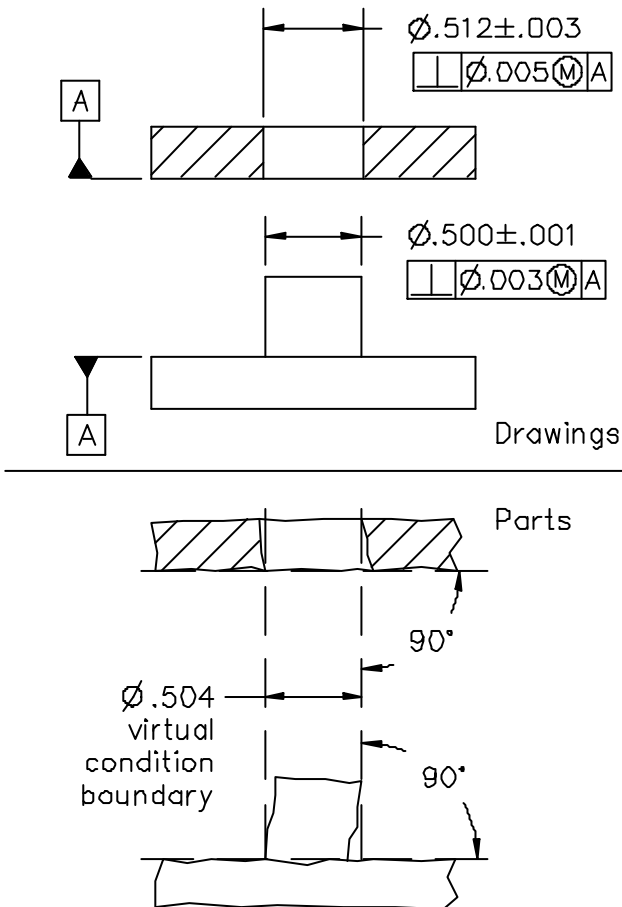


Figure 5-29 Using virtual condition boundaries to restrain orientation between mating features

For each example part in Fig. 5-29, we've restrained the virtual condition boundary perpendicular to the flange face. The lower portion of Fig. 5-29 shows how matability is assured for any part having a pin that can fit inside its $\varnothing.504$ MMC virtual condition boundary and any part having a hole that can contain its $\varnothing.504$ MMC virtual condition boundary.

5.6.3.3 Level 4—Virtual Condition Boundary for Location

For two mating features of size, Level 3's virtual condition boundary for orientation can only assure assemblability in the absence of any location restraint between the two features, for example, where no other mating features impede optimal location alignment between our pin and hole. In Fig. 5-30, we've

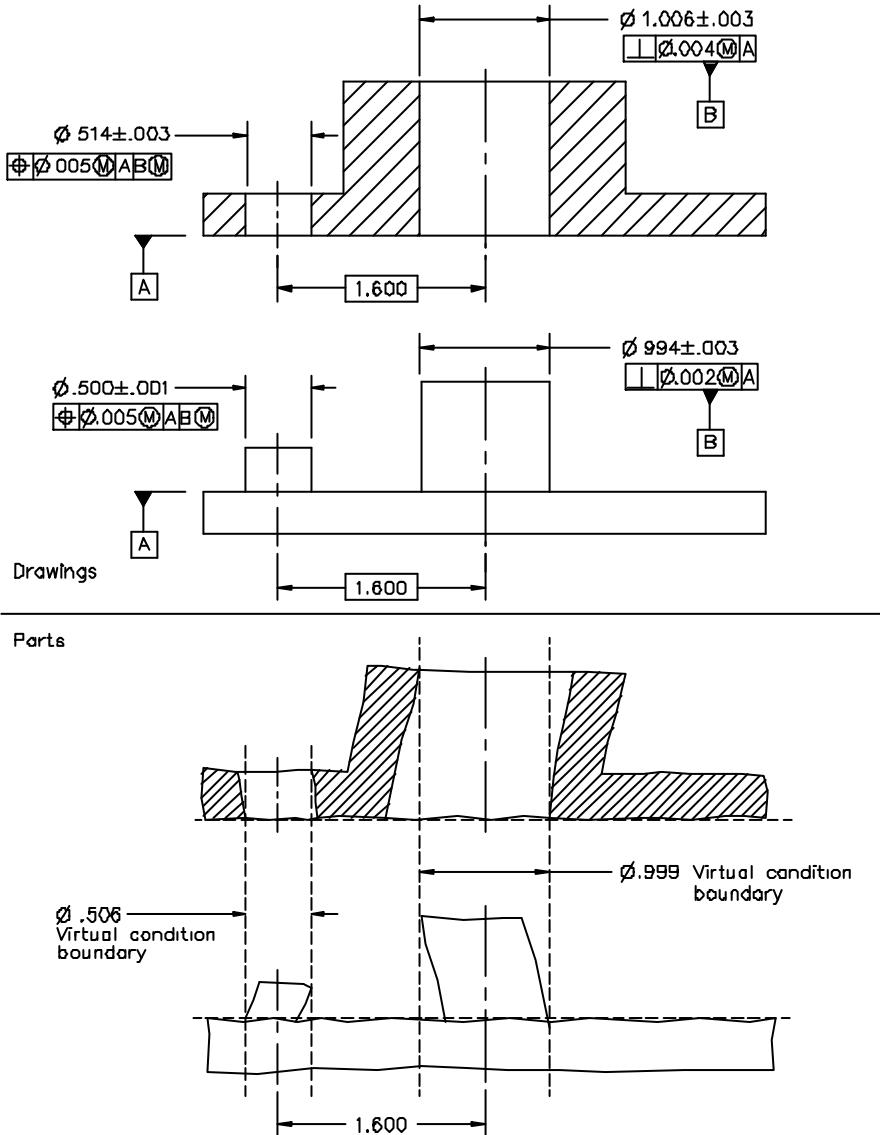


Figure 5-30 Using virtual condition boundaries to restrain location (and orientation) between mating features

moved the pin and hole close to the edges of the flanges and added a larger bore and boss mating interface at the center of the flanges. When the flange faces are bolted together tightly and the boss and bore are fitted together, the pin and the hole must each still be very square to their respective flange faces. However, the parts can no longer slide freely to optimize the location alignment between the pin and the hole. Thus, the pin and the hole must each additionally be accurately located relative to its respective boss or bore.

A positional tolerance applied to a feature of size, modified to MMC or LMC, takes the virtual condition boundary one step further to Level 4. For details on how to apply a positional tolerance, see section 5.11.2. In addition to perfect form and perfect orientation, the new boundary shall have perfect location in all applicable degrees of freedom relative to any datum feature(s) we select (see section 5.9.7). The shape and size of the virtual condition boundary for location are governed by the same rules as for form at Level 2 and orientation at Level 3, with one addition. For a spherical feature, the tolerance is preceded by the “S \varnothing ” symbol and specifies a virtual condition boundary that is a sphere. A single feature of size can be subject to multiple virtual condition boundaries—one boundary for each form, orientation, and location tolerance applied.

In Fig. 5-30, we’ve identified four datums and added dimensions and tolerances for our example assembly. The central boss has an MMC size limit of $\varnothing.997$ and a perpendicularity tolerance of $\varnothing.002$ at MMC. Since it’s an external feature of size, its virtual condition is $\varnothing.997 + \varnothing.002 = \varnothing.999$. The bore has an MMC size limit of $\varnothing 1.003$ and a perpendicularity tolerance of $\varnothing.004$ at MMC. Since it’s an internal feature of size, its virtual condition is $\varnothing 1.003 - \varnothing.004 = \varnothing.999$. Notice that for each perpendicularity tolerance, the datum feature is the flange face. Each virtual condition boundary for orientation is restrained perfectly perpendicular to its referenced datum, derived from the flange face. As the lower portion of Fig. 5-30 shows, the boss and bore will mate every time.

The pin and hole combination requires MMC virtual condition boundaries with location restraint added. Notice that for each positional tolerance, the primary datum feature is the flange face and the secondary datum feature is the central boss or bore. Each virtual condition boundary for location is restrained perfectly perpendicular to its referenced primary datum, derived from the flange face. Each boundary is additionally restrained perfectly located relative to its referenced secondary datum, derived from the boss or bore. This restraint of both orientation and location on each part is crucial to assuring perfect alignment between the boundaries on both parts, and thus, assemblability. The pin has an MMC size limit of $\varnothing.501$ and a positional tolerance of $\varnothing.005$ at MMC. Since it’s an external feature of size, its virtual condition is $\varnothing.501 + \varnothing.005 = \varnothing.506$. The hole has an MMC size limit of $\varnothing.511$ and a positional tolerance of $\varnothing.005$ at MMC. Since it’s an internal feature of size, its virtual condition is $\varnothing.511 - \varnothing.005 = \varnothing.506$. Any pin contained within its $\varnothing.506$ boundary can assemble with any hole containing its $\varnothing.506$ boundary. Try that without GD&T!

5.6.3.4 Level 3 or 4 Virtual Condition Equal to Size Limit (Zero Tolerance)

All the tolerances in our example assembly were chosen to control the fit between the two parts. Subsequent chapters deal with the myriad considerations involved in determining fits. To simplify our example, we matched virtual condition sizes for each pair of mating features. All our intermediate values, however, were chosen arbitrarily.

For example, in Fig. 5-30, the boss’s functional extremes are at $\varnothing.991$ and $\varnothing.999$. Between them, the total tolerance is $\varnothing.008$. Based on our own assumptions about process variation, we arbitrarily divided this into $\varnothing.006$ for size and $\varnothing.002$ for orientation. Thus, the $\varnothing.997$ MMC size limit has no functional significance. We might just as well have divided the $\varnothing.008$ total into $\varnothing.004 + \varnothing.004$, $\varnothing.006 + \varnothing.002$, or even $\varnothing.008 + \varnothing.000$.

In a case such as this, where the only MMC design consideration is a clearance fit, it's not necessary for the designer to apportion the fit tolerance. Why not give it all to the manufacturing process and let the process divvy it up as needed? This is accomplished by stretching the MMC size limit to equal the MMC virtual condition size and reducing the orientation or positional tolerance to zero.

Fig. 5-31 shows our example assembly with orientation and positional tolerances of zero. Notice that now, the central boss has an MMC size limit of $\varnothing.999$ and a perpendicularity tolerance of $\varnothing.000$ at MMC.

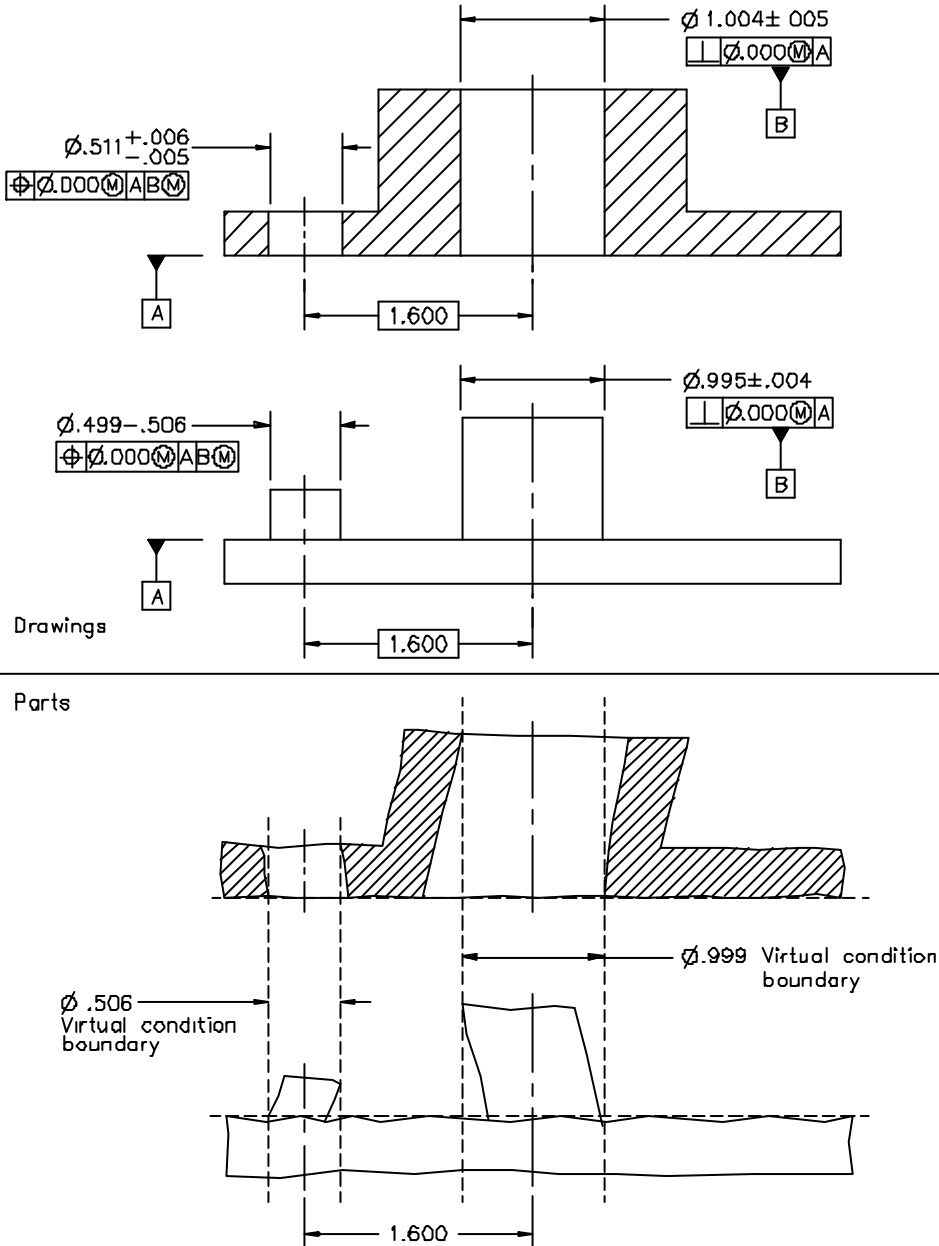


Figure 5-31 Zero orientation tolerance at MMC and zero positional tolerance at MMC

Since it's an external feature of size, its virtual condition is $\varnothing.999 + \varnothing.000 = \varnothing.999$.

Compare the lower portions of Figs. 5-30 and 5-31. The conversion to zero orientation and positional tolerances made no change to any of the virtual condition boundaries, and therefore, no change in assemblability and functionality. However, manufacturability improved significantly for both parts. Allowing the process to apportion tolerances opens up more tooling choices. In addition, a perfectly usable part having a boss measuring $\varnothing.998$ with perpendicularity measuring $\varnothing.0006$ will no longer be rejected.

The same rationale may be applied where a Level 3 or 4 LMC virtual condition exists. Unless there's a functional reason for the feature's LMC size limit to differ from its LMC virtual condition, make them equal by specifying a zero orientation or positional tolerance at LMC, as appropriate.

Some novices may be alarmed at the sight of a zero tolerance. "How can anything be made perfect?" they ask. Of course, a zero tolerance doesn't require perfection; it merely allows parity between two different levels of control. The feature shall be manufactured with size and orientation adequate to clear the virtual condition boundary. In addition, the feature shall nowhere encroach beyond its opposite size limit boundary.

5.6.3.5 Resultant Condition Boundary

For the $\varnothing.514$ hole in Fig. 5-30, we have primary and secondary design requirements. Since the hole must clear the $\varnothing.500$ pin in the mating part, we control the hole's orientation and location with a positional tolerance modified to MMC. This creates an MMC virtual condition boundary that guarantees air space for the mating pin. But now, we're worried that the wall might get too thin between the hole and the part's edge.

To address this secondary concern, we need to determine the farthest any point around the hole can range from "true position" (the ideal center). That distance constitutes a worst-case perimeter for the hole shown in Fig. 5-32 and called the *resultant condition boundary*. We can then compare the resultant condition boundary with that for the flange diameter and calculate the worst-case thin wall. We may then need to adjust the positional tolerance and/or the size limits for the hole and/or the flange.

Resultant condition is defined as a variable value obtained by adding the total allowable geometric tolerance to (or subtracting it from) the feature's actual mating size. Tables in Y14.5 show resultant condition values for feature sizes between the size limits. However, the only resultant condition value that anyone cares about is the single worst-case value defined below, as determined by three factors: 1) the feature's type (internal or external); 2) the feature's size limits; and 3) the specified geometric tolerance value.

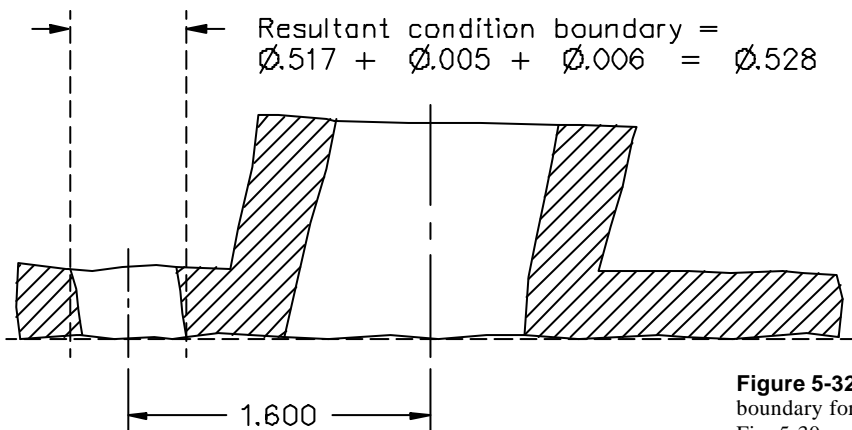


Figure 5-32 Resultant condition boundary for the $\varnothing.514$ hole in Fig. 5-30

For an internal feature of size controlled at MMC:

Resultant condition = LMC size limit + geometric tolerance + size tolerance

For an external feature of size controlled at MMC:

Resultant condition = LMC size limit – geometric tolerance – size tolerance

For an internal feature of size controlled at LMC:

Resultant condition = MMC size limit – geometric tolerance – size tolerance

For an external feature of size controlled at LMC:

Resultant condition = MMC size limit + geometric tolerance + size tolerance

5.6.4 Method for RFS

A geometric tolerance applied to a feature of size with no modifying symbol applies RFS. A few types of tolerances can only apply in an RFS context. Instead of a boundary, a Level 2, 3, or 4 tolerance RFS establishes a central tolerance zone, within which a geometric element derived from the feature shall be contained. Each higher-level tolerance adds a degree of constraint demanded by the feature's functional requirements, as shown in Fig. 5-33(a) through (d). However, all lower-level controls remain in effect, regardless of their material condition contexts. Thus, a single feature can be subject to many tolerance zones and boundaries simultaneously. Unfortunately, tolerance zones established by RFS controls cannot be simulated by tangible gauges. This often becomes an important design consideration.

5.6.4.1 Tolerance Zone Shape

The geometrical shape of the RFS tolerance zone usually corresponds to the shape of the controlled feature and is expressed with the tolerance value, as follows.

For a Width-Type Feature—Where no modifying symbol precedes the tolerance value, the tolerance specifies a tolerance zone bounded by two parallel planes separated by a distance equal to the specified tolerance. The tolerance planes extend over the entire length and breadth of the actual feature.

For a Cylindrical Feature—The tolerance value is preceded by the “ \varnothing ” symbol and specifies a tolerance zone bounded by a cylinder having a diameter equal to the specified tolerance. The tolerance cylinder extends over the entire length of the actual feature.

For a Spherical Feature—The tolerance is preceded by the “ $S\varnothing$ ” symbol and specifies a tolerance zone bounded by a sphere having a diameter equal to the specified tolerance.

5.6.4.2 Derived Elements

A multitude of geometric elements can be derived from any feature. A geometric tolerance RFS applied to a feature of size controls one of these five:

- Derived median line (from a cylindrical feature)
- Derived median plane (from a width-type feature)
- Feature center point (from a spherical feature)
- Feature axis (from a cylindrical feature)
- Feature center plane (from a width-type feature)

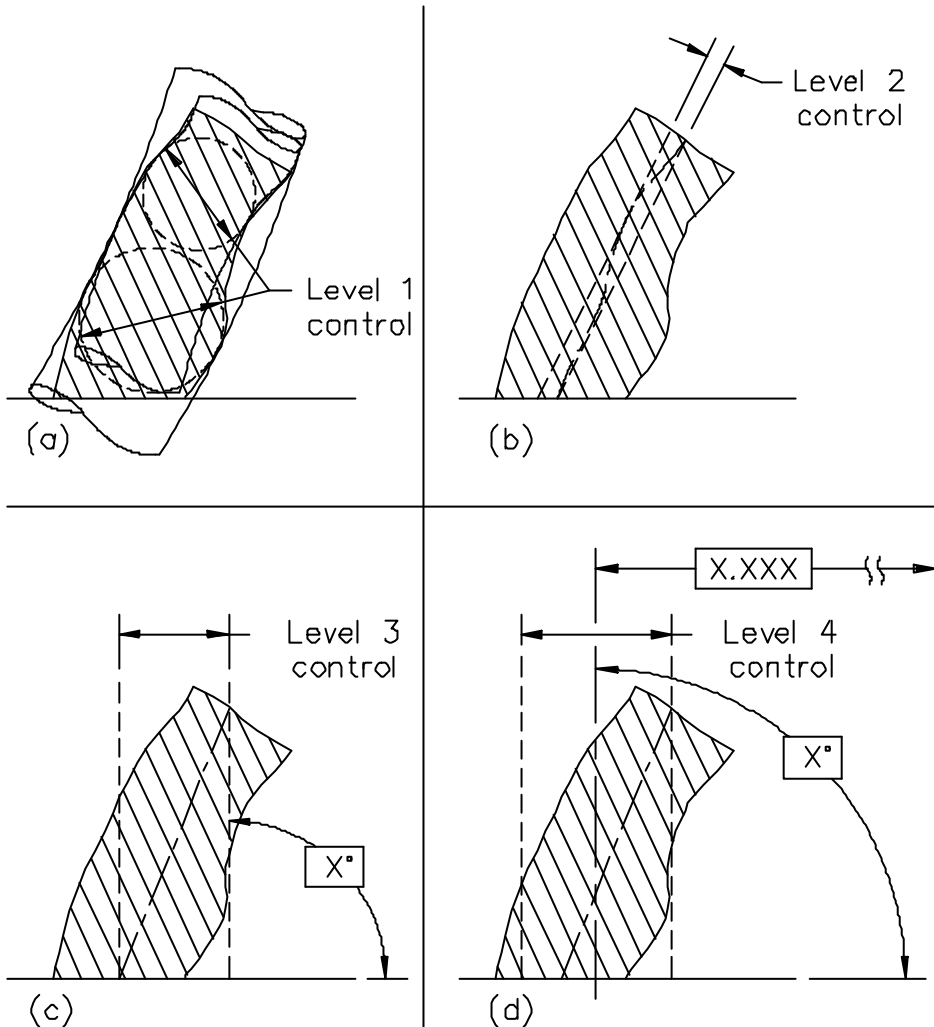


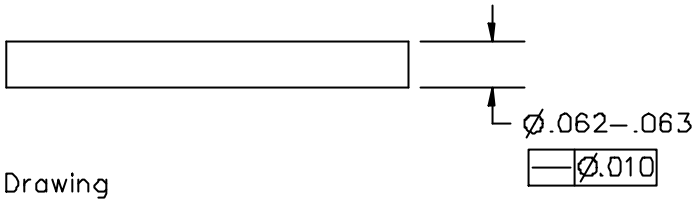
Figure 5-33 Levels of control for geometric tolerances applied RFS

A Level 2 (straightness or flatness) tolerance nullifies Rule #1's boundary of perfect form at MMC. Instead, the separate tolerance controls overall feature form by constraining the derived median line or derived median plane, according to the type of feature.

A cylindrical feature's *derived median line* is an imperfect line (abstract) that passes through the center points of all cross sections of the feature. These cross sections are normal to the axis of the actual mating envelope. The cross section center points are determined as per ANSI B89.3.1.

A width-type feature's *derived median plane* is an imperfect plane (abstract) that passes through the center points of all line segments bounded by the feature. These line segments are normal to the actual mating envelope.

In Fig. 5-34, the absence of a material condition modifier symbol means the straightness tolerance applies RFS by default. This specifies a tolerance zone bounded by a cylinder having a diameter equal to the tolerance value, within which the derived median line shall be contained. In Fig. 5-35, the flatness tolerance applies RFS by default. This specifies a tolerance zone bounded by two parallel planes separated by a distance equal to the tolerance value, within which the entire derived median plane shall be contained. Both size limits are still in force, but neither the spine for the MMC size boundary nor the spine for the LMC size boundary need be perfectly formed. A straightness or flatness tolerance value may be less than, equal to, or greater than the size tolerance.



Drawing

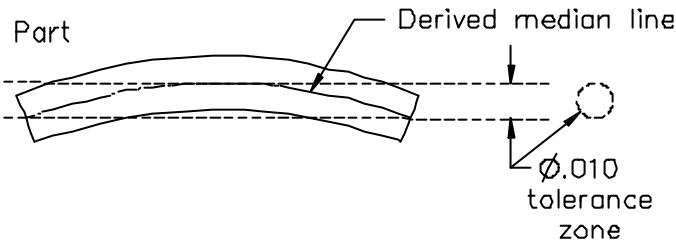
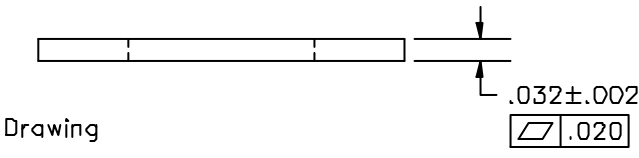


Figure 5-34 Tolerance zone for straightness control RFS

As you can imagine, deriving a median line or plane is a complex procedure that's extremely difficult without the help of a microprocessor-based machine. But where it's necessary to control overall form with a tolerance that remains constant, regardless of feature size, there are no simpler options. However, once we've assured overall form with Rule #1 or a separate form tolerance, we can apply Level 3 and 4 tolerances to geometric elements that are more easily derived: a center point, perfectly straight axis, or perfectly flat center plane. These elements must be defined and derived to represent the features' worst-case functionality.



Drawing

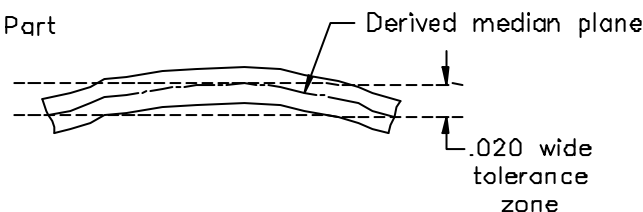


Figure 5-35 Tolerance zone for flatness control RFS

In an RFS context, the *feature center point*, *feature axis*, or *feature center plane* is the center of the feature's *actual mating envelope*. In all cases, a feature's axis or center plane extends for the full length and/or breadth of the feature.

The *actual mating envelope* is a surface, or pair of parallel-plane surfaces, of perfect form, which correspond to a part feature of size as follows:

- (a) *For an External Feature.* A similar perfect feature counterpart of smallest size, which can be circumscribed about the feature so that it just contacts the feature surface(s). For example, a smallest cylinder of perfect form or two parallel planes of perfect form at minimum separation that just contact(s) the surface(s).
- (b) *For an Internal Feature.* A similar perfect feature counterpart of largest size, which can be inscribed within the feature so that it just contacts the feature surface(s). For example, a largest cylinder of perfect form or two parallel planes of perfect form at maximum separation that just contact(s) the surface(s).

In certain cases, the orientation, or the orientation and location of an actual mating envelope shall be restrained to one or two datums (see Fig. 5-36 and Table 5-3). In Fig. 5-37, for example, the true geometric counterpart of datum feature B is the actual mating envelope (smallest perfect cylinder) restrained perpendicular to datum plane A.

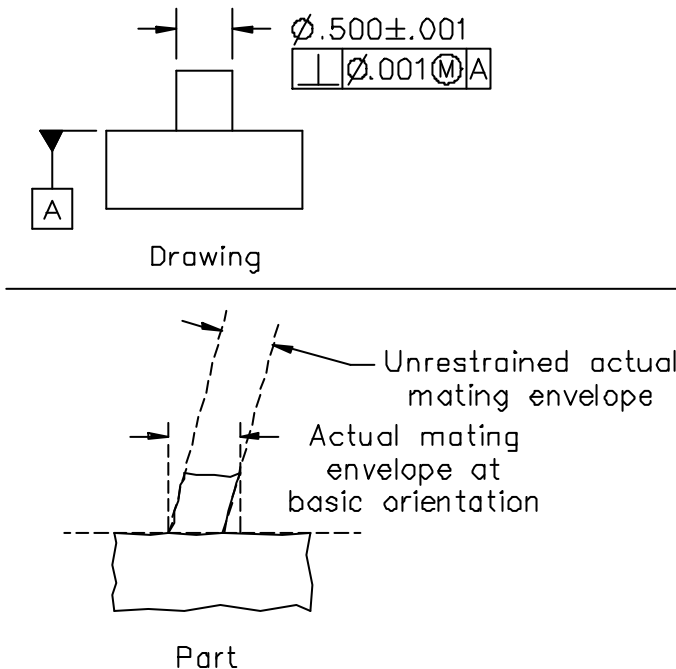


Figure 5-36 Example of restrained and unrestrained actual mating envelopes

Be careful not to confuse the actual mating envelope with the boundary of perfect form at MMC “envelope.” Our above definitions are cobbled together from both Y14.5 and the Math Standard, since the standards differ slightly. Table 5-3 shows that in most cases, the actual mating envelope is unrestrained—that is, allowed to achieve any orientation and location when fitted to the feature. As we’ll discuss later, when simulating a secondary or tertiary datum feature RFS, the actual mating envelope shall be oriented (held square) to the higher precedence datum(s). Obviously, that restraint will produce a different fit.

Table 5-3 Actual mating envelope restraint

PURPOSE OF ENVELOPE	APPROPRIATE RESTRAINT	
	Unrestrained	Restrained to higher datum(s)
Evaluate conformance to:		
Rule #1	X	
orientation tolerance	X	
positional tolerance	X	
Establish True Geometric Counterpart RFS for a datum feature:		
primary	X	
secondary, tertiary		X
Actual mating size of datum feature for DRF displacement		
primary	X	
secondary, tertiary		X

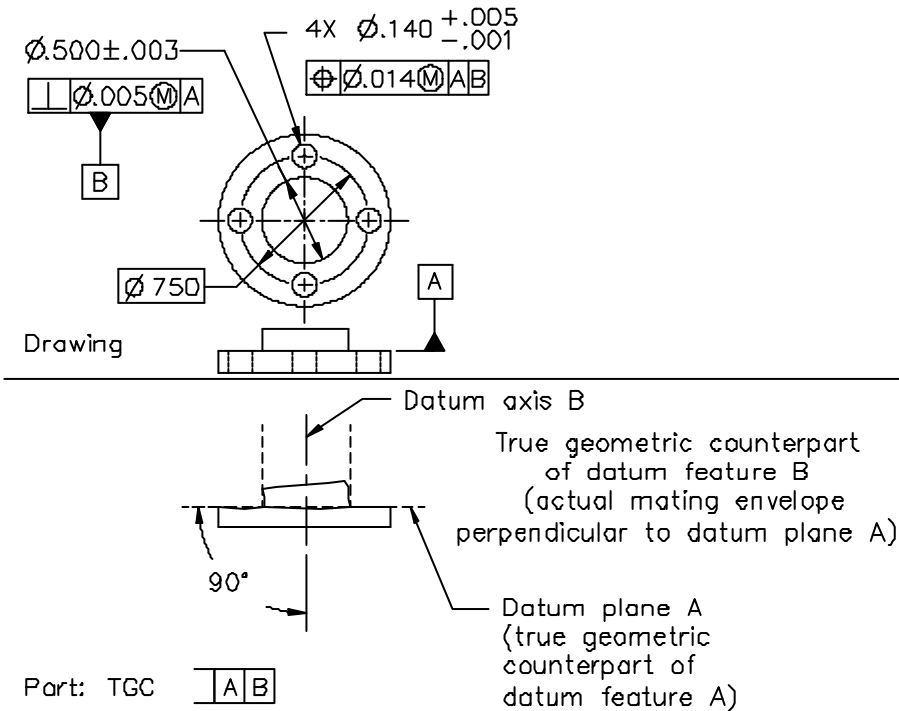


Figure 5-37 The true geometric counterpart of datum feature B is a restrained actual mating envelope

There are even some cases where the actual mating envelope's location shall be held stationary relative to the higher precedence datum(s). In addition, when calculating positional tolerance deviations, there are circumstances where a "restrained" actual mating envelope shall be used. We'll explain these applications in greater detail in later sections.

In practice, the largest cylindrical gage pin that can fit in a hole can often simulate the hole's actual mating envelope. The actual mating envelope for a slot can sometimes be approximated by the largest stack of Webber (or "Jo") blocks that can fit. External features are a little tougher, but their actual mating envelopes might be simulated with cylindrical ring gages or Webber block sandwiches.

Cases calling for a restrained actual mating envelope really challenge hard gaging methods. Traditionally, inspectors have fixtured parts to coordinate measuring machine (CMM) tables (on their datum feature surfaces) and held cylindrical gage pins in a drill chuck in the CMM's ram. This practice is only marginally satisfactory, even where relatively large tolerances are involved.

5.6.5 Alternative "Center Method" for MMC or LMC

As we explained in section 5.6.3, Level 2, 3, and 4 geometric tolerances applied to features of size and modified to MMC or LMC establish virtual condition boundaries for the features. Chapter 19 explains how functional gages use pins, holes, slots, tabs, and other physical shapes to simulate the MMC virtual condition boundaries, emulating worst-case features on the mating part as if each mating feature were manufactured at its MMC with its worst allowable orientation and location. However, without a functional gage or sophisticated CMM software, it might be very difficult to determine whether or not a feature encroaches beyond its virtual condition boundary. Therefore, the standards provide an alternative method that circumvents virtual boundaries, enabling more elementary inspection techniques. We call this alternative the *center method*.

Where a Level 2, 3, or 4 geometric tolerance is applied to a feature of size in an MMC or LMC context, the tolerance may optionally be interpreted as in an RFS context—that is, it establishes a central tolerance zone, within which a geometric element derived from the feature shall be contained. However, unlike in the RFS context, the MMC or LMC tolerance zone shall provide control approximating that of the virtual condition boundary. To accomplish this, the size of the tolerance zone shall adjust according to the feature's actual size.

5.6.5.1 Level 3 and 4 Adjustment—Actual Mating/Minimum Material Sizes

The adjustment for Level 3 and 4 tolerances is very simple: The tolerance zone is uniformly enlarged by *bonus tolerance*—a unit value to be added to the specified geometric tolerance.

At MMC—Bonus tolerance equals the arithmetic difference between the feature's actual mating size and its specified MMC size limit.

Actual mating size is **the dimensional value of the actual mating envelope** (defined in section 5.6.4.2), and represents the worst-case mating potential for a feature of size. See Fig. 5-38.

Thus, actual mating size is the most suitable measure of actual size in clearance-fit applications or for most features having a boundary of perfect form at MMC. For a hole having an actual mating size $\varnothing.001$ larger than its MMC, $\varnothing.001$ of bonus tolerance is added to the specified geometric tolerance. Likewise, for a tab $.002$ smaller than its MMC, $.002$ is added to the specified tolerance value.

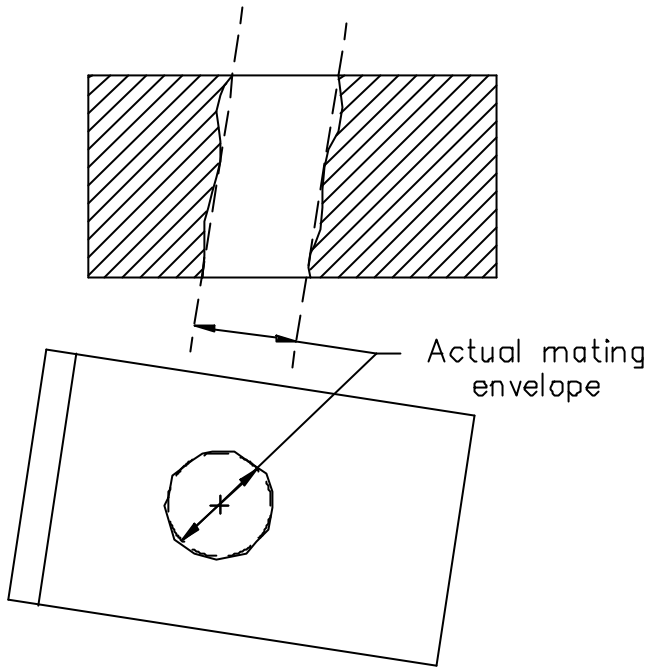


Figure 5-38 Actual mating envelope of an imperfect hole

At LMC—Bonus tolerance equals the arithmetic difference between the feature’s actual minimum material size and its specified LMC size limit.

Actual minimum material size is the dimension of the actual minimum material envelope.

Actual minimum material envelope is defined according to the type of feature, as follows:

(a) *For an External Feature.* A similar perfect feature counterpart of largest size, which can be inscribed within the feature so that it just contacts the surface(s).

(b) *For an Internal Feature.* A similar perfect feature counterpart of smallest size, which can be circumscribed about the feature so that it just contacts the surface(s).

In certain cases, the orientation, or the orientation and location of an actual minimum material envelope shall be restrained to one or two datums.

Notice from Fig. 5-39 that the actual minimum material envelope is the inverse of the actual mating envelope. While the actual mating envelope resides in the “air” at the surface of a feature, the actual minimum material envelope is embedded in part material. That makes it impossible to simulate with tangible gages. The actual minimum material envelope can only be approximated by scanning point data into a computer and modeling the surface—a process called *virtual gaging* or *softgaging*.

Let’s consider a cast boss that must have an adequate “shell” of part material all around for cleanup in a machining operation. If its LMC size limit is $\text{Ø}.387$ and its actual minimum material size is $\text{Ø}.390$, a “bonus” of $\text{Ø}.003$ shall be added to the specified geometric tolerance.

In section 5.6.3.1, we described some rare features having boundaries of perfect form at both MMC and LMC. Those features have an actual mating envelope and actual mating size that’s used in the context of the geometric tolerance and/or datum reference at MMC. For the LMC context, the same feature additionally has an actual minimum material envelope and actual minimum material size. As might be apparent from Fig. 5-39, the greater the feature’s form deviation (and orientation deviation, as applicable), the greater is the difference between the two envelopes and sizes.

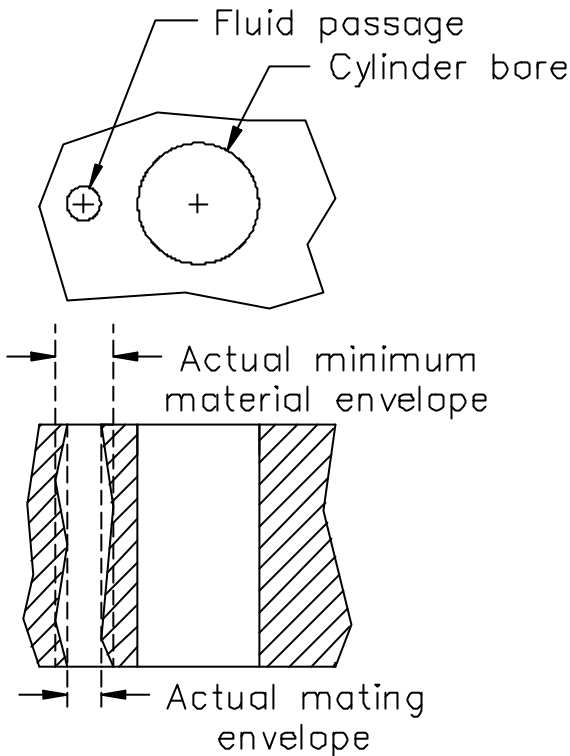


Figure 5-39 Actual minimum material envelope of an imperfect hole

5.6.5.2 Level 2 Adjustment—Actual Local Sizes

Since Level 3 and 4 tolerances impose no additional form controls, the “center method” permits use of a uniform tolerance zone and an all-encompassing envelope size. Level 2 tolerances, however, are intended to control feature form. Thus, the tolerance zone must interact with actual feature size independently at each cross section of the feature. Though the effective control is reduced from 3-D down to 2-D, inspection is paradoxically more complicated. Perhaps because there’s rarely any reason to use the alternative “center method” for Level 2 tolerances, neither Y14.5 nor the Math Standard defines it thoroughly. In our own following explanations, we’ve extended actual mating/minimum material envelope principles to emulate accurately the controls imposed by Level 2 virtual condition boundaries.

Straightness of a Cylindrical Feature at MMC—The central tolerance zone is bounded by a revolute, within which the derived median line shall be contained. At each cross-sectional slice, the diameter of the tolerance zone varies according to the actual mating local size. Within any plane perpendicular to the axis of the actual mating envelope, *actual mating local size* is the diameter of the largest perfect circle that can be inscribed within an internal feature, or the smallest that can be circumscribed about an external feature, so that it just contacts the feature surface. The straightness tolerance zone local diameter equals the stated straightness tolerance value plus the diametral difference between the actual mating local size and the feature’s MMC limit size.

At any cross section of the pin shown in Fig. 5-26, as the pin’s actual mating local size approaches MMC ($\varnothing.063$), the straightness tolerance zone shrinks to the specified diameter ($\varnothing.010$). Conversely, as the pin’s actual mating local size approaches LMC ($\varnothing.062$), the tolerance zone expands to $\varnothing.011$. Either way, for any pin satisfying both its size limits and its straightness tolerance, the surface of the pin will nowhere encroach beyond its $\varnothing.073$ virtual condition boundary.

Straightness of a Cylindrical Feature at LMC—The central tolerance zone is bounded by a revolute, within which the derived median line shall be contained. At each cross-sectional slice, the diameter of the tolerance zone varies according to the actual minimum material local size. Within any plane perpendicular to the axis of the actual minimum material envelope, *actual minimum material local size* is the diameter of the smallest perfect circle that can be circumscribed about an internal feature, or the largest that can be inscribed within an external feature, so that it just contacts the feature surface. The straightness tolerance zone local diameter equals the stated straightness tolerance value plus the diametral difference between the actual minimum material local size and the feature’s LMC limit size.

Flatness of a Width-Type Feature at MMC or LMC—The central tolerance zone is bounded by two mirror image imperfect planes, within which the derived median plane shall be contained. At each point on the derived median plane, the corresponding local width of the tolerance zone equals the stated flatness tolerance value plus the difference between the feature’s actual local size and the feature’s MMC (in an MMC context) or LMC (in an LMC context) limit size. *Actual local size* is the distance between two opposite surface points intersected by any line perpendicular to the center plane of the actual mating envelope (MMC context), or of the actual minimum material envelope (LMC context).

At any cross section of the washer shown in Fig. 5-27, as the washer’s actual local size approaches MMC (.034), the flatness tolerance zone shrinks to the specified width (.020). Conversely, as the washer’s actual local size approaches LMC (.030), the tolerance zone expands to .024. Either way, for any washer satisfying both its size limits and its flatness tolerance, neither surface of the washer will anywhere encroach beyond the .054 virtual condition boundary.

5.6.5.3 Disadvantages of Alternative “Center Method”

By making the geometric tolerance interact with the feature’s actual size, the “center method” closely emulates the preferred (virtual condition) *boundary method*. For a hypothetical perfectly formed and perfectly oriented feature, the two methods yield identical conformance results. For imperfect features, however, the Math Standard offers a detailed explanation of how the “center method” might reject a barely conforming feature, or worse, accept a slightly out-of-tolerance feature. Be very careful with older CMMs and surface plate techniques roughly employing the “center method.” Generally, the boundary method will be more forgiving of marginal features, but will never accept a nonfunctional one.

The Math Standard uses actual mating size for all actual envelope size applications in RFS and MMC contexts, and applies actual minimum material size in all LMC contexts. Y14.5 does not yet recognize actual minimum material size and uses actual mating size in all contexts. In an LMC context, local voids between the feature surface and the actual mating envelope represent portions of the feature at risk for violating the LMC virtual condition boundary. Since actual mating size is unaffected by such voids, it can’t provide accurate emulation of the LMC virtual condition boundary. This discrepancy causes some subtle contradictions in Y14.5’s LMC coverage, which this chapter circumvents by harmonizing with the Math Standard.

5.6.6 Inner and Outer Boundaries

Many types of geometric tolerances applied to a feature of size, for example, runout tolerances, establish an *inner boundary* and/or *outer boundary* beyond which the feature surface(s) shall not encroach. Since the standards don’t define feature controls in terms of these inner and outer boundaries, the boundaries are considered the result of other principles at work. See section 5.12.9. They’re sometimes useful in tolerance calculations. See Chapter 9, section 9.3.3.3.

5.6.7 When Do We Use a Material Condition Modifier?

The functional differences between RFS, MMC, and LMC contexts should now be clear. Obviously, an MMC or LMC modifier can only be associated with a feature of size or a bounded feature. A modifier can only apply to a datum reference in a feature control frame, or to a straightness, flatness, orientation, or positional tolerance in a feature control frame. In all such places, we recommend designers use a modifier, either MMC or LMC, unless there is a specific requirement for the unique properties of RFS.

MMC for clearance fits—Use MMC for any feature of size that assembles with another feature of size on a mating part and the foremost concern is that the two mating features clear (not interfere with) each other. Use MMC on any datum reference where the datum feature of size itself makes a clearance fit, and the features controlled to it likewise make clearance fits. Because clearance fits are so common, and because MMC permits functional gaging, many designers have wisely adopted MMC as a default. (Previously, Y14.5 made it the default.) Where a screw thread must be controlled with GD&T or referenced as a datum, try to use MMC.

LMC for minimum stock protection—Use LMC where you must guarantee a minimum “shell” of material all over the surface of any feature of size, for example:

- For a cast, forged, or rough-machined feature to assure stock for cleanup in a subsequent finishing operation
- For a nonmating bore, fluid passage, etc., to protect minimum wall thickness for strength
- For a nonmating boss around a hole, to protect minimum wall thickness for strength
- For the gaging features of a functional gage to assure the gage won't clear a nonconforming part
- For a boss that shall completely cover a hole in the mating part

Where a fluid passage is drilled next to a cylinder bore, as shown in Fig. 5-39, the designer may be far more concerned with the thinnest wall between them than with the largest pin that can fit into the fluid passage. An MMC virtual condition boundary can't prevent a void deep down inside the hole created by an errant drill. In cases such as this, where we're more concerned with presence of material than with a clearance fit, LMC is preferred.

You don't often see LMC applied to datum features, but consider an assembly where datum features of size pilot two mating parts that must be well centered to each other. LMC applied to both datum features guarantees a minimal offset between the two parts regardless of how loose the fit. This is a valuable technique for protecting other mating interfaces in the assembly. And on functional gages, LMC is an excellent choice for datum references.

Compared to MMC, LMC has some disadvantages in gaging and evaluation. It's difficult to assess the actual minimum material size. Functional gages cannot be used.

RFS for centering—RFS is obsessed with a feature's center to the point of ignorance of the feature's actual size. In fact, RFS allows no dynamic interaction between size and location or between size and orientation of a feature. However, this apparent limitation of RFS actually makes it an excellent choice for self-centering mating interfaces where the mating features always fit together snugly and center on each other regardless of their actual mating sizes. Examples of self-centering mating interfaces include the following:

- Press fits
- Tapers, such as Morse tapers and countersinks for flat-head screws
- Elastic parts or elastic intermediate parts, such as O-rings
- An adjustable interface where an adjusting screw, shim, sleeve, etc., will be used in assembly to center a mating part
- Glued or potted assemblies

In such interfaces, it's obvious to the designer that the actual sizes of the mating features have no relevance to the allowable orientation or positional tolerance for those features. In the case of an external O-ring groove, for example, MMC would be counterproductive, allowing eccentricity to increase as diameter size gets smaller. Here, RFS is the wiser choice.

There are certain geometric characteristics, such as runout and concentricity, where MMC and LMC are so utterly inappropriate that the rules prohibit material condition modifiers. For these types of tolerances, RFS always applies.

Y14.5 allows RFS to be applied to any tolerance and any datum reference in conjunction with any feature of size having a defined center. In fact, RFS principles now apply by default in the absence of any material condition modifier. (Note that's different from earlier editions of Y14.5.) But RFS is versatile like a monkey wrench. You can use it on everything, but for most of your choices, there is a more suitable tool (MMC or LMC) that will fit the work better and cost less. For example, RFS is a poor choice in clearance-fit mating interfaces because it doesn't allow dynamic tolerance interaction. That means smaller tolerances, usable parts rejected, and higher costs.

Remember that RFS principles are based on a feature's center. To verify most RFS controls, the inspector must derive the center(s) of the involved feature(s). Functional gages with fixed-size elements cannot be used with RFS. RFS applied to a feature pattern referenced as a datum, or to any type of feature for which Y14.5 doesn't define a center, is sure to provoke a debate somewhere and waste more money.

FAQ: *Should I use RFS instead of MMC whenever I need greater precision?*

A: Not always. A tolerance applied RFS is more restrictive than an equal tolerance modified to MMC. That fact leads to the common misconception that RFS is therefore a more precise tool. This is like comparing the precision of a saw and a hammer. We've tried to emphasize the differences between MMC, LMC, and RFS. Each tool is the most precise for its intended function. RFS works differently from MMC, often with different rules and different results. As a broadly general statement based on drawings we've seen, MMC is hugely underused, LMC is somewhat underused, and RFS is hugely overused.

FAQ: *Why, then, is RFS now the default?*

A: For what it's worth, the default now agrees with the ISO 8015 standard. It's like "training wheels" for users who might fail to comprehend properly and apply RFS where it's genuinely needed.

5.7 Size Limits (Level 1 Control)

For every feature of size, the designer shall specify the largest and the smallest the feature can be. In section 5.6.1, we discussed the exact requirements these size limits impose on the feature. The standards provide three options for specifying size limits on the drawing: symbols for limits and fits, *limit dimensioning*, and *plus and minus tolerancing*. Where tolerances directly accompany a dimension, it's important to coordinate the number of decimal places expressed for each value to prevent confusion. The rules depend on whether the dimension and tolerance values are expressed in inches or millimeters.

5.7.1 Symbols for Limits and Fits

Inch or metric size limits may be indicated using a standardized system of preferred sizes and fits. Using this system, standard feature sizes are found in tables in ANSI B4.1 (inch) or ANSI B4.2 (metric), then expressed on the drawing as a basic size followed by a tolerance symbol, for example, $\varnothing.625$ LC5 or 30 f7.

5.8.1 Straightness Tolerance for Line Elements

Where a straightness tolerance feature control frame is placed according to option (b) in Table 5-1 (leader-directed to a feature surface or attached to an extension line of a feature surface), the tolerance controls only line elements of that surface. The feature control frame may only appear in a view where the controlled surface is represented by a straight line. The tolerance specifies a tolerance zone plane containing a tolerance zone bounded by two parallel lines separated by a distance equal to the tolerance value. As the tolerance zone plane sweeps the entire feature surface, the surface's intersection with the plane shall everywhere be contained within the tolerance zone (between the two lines). Within the plane, the orientation and location of the tolerance zone may adjust continuously to the part surface while sweeping. See Fig. 5-40.

Of a Cylindrical or Conical Feature—The straightness tolerance zone plane shall be swept radially about the feature's axis, always containing that axis. (Note that the axis of a cone isn't explicitly defined.) Within the rotating tolerance zone plane, the tolerance zone's orientation relative to the feature axis may adjust continuously. Since Rule #1 already controls a cylinder's surface straightness within size limits, a separate straightness tolerance applied to a cylindrical feature must be less than the size tolerance to be meaningful.

Of a Planar Feature—The orientation and sweep of the tolerance zone plane is not explicitly related to any other part feature. The plane is merely implied to be parallel to the view plane and swept perpendicular to the view plane (toward and away from the viewer). Again, the zone itself may tilt and shift within the tolerance zone plane to accommodate gross surface undulations. See Fig. 5-40. Where it's important to relate the tolerance zone plane to datums, specify instead a profile of a line tolerance, as described in section 5.13.8.

For a width-type feature of size, Rule #1 automatically limits the flatness and straightness deviation of each surface—no extra charge. Thus, to have any meaning, a separate straightness tolerance applied to either single surface must be less than the total size tolerance.

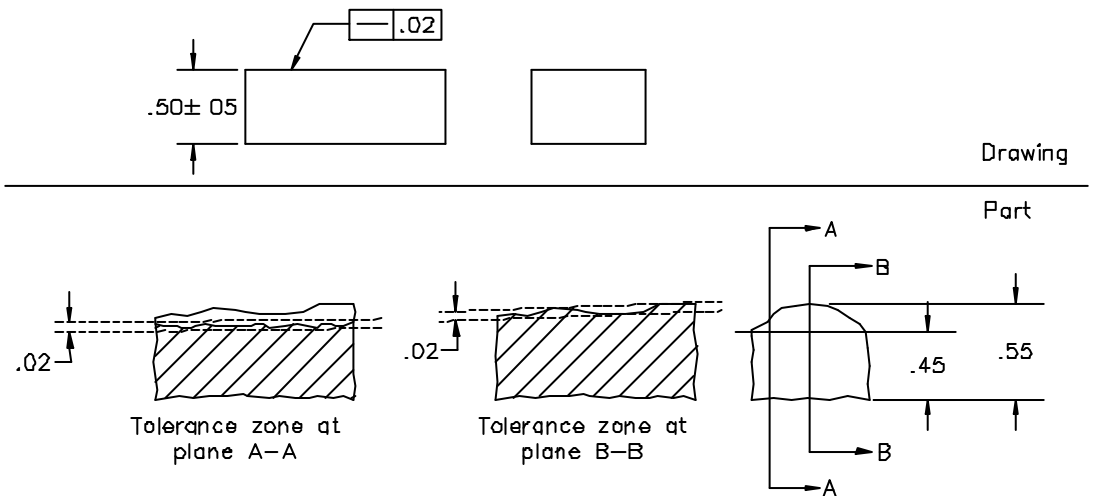


Figure 5-40 Straightness tolerance for line elements of a planar feature

5.8.2 Straightness Tolerance for a Cylindrical Feature

A straightness tolerance feature control frame placed according to options (a) or (d) in Table 5-1 (associated with a diameter dimension) replaces Rule #1's requirement for perfect form at MMC with a separate tolerance controlling the overall straightness of the cylindrical feature. Where the tolerance is modified to MMC or LMC, it establishes a Level 2 virtual condition boundary as described in section 5.6.3.1 and Figs. 5-17(b) and 5-18(b). Alternatively, the "center method" described in section 5.6.5.2 may be applied to a straightness tolerance at MMC or LMC, but there's rarely any benefit to offset the added complexity. Unmodified, the tolerance applies RFS and establishes a central tolerance zone as described in section 5.6.4.1, within which the feature's derived median line shall be contained.

5.8.3 Flatness Tolerance for a Single Planar Feature

Where a flatness tolerance feature control frame is placed according to options (b) or (c) in Table 5-1 (leader-directed to a feature or attached to an extension line from the feature), the tolerance applies to a single nominally flat feature. The flatness feature control frame may be applied only in a view where the element to be controlled is represented by a straight line. This specifies a tolerance zone bounded by two parallel planes separated by a distance equal to the tolerance value, within which the entire feature surface shall be contained. The orientation and location of the tolerance zone may adjust to the part surface. See Fig. 5-41. A flatness tolerance cannot control whether the surface is fundamentally concave, convex, or stepped; just the maximum range between its highest and lowest undulations.

For a width-type feature of size, Rule #1 automatically limits the flatness deviation of each surface. Thus, to have any meaning, a separate flatness tolerance applied to either single surface must be less than the total size tolerance.

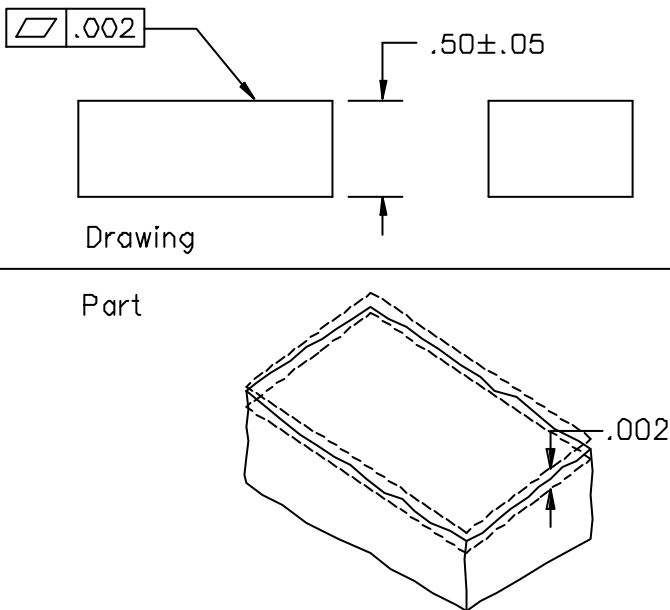


Figure 5-41 Flatness tolerance for a single planar feature

5.8.4 Flatness Tolerance for a Width-Type Feature

A flatness tolerance feature control frame placed according to options (a) or (d) in Table 5-1 (associated with a width dimension) replaces Rule #1's requirement for perfect form at MMC with a separate tolerance controlling the overall flatness of the width-type feature. Where the tolerance is modified to MMC or

LMC, it establishes a Level 2 virtual condition boundary as described in section 5.6.3.1 and Figs. 5-17(b) and 5-18(b). Alternatively, the “center method” described in section 5.6.5.2 may be applied to a flatness tolerance at MMC or LMC, but there’s rarely any benefit to offset the added complexity. Unmodified, the tolerance applies RFS and establishes a central tolerance zone as described in section 5.6.4.1, within which the feature’s derived median plane shall be contained.

This application of a flatness tolerance is an extension of the principles of section 5.8.2. Y14.5 suggests an equivalent control using the “straightness” characteristic symbol. We think it’s inappropriate to establish a parallel plane tolerance zone using the straightness symbol. However, where strict adherence to Y14.5 is needed, the “straightness” symbol should be used.

5.8.5 Circularity Tolerance

A circularity tolerance controls a feature’s *circularity* (roundness) at individual cross sections. Thus, a circularity tolerance may be applied to any type of feature having uniformly circular cross sections, including spheres, cylinders, revolutes (such as cones), tori (doughnut shapes), and bent rod and tubular shapes.

Where applied to a nonspherical feature, the tolerance specifies a tolerance zone plane containing an annular (ring-shaped) tolerance zone bounded by two concentric circles whose radii differ by an amount equal to the tolerance value. See Fig. 5-42. The tolerance zone plane shall be swept along a simple, nonself-

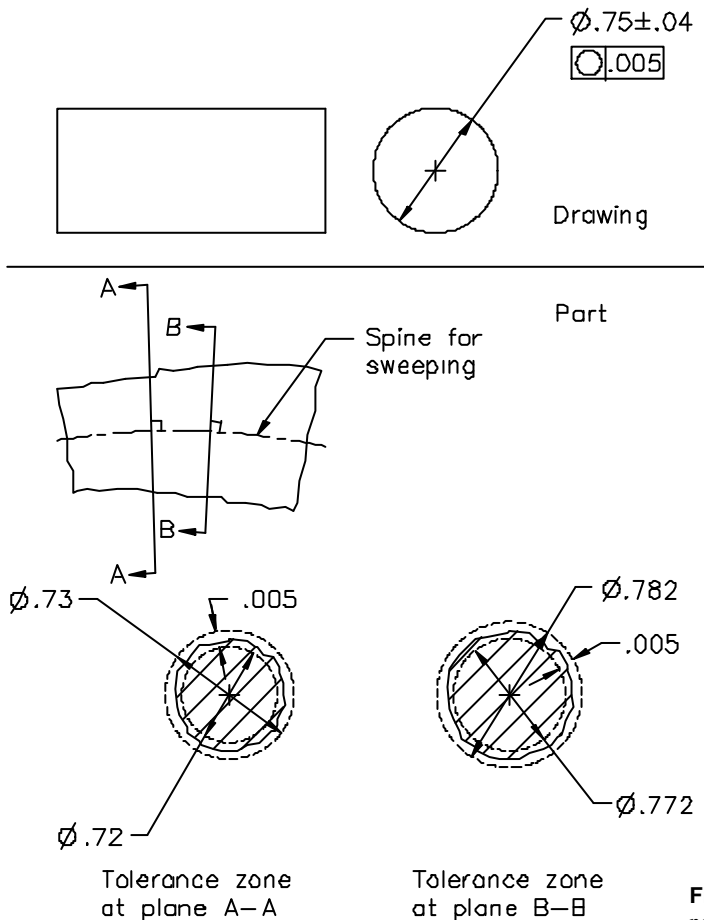


Figure 5-42 Circularity tolerance (for nonspherical features)

intersecting, tangent-continuous curve (spine). At each point along the spine, the tolerance zone plane shall be perpendicular to the spine and the tolerance zone centered on the spine. As the tolerance zone plane sweeps the entire feature surface, the surface's intersection with the plane shall everywhere be contained within the annular tolerance zone (between the two circles). While sweeping, the tolerance zone may continually adjust in overall size, but shall maintain the specified radial width. This effectively removes diametral taper from circularity control. Additionally, the spine's orientation and curvature may be adjusted within the aforementioned constraints. This effectively removes axial straightness from circularity control. The circularity tolerance zone need not be concentric with either size limit boundary.

A circularity tolerance greater than the total size tolerance has no effect. A circularity tolerance between the full size tolerance and one-half the size tolerance limits only single-lobed (such as D-shaped and egg-shaped) deviations. A circularity tolerance must be less than half the size tolerance to limit multi-lobed (such as elliptical and tri-lobed) deviations.

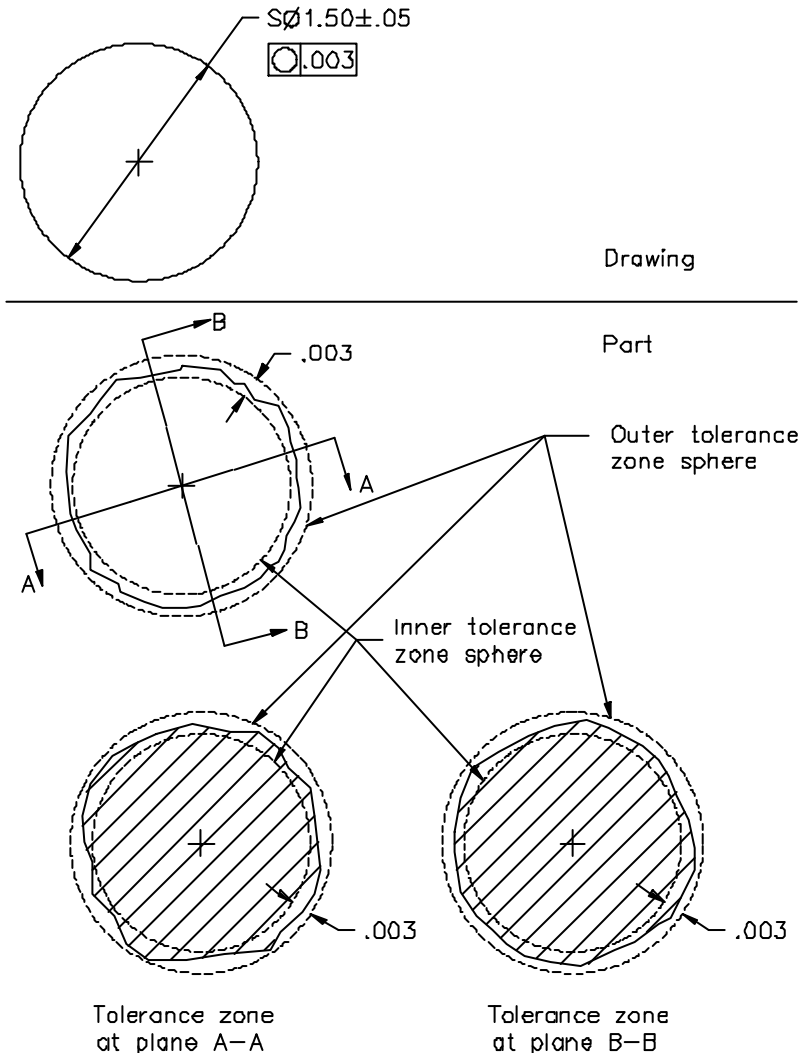


Figure 5-43 Circularity tolerance applied to a spherical feature

Note that Y14.5's explanation refers to an "axis," which could be interpreted as precluding curvature of the spine. Either way, most measuring equipment can only inspect circularity relative to a straight line.

5.8.5.1 Circularity Tolerance Applied to a Spherical Feature

The standards also use a tolerance zone plane to explain a circularity tolerance applied to a spherical feature. Since any pair of surface points can be included in such a plane, their respective distances from a common center shall not differ by more than the circularity tolerance. Therefore, the explanation can be simplified as follows: The tolerance specifies a tolerance zone bounded by two concentric spheres whose radii differ by an amount equal to the tolerance value. The tolerance zone may adjust in overall size, but shall maintain the specified radial width. All points on the considered spherical feature shall be contained within the tolerance zone (between the two spheres). See Fig. 5-43. Since the tolerance zone need not be concentric with either size limit boundary, a circularity tolerance must be less than half the size tolerance to limit multi-lobed form deviations.

5.8.6 Cylindricity Tolerance

A **cylindricity tolerance is a composite control of form that includes circularity, straightness, and taper of a cylindrical feature.** A cylindricity tolerance specifies a tolerance zone bounded by two concentric cylinders whose radii differ by an amount equal to the tolerance value. See Fig. 5-44. The entire feature surface shall be contained within the tolerance zone (between the two cylinders). The tolerance zone cylinders may adjust to any diameter, provided their radial separation remains equal to the tolerance value. This effectively removes feature size from cylindricity control. As with circularity tolerances, a cylindricity tolerance must be less than half the size tolerance to limit multi-lobed form deviations. Since neither a cylindricity nor a circularity tolerance can nullify size limits for a feature, there's nothing to be gained by modifying either tolerance to MMC or LMC.

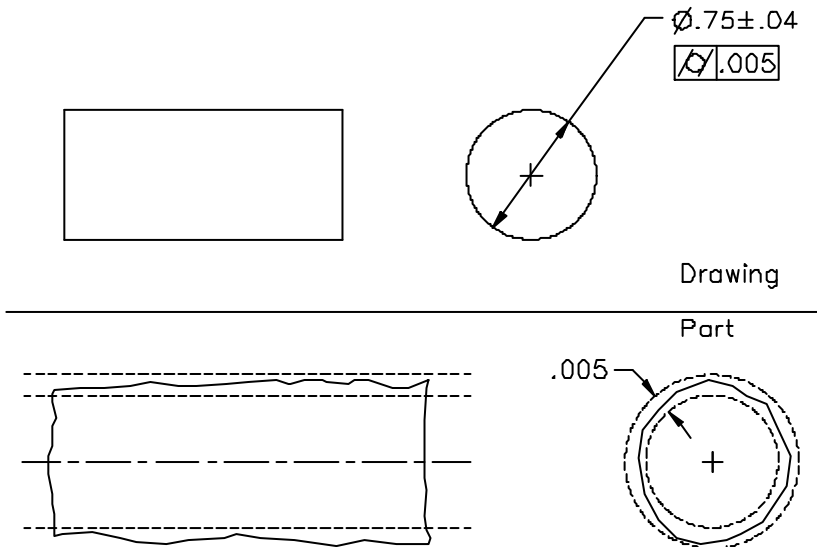


Figure 5-44 Cylindricity tolerance

5.8.7 Circularity or Cylindricity Tolerance with Average Diameter

The thin-wall nylon bushing shown in Fig. 5-45 is typical of a nonrigid part having diameters that fit rather closely with other parts in assembly. If customary diameter size limits were specified, no matter how liberal, their inherent circularity control would be overly restrictive for the bushing in its free state (unassembled). The part's diameters in the free state cannot and need not stay as round as they'll be once restrained in assembly. We need a different way to control size-in-assembly, while at the same time guarding against collapsed or grotesquely out-of-round bushings that might require excessive assembly force or jam in automated assembly equipment.

The solution is to specify limits for the feature's *average diameter* along with a generous circularity tolerance. Where a diameter tolerance is followed by the note AVG, the size limit boundaries described in section 5.6.1 do not apply. Instead, the tolerance specifies limits for the feature's average diameter. *Average diameter* is defined somewhat nebulously as the average of at least four two-point diameter measurements. A contact-type gage may deflect the part, yielding an unacceptable measurement. Where practicable, average diameter may be found by dividing a peripheral tape measurement by π . When the part is restrained in assembly, its effective mating diameter should correspond closely to its average diameter in the free state.

Though we told you our nylon bushing is a nonrigid part, the drawing itself (Fig. 5-45) gives no indication of the part's rigidity. In particular, there's no mention of restraint for verification as described in section 5.5.1. Therefore, according to Fundamental Rule (1), a drawing user shall interpret all dimensions and tolerances, including the circularity tolerance, as applying in the free state. The standard

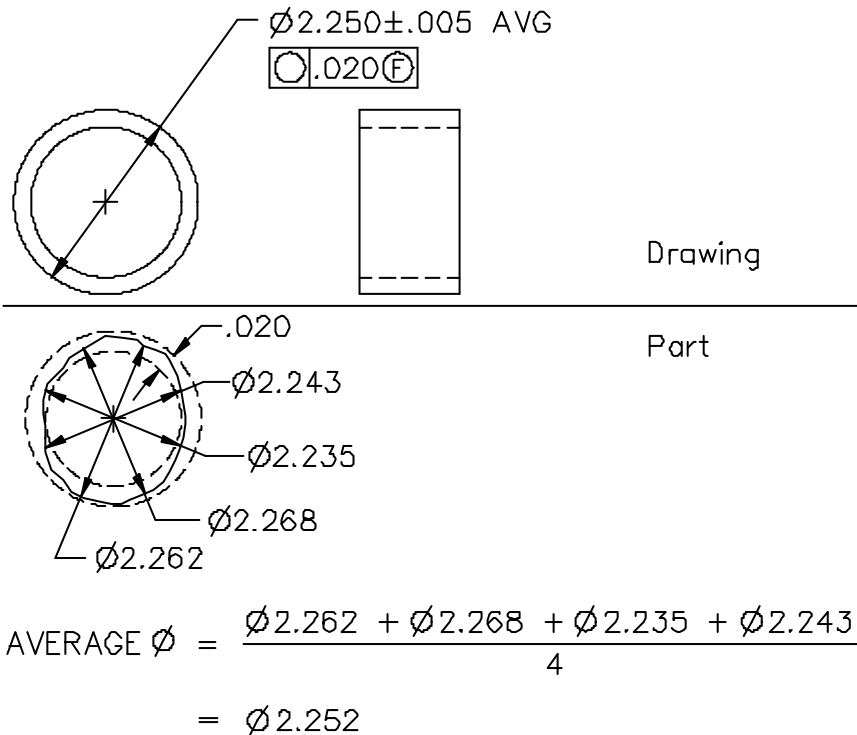


Figure 5-45 Circularity tolerance with average diameter

implies average diameter can only be used in conjunction with the “free state” symbol. For that reason only, we’ve added the “free state” symbol after the circularity tolerance value. A feature’s conformance to both tolerances shall be evaluated in the free state—that is, with no external forces applied to affect its size or form.

The same method may be applied to a longer nonrigid cylindrical feature, such as a short length of vinyl tubing. Simply specify a relatively liberal cylindrical tolerance modified to “free state,” along with limits for the tube’s average diameter.

5.8.8 Application Over a Limited Length or Area

Some designs require form control over a limited length or area of the surface, rather than the entire surface. In such cases, draw a heavy chain line adjacent to the surface, basically dimensioned for length and location as necessary. See Fig. 5-46. The form tolerance applies only within the limits indicated by the chain line.

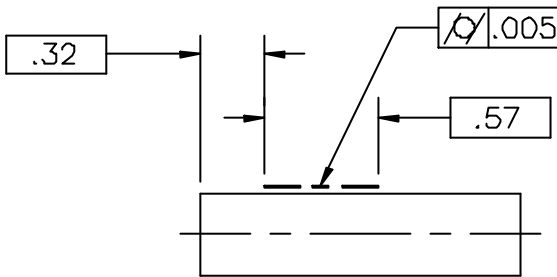


Figure 5-46 Circularity tolerance applied over a limited length

5.8.9 Application on a Unit Basis

There are many features for which the design could tolerate a generous amount of form deviation, provided that deviation is evenly distributed over the total length and/or breadth of the feature. This is usually the case with parts that are especially long or broad in proportion to their cross-sectional areas.

The 6' piece of bar stock shown in Fig. 5-47 could be severely bowed after heat-treating. But if the bar is then sawed into 6" lengths, we're only concerned with how straight each 6" length is. The laminated honeycomb panel shown in Fig. 5-48 is an airfoil surface. Gross flatness of the entire surface can reach .25". However, any abrupt surface variation within a relatively small area, such as a dent or wrinkle, could disturb airflow over the surface, degrading performance.

These special form requirements can be addressed by specifying a form (only) tolerance on a unit basis. The size of the unit length or area, for example 6.00 or 3.00 X 3.00, is specified to the right of the form tolerance value, separated by a slash. This establishes a virtual condition boundary or tolerance zone as usual, except limited in length or area to the specified dimension(s). As the limited boundary or tolerance zone sweeps the entire length or area of the controlled feature, the feature's surface or derived element (as applicable) shall conform at every location.

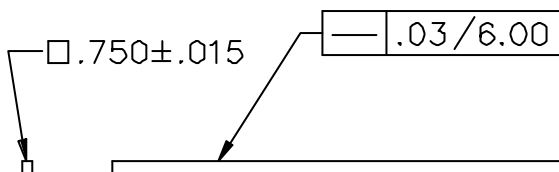


Figure 5-47 Straightness tolerance applied on a unit basis

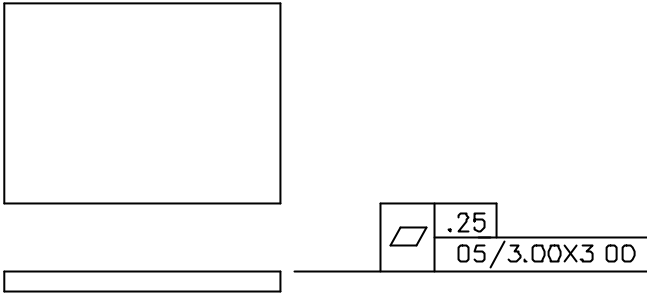


Figure 5-48 Flatness tolerance applied on a unit basis

Since the bar stock in Fig. 5-47 may be bowed no more than .03" in any 6" length, its accumulated bow over 6' cannot exceed 4.38". The automated saw can handle that. In contrast, the airfoil in Fig. 5-48 may be warped as much as .05" in any 3 x 3" square. Its maximum accumulated warp over 36" is 6.83". A panel that bowed won't fit into the assembly fixture. Thus, for the airfoil, a compound feature control frame is used, containing a single "flatness" symbol with two stacked segments. The upper segment specifies a flatness tolerance of .25" applicable to the entire surface. The lower segment specifies flatness per unit area, not to exceed .05" in any 3 x 3" square. Obviously, the per-unit tolerance value must be less than the total-feature tolerance.

5.8.10 Radius Tolerance

A *radius* (plural, *radii*) is a portion of a cylindrical surface encompassing less than 180° of arc length. A radius tolerance, denoted by the symbol R, establishes a zone bounded by a minimum radius arc and a maximum radius arc, within which the entire feature surface shall be contained. As a default, each arc shall be tangent to the adjacent part surfaces. See Fig. 5-49. Where a center is drawn for the radius, as in Fig. 5-50, two concentric arcs of minimum and maximum radius bound the tolerance zone. Within the tolerance zone, the feature's contour may be further refined with a "controlled radius" tolerance, as described in the following paragraph.

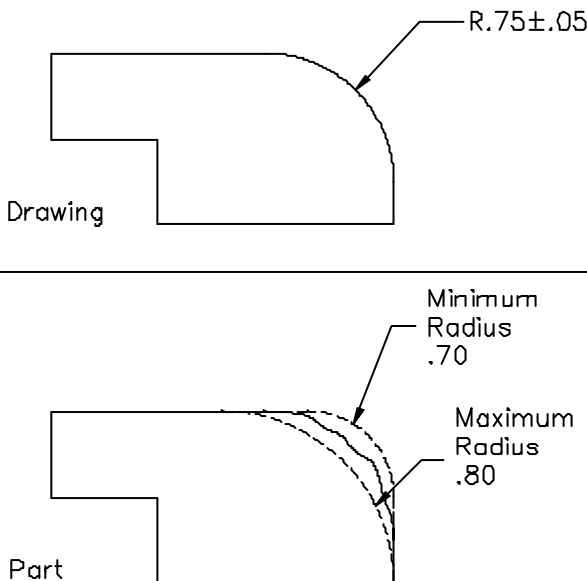


Figure 5-49 Radius tolerance zone (where no center is drawn)

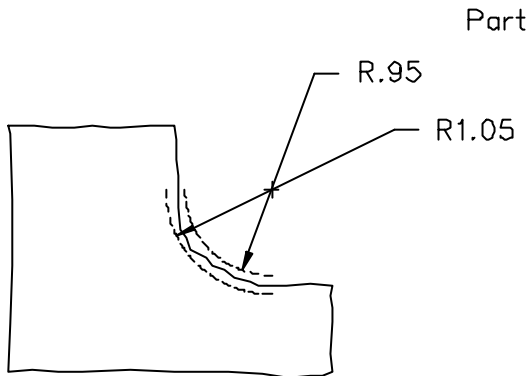
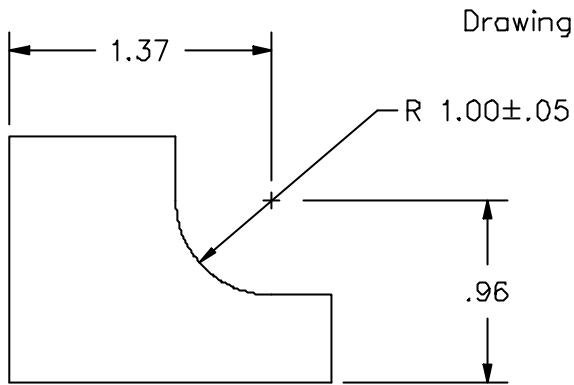


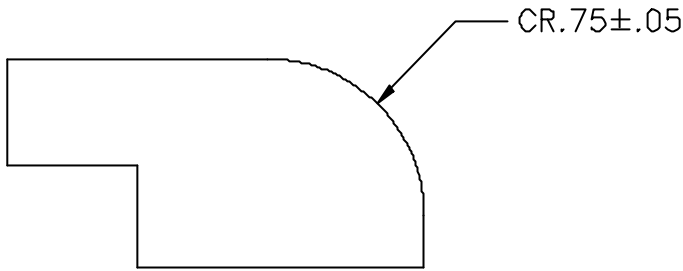
Figure 5-50 Radius tolerance zone where a center is drawn

5.8.10.1 Controlled Radius Tolerance

Where the symbol CR is applied to a radius, the tolerance zone is as described in section 5.8.10, but there are additional requirements for the surface. The surface contour shall be **a fair curve without reversals**. We interpret this to mean a tangent-continuous curve that is everywhere concave or convex, as shown in Fig. 5-51. Before the 1994 Revision of Y14.5, there was no CR symbol, and these additional controls applied to every radius tolerance. The standard implies that CR can only apply to a tangent radius, but we feel that by extension of principle, the refinement can apply to a “centered” radius as well.

5.8.11 Spherical Radius Tolerance

A *spherical radius* is a portion of a spherical surface encompassing less than 180° of arc length. A spherical radius tolerance, denoted by the symbol SR, establishes a zone bounded by a minimum radius arc and a maximum radius arc, within which the entire feature surface shall be contained. As a default, each arc shall be tangent to the adjacent part surfaces. Where a center is drawn for the radius, two concentric spheres of minimum and maximum radius bound the tolerance zone. The standards don’t address “controlled radius” refinement for a spherical radius.



Drawing

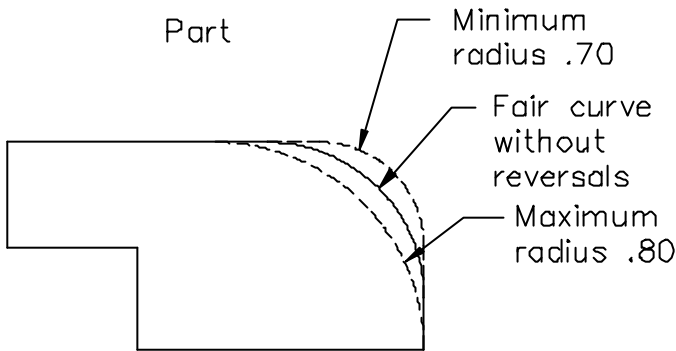


Figure 5-51 Controlled radius tolerance zone

5.8.12 When Do We Use a Form Tolerance?

As we explain in the next section, datum simulation methods can accommodate warped and/or out-of-round datum features. However, datum simulation will usually be more repeatable and error free with well-formed datum features. We discuss this further in section 5.9.12.

As a general rule, apply a form (only) tolerance to a nondatum feature only where there is some risk that the surface will be manufactured with form deviations severe enough to cause problems in subsequent manufacturing operations, inspection, assembly, or function of the part. For example, a flatness tolerance might be appropriate for a surface that seals with a gasket or conducts heat to a heat sink. A roller bearing might be controlled with a cylindricity tolerance. A conical bearing race might have both a straightness of surface elements tolerance and a circularity tolerance. However, such a conical surface might be better controlled with profile tolerancing as explained in section 5.13.11.

FAQ: *If feature form can be controlled with profile tolerances, why do we need all the form tolerance symbols?*

A: In section 5.13.11, we explain how profile tolerances may be used to control straightness or flatness of features. While such applications are a viable option, most drawing users prefer to see the “straightness” or “flatness” characteristic symbols because those symbols convey more information at a glance.

5.9 Datuming

5.9.1 What Is a Datum?

According to the dictionary, a *datum* is a single piece of information. In logic, a datum may be a given starting point from which conclusions may be drawn. In surveying, a datum is any level surface, line, or point used as a reference in measuring. Y14.5's definition embraces all these meanings.

A datum is a theoretically exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature. A datum is the origin from which the location or geometric characteristics of features of a part are established.

A datum feature is an actual feature of a part that is used to establish a datum.

A *datum reference* is an alpha letter appearing in a compartment following the geometric tolerance in a feature control frame. It specifies a datum to which the tolerance zone or acceptance boundary is basically related. A feature control frame may have zero, one, two, or three datum references.

The diagram in Fig. 5-52 shows that a “datum feature” begets a “true geometric counterpart,” which begets a “datum,” which is the building block of a “datum reference frame,” which is the basis for tolerance zones for other features. Even experts get confused by all this, but keep referring to Fig. 5-52 and we'll sort it out one step at a time.

5.9.2 Datum Feature

In section 5.1.5, we said the first step in GD&T is to “identify part surfaces to serve as origins and provide specific rules explaining how these surfaces establish the starting point and direction for measurements.” Such a part surface is called a *datum feature*.

According to the Bible, about five thousand years ago, God delivered some design specifications for a huge water craft to a nice guy named Noah. “Make thee an ark of gopher wood... The length of the ark shall be three hundred cubits, the breadth of it fifty cubits, and the height of it thirty cubits.” Modern scholars are still puzzling over the ark's material, but considering the vessel would be half again bigger than a football field, Noah likely had to order material repeatedly, each time telling his sons, “Go fer wood.” For the “height of thirty cubits” dimension, Noah's sons, Shem and Ham, made the final measurement from the level ground up to the top of the “poop” deck, declaring the measured size conformed to the Holy Specification “close enough.” Proudly looking on from the ground, Noah was unaware he was standing on the world's first datum feature!

Our point is that builders have long understood the need for a consistent and uniform origin from which to base their measurements. For the ancients, it was a patch of leveled ground; for modern manufacturers, it's a flat surface or a straight and round diameter on a precision machine part. Although any type of part feature can be a datum feature, selecting one is a bit like hiring a sheriff who will provide a strong moral center and direction for the townsfolk. What qualifications should we look for?

5.9.2.1 Datum Feature Selection

The most important quality you want in a datum feature (or a sheriff) is leadership. A good datum feature is a surface that most strongly influences the orientation and/or location of the part in its assembly. We call that a “functional” datum feature. Rather than being a slender little wisp, a good datum feature, such

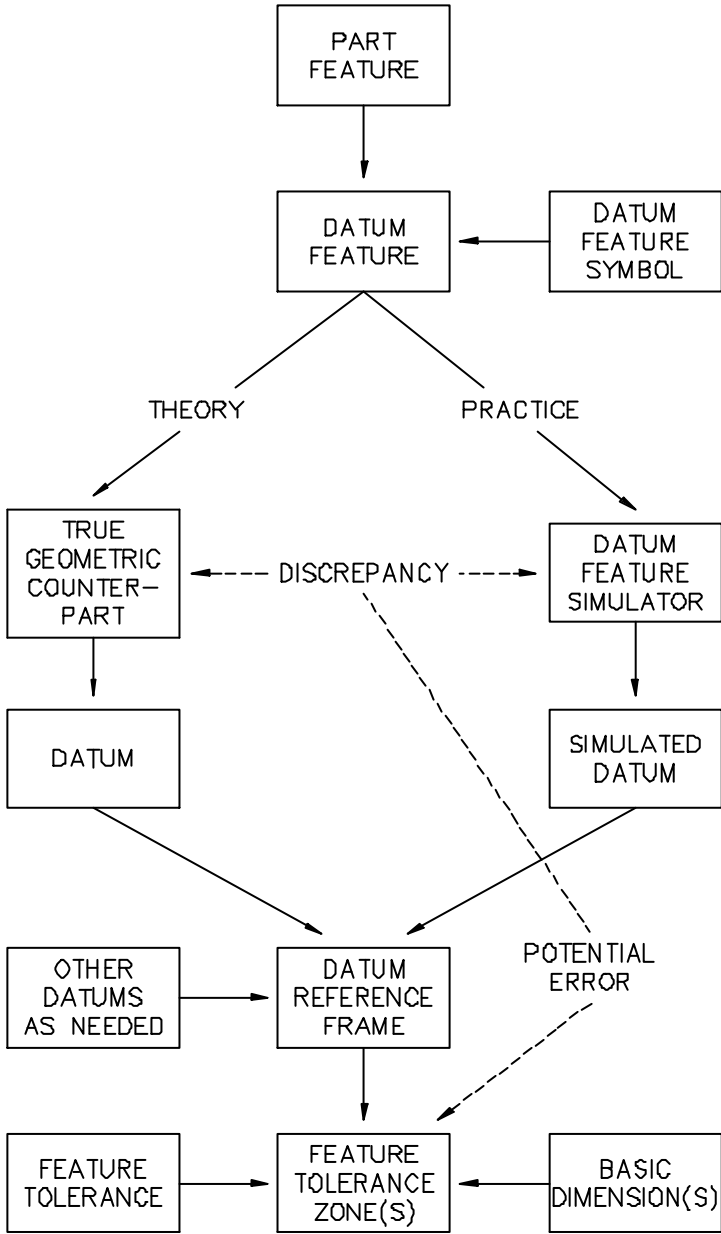


Figure 5-52 Establishing datum reference frames from part features

as that shown in Fig. 5-53, should have “broad shoulders” able to take on the weight of the part and provide stability. Look for a “straight arrow” with an even “temperament” and avoid “moody” and unfinished surfaces with high and low spots. Just as you want a highly visible sheriff, choose a datum feature that’s likewise always accessible for fixturing during manufacturing, or for inspection probing at various stages of completion.

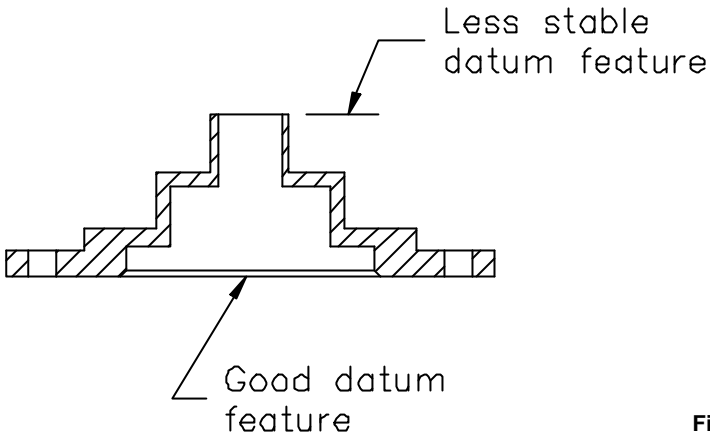


Figure 5-53 Selection of datum features

5.9.2.2 Functional Hierarchy

It's tough to judge leadership in a vacuum, but you can spot it intuitively when you see how a prospect relates to others. Fig. 5-54 shows three parts of a car engine: engine block, cylinder head, and rocker arm cover. Intuitively, we rank the dependencies of the pieces: The engine block is our foundation to which we bolt on the cylinder head, to which we in turn bolt on the rocker arm cover. And in fact, that's the

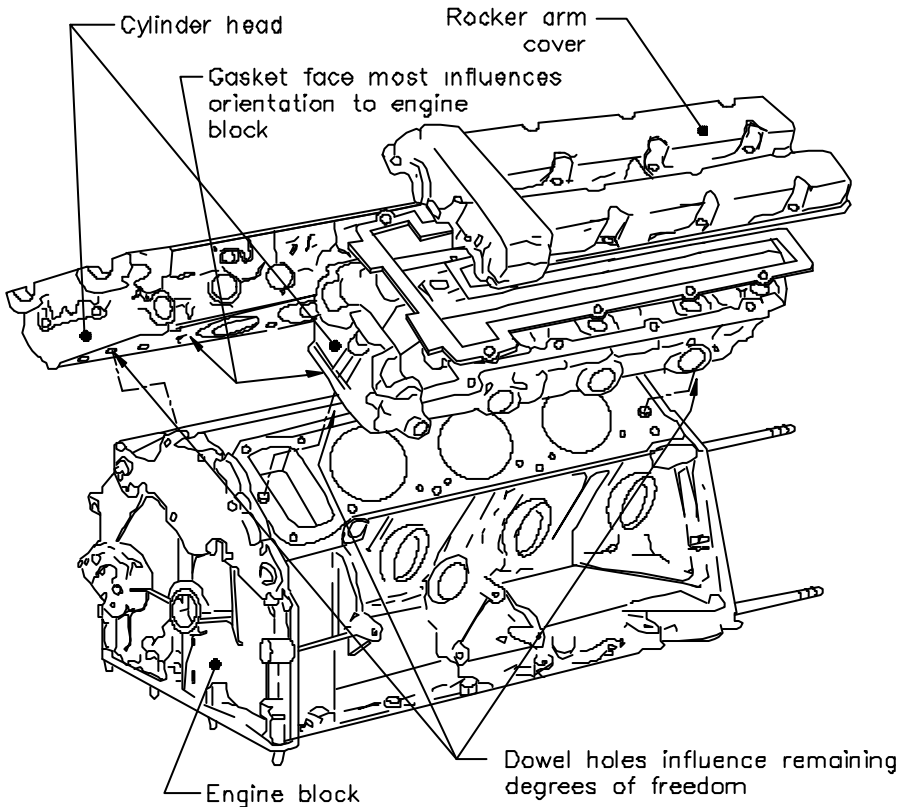


Figure 5-54 Establishing datums on an engine cylinder head

typical assembly sequence. Thus, in “interviewing” candidates for datum feature on the cylinder head, we want the feature that most influences the head’s orientation to the engine block. A clear choice would be the bottom (head gasket) face. The two dowel holes are the other key players, influencing the remaining degree of orientation as well as the location of the head on the block. These datum features, the bottom face and the dowel holes, satisfy all our requirements for good, functional datum features. To select the upper surface of the cylinder head (where the rocker cover mounts) as a datum feature for the head seems backwards—counterintuitive.

In our simple car engine example, functional hierarchy is based on assembly sequence. In other types of devices, the hierarchy may be influenced or dominated by conflicting needs such as optical alignment. Thus, datum feature selection can sometimes be as much art as science. In a complicated assembly, two experts might choose different datum features.

5.9.2.3 Surrogate and Temporary Datum Features

Often, a promising candidate for datum feature has all the leadership, breadth, and character we could ever hope for and would get sworn in on the spot if only it weren’t so reclusive or inaccessible. There are plenty of other factors that can render a functional datum feature useless to us. Perhaps it’s an O-ring groove diameter or a screw thread—those are really tough to work with. In such cases, it may be wiser to select a nonfunctional *surrogate datum feature*, as we’ve done in Fig. 5-55. A prudent designer might choose a broad flange face and a convenient outside diameter for surrogate datum features even though in assembly they contact nothing but air.

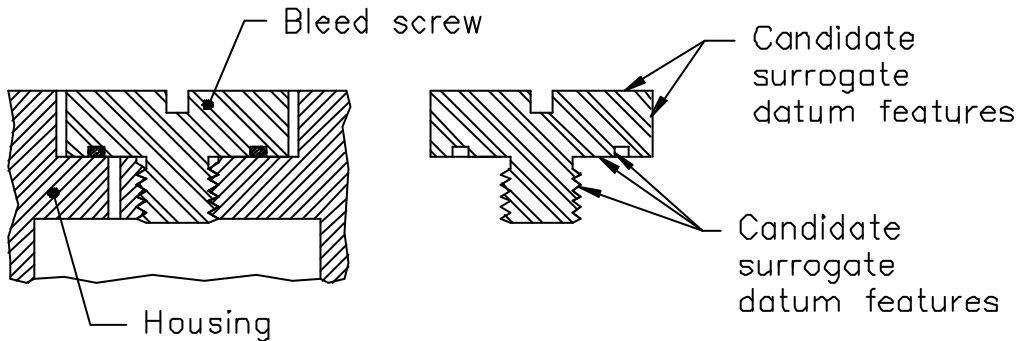


Figure 5-55 Selecting nonfunctional datum features

Many parts require multiple steps, or *operations*, in multiple machines for their manufacture. Such parts, especially castings and forgings, may need to be fixtured or inspected even before the functional datum features are finished. A thoughtful designer will anticipate these manufacturing needs and identify some *temporary datum features* either on an intermediate operation drawing or on the finished part drawing.

The use of surrogate and temporary datum features often requires extra precautions. These nonfunctional surfaces may have to be made straighter, rounder, and/or smoother than otherwise necessary. Also, the relationship between these features and the real, functional features may have to be closely controlled to prevent tolerances from stacking up excessively. There is a cost tradeoff in passing over functional datum features that may be more expensive to work with in favor of nonfunctional datum features that may be more expensive to manufacture.

5.9.2.4 Identifying Datum Features

Once a designer has “sworn in” a datum feature, he needs to put a “badge” on it to denote its authority. Instead of a star, we use the “datum feature” symbol shown in Fig. 5-56. The symbol consists of a capital letter enclosed in a square frame, a leader line extending from the frame to the datum feature, and a terminating triangle. The triangle may optionally be solid filled, making it easier to spot on a busy drawing.

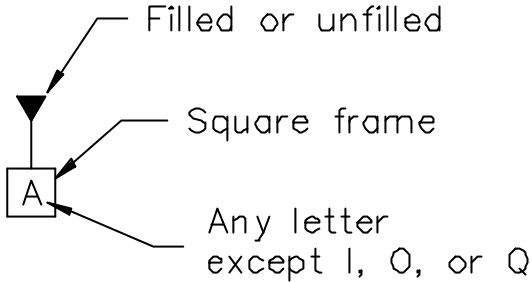


Figure 5-56 Datum feature symbol

Each datum feature shall be identified with a different letter of the alphabet (except I, O, or Q). When the alphabet is exhausted, double letters (AA through AZ, BA through BZ, etc.) are used and the frame is elongated to fit. Datum identifying letters have no meaning except to differentiate datum features. Though letters need not be assigned sequentially, or starting with A, there are advantages and disadvantages to doing both. In a complicated assembly, it may be desirable to coordinate letters among various drawings, so that the same feature isn't B on the detail part drawing, and C on the assembly drawing. It can be confusing when two different parts in an assembly both have a datum feature G and those features don't mate. On the other hand, someone reading one of the detail part drawings can be frustrated looking for nonexistent datums where letters are skipped. Such letter choices are usually left to company policy, and may be based on the typical complexity of the company's drawings.

The datum feature symbol is applied to the concerned feature surface outline, extension line, dimension line, or feature control frame as follows:

(a) placed on the outline of a feature surface, or on an extension line of the feature outline, clearly separated from the dimension line, when the datum is the surface itself. See Fig. 5-57(a).

(b) placed on an extension of the dimension line of a feature of size when the datum is the axis or center plane. If there is insufficient space for the two arrows, one of them may be replaced by the datum feature triangle. See Fig. 5-57(b).

(c) placed on the outline of a cylindrical feature surface or an extension line of the feature outline, separated from the size dimension, when the datum is the axis. The triangle may be drawn tangent to the feature. See Fig. 5-57(c).

(d) placed on a dimension leader line to the feature size dimension where no geometrical tolerance and feature control frame are used. See Fig. 5-57(d).

(e) placed on the planes established by datum targets on complex or irregular datum features (see section 5.9.13.6), or to reidentify previously established datum axes or planes on repeated or multisheet drawing requirements. Where the same datum feature symbol is repeated to identify the same feature in other locations of a drawing, it need not be identified as reference.

(f) placed above or below and attached to the feature control frame when the feature (or group of features) controlled is the datum axis or datum center plane. See Fig. 5-57(e).

(g) placed on a chain line that indicates a partial datum feature.

Formerly, the “datum feature” symbol consisted of a rectangular frame containing the datum-identifying letter preceded and followed by a dash. Because the symbol had no terminating triangle, it was placed differently in some cases.

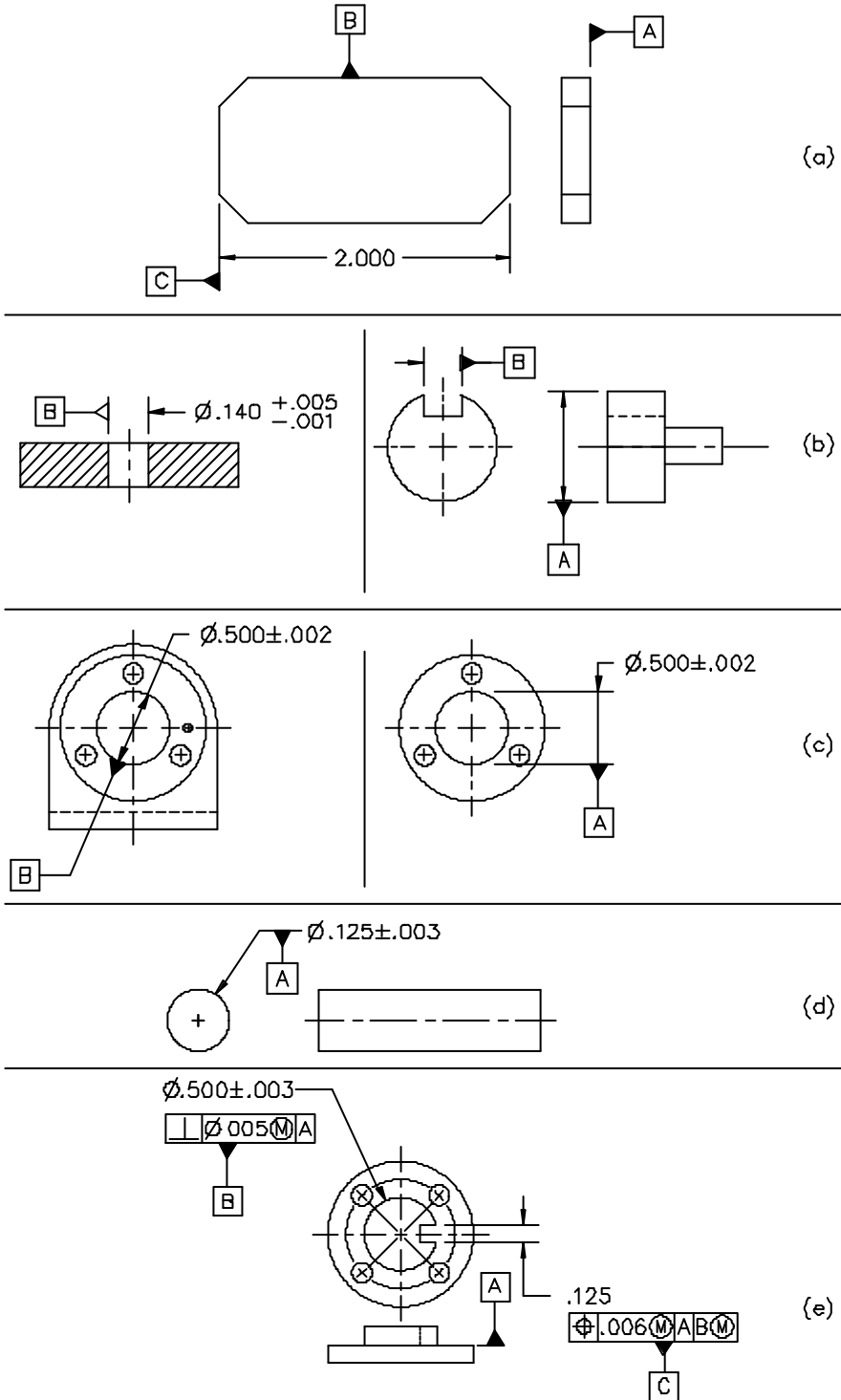


Figure 5-57 Methods of applying datum feature symbols

5.9.3 True Geometric Counterpart (TGC)—Introduction

Simply deputizing a part surface as a “datum feature” still doesn’t give us the uniform origin necessary for highly precise measurements. As straight, flat, and/or round as that feature may be, it still has slight irregularities in its shape that could cause differences in repeated attempts to reckon from it. To eliminate such measurement variation, we need to reckon from a geometric shape that’s, well, perfect. Such a perfect shape is called a *true geometric counterpart (TGC)*.

If we look very closely at how parts fit together in Fig. 5-58, we see they contact each other only at a few microscopic points. Due to infinitesimal variations and irregularities in the manufacturing process, these few peaks or *high points* stand out from the surrounding part surface. Now, we realize that when parts are clamped together with bolts and other fastening forces, sometimes at thousands of pounds per square inch, surface points that were once the elite “high” get brutally mashed down with the rank and file. Flanges warp and bores distort. Flat head screws stretch and bend tortuously as their cones squash into countersinks. We hope these plastic deformations and realignments are negligible in proportion to assembly tolerances. In any event, we lack the technology to account for them. Thus, GD&T’s datum principles are based on the following assumptions: 1) The foremost design criterion is matability; and 2) high points adequately represent a part feature’s matability. Thus, like it or not, all datum methods are based on surface high points.

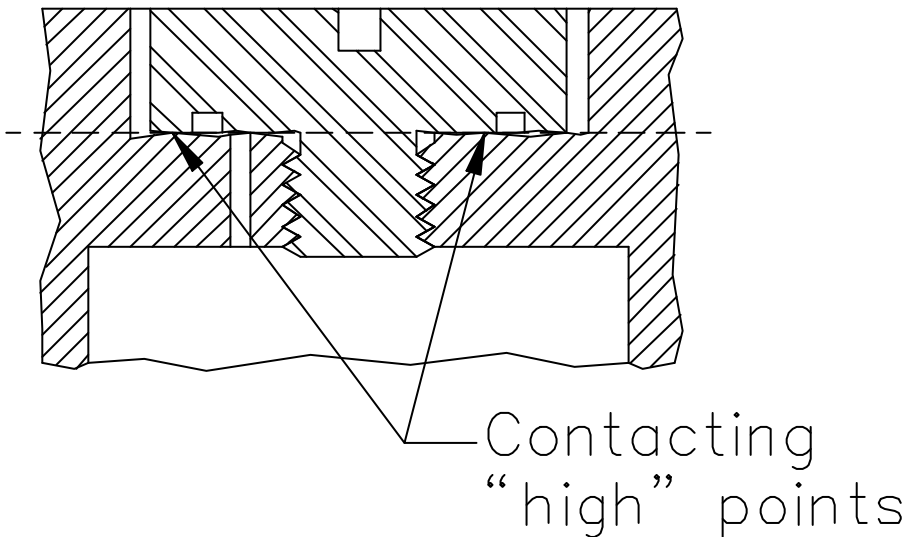


Figure 5-58 Parts contacting at high points

From Table 5-4, you’ll notice for every datum feature, there’s at least one TGC (perfect shape) that’s related to its surface high points. In many cases, the TGC and the datum feature surface are conceptually brought together in space to where they contact each other at one, two, or three high points on the datum feature surface. In some cases, the TGC is custom fitted to the datum feature’s high points. In yet other cases, the TGC and datum feature surface are meant to clear each other. We’ll explain the table and the three types of relationships in the following sections.

Table 5-4 Datum feature types and their TGCs

Datum Feature Type	Datum Precedence	True Geometric Counterpart (TGC)	Restraint of TGC*	Contact Points	Typical Datum Simulator(s)
nominally flat plane	primary	tangent plane	none	1-3	surface plate or other flat base
	secondary or tertiary	tangent plane	O	1-2	restrained square or fence
math-defined (contoured) plane	primary	tangent math-defined contour	none	1-6	contoured fixture
	secondary or tertiary	tangent math-defined contour	O	1-2	restrained contoured fixture
feature of size, RFS	primary	actual mating envelope	none	3-4	adjustable-size chuck, collet, or mandrel; fitted gage pin, ring, or Jo blocks
	secondary or tertiary	actual mating envelope	O	2-3	same as for primary (above), but restrained
feature of size, MMC	primary	boundary of perfect form at MMC	none	0-4	gage pin, ring, or Jo blocks, at MMC size
	primary w/straightness or flatness tol at MMC	MMC virtual condition boundary	none	0-4	gage pin, ring, or Jo blocks, at MMC virtual condition size
	secondary or tertiary	MMC virtual condition boundary	O,L	0-2	restrained pin, hole, block, or slot, at MMC virtual condition size
feature of size, LMC	primary	boundary of perfect form at LMC	none	0-4	computer model at LMC size
	primary w/straightness or flatness tol at LMC	LMC virtual condition boundary	none	0-4	computer model at LMC virtual condition size
	secondary or tertiary	LMC virtual condition boundary	O,L	0-2	computer model at LMC virtual condition size
bounded feature, MMC	primary	MMC profile boundary	none	0-5	fixture or computer model
	secondary or tertiary	MMC virtual condition boundary	O,L	0-3	fixture or computer model
bounded feature, LMC	primary	LMC profile boundary	none	0-5	computer model
	secondary or tertiary	LMC virtual condition boundary	O,L	0-3	computer model

* to higher-precedence datum(s) O = restrained in orientation, L = restrained in location

5.9.4 Datum

Remember the definition: A *datum* is a **theoretically exact point, axis, or plane derived from the true geometric counterpart of a specified datum feature**. Once we have a TGC for a feature, it's simple to derive the datum from it based on the TGC's shape. This is shown in Table 5-5.

Table 5-5 TGC shape and the derived datum

TGC SHAPE	DERIVED DATUM
tangent plane	identical plane
math-defined contour	3 mutually perpendicular planes (complete DRF)
sphere	(center) point
cylinder	axis (straight line)
opposed parallel planes	(center) plane
revolute	axis and point along axis
bounded feature	2 perpendicular planes

5.9.5 Datum Reference Frame (DRF) and Three Mutually Perpendicular Planes

Datums can be thought of as building blocks used to build a dimensioning grid called a *datum reference frame (DRF)*. The simplest DRFs can be built from a single datum. For example, Fig. 5-59(a) shows how a datum plane provides a single dimensioning axis with a unique orientation (perpendicular to the plane) and an origin. This DRF, though limited, is often sufficient for controlling the orientation and/or location of other features. Fig. 5-59(b) shows how a datum axis provides one dimensioning axis having an orientation with no origin, and two other dimensioning axes having an origin with incomplete orientation. This DRF is adequate for controlling the coaxiality of other features.

Simple datums may be combined to build a 2-D Cartesian coordinate system consisting of two perpendicular axes. This type of DRF may be needed for controlling the location of a hole. Fig. 5-60 shows the ultimate: a 3-D Cartesian coordinate system having a dimensioning axis for height, width, and depth. This top-of-the-line DRF has three mutually perpendicular planes and three mutually perpendicular axes. Each of the three planes is perpendicular to each of the other two. The line of intersection of each pair of planes is a dimensioning axis having its origin at the point where all three axes intersect. Using this DRF, the orientation and location of any type of feature can be controlled to any attitude, anywhere in space. Usually, it takes two or three datums to build this complete DRF.

Since each type of datum has different abilities, it's not very obvious which ones can be combined, nor is it obvious how to build the DRF needed for a particular application. In the following sections, we'll help you select datums for each type of tolerance. In the meantime, we'll give you an idea of what each datum can do.

5.9.6 Datum Precedence

Where datums are combined to build a DRF, they shall always be basically (perfectly) oriented to each other. In some cases, two datums shall also be basically located, one to the other. Without that perfect alignment, the datums won't define a unique and unambiguous set of mutually perpendicular planes or axes.

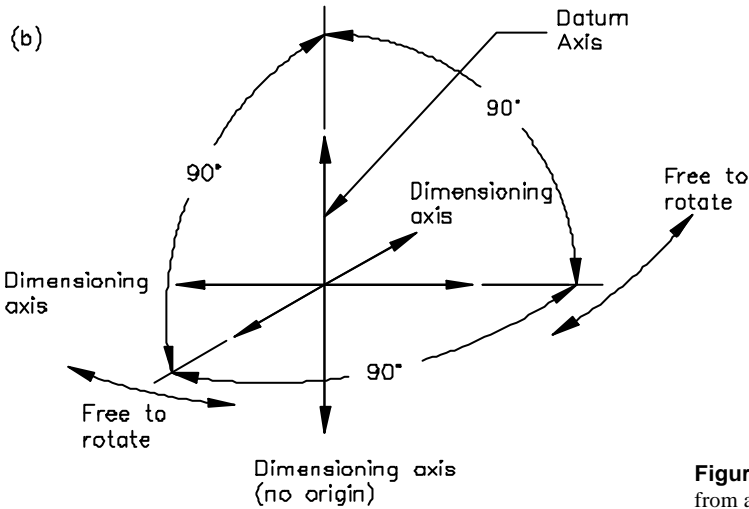
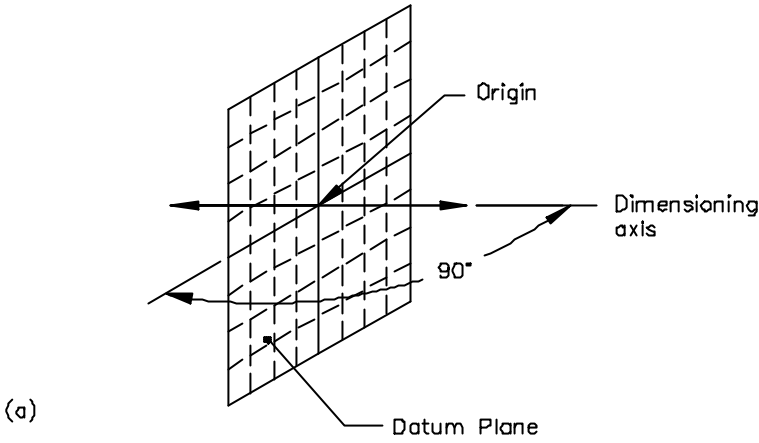


Figure 5-59 Building a simple DRF from a single datum

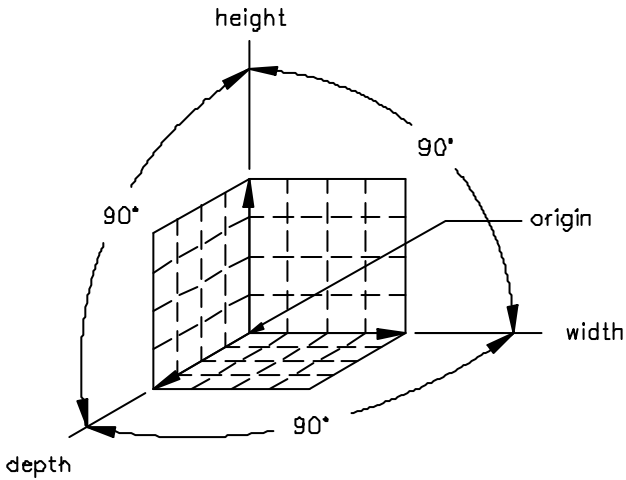


Figure 5-60 3-D Cartesian coordinate system

In functional hierarchy, Fig. 5-61's "cover" is a part that will be mounted onto a "base." The cover's broad face will be placed against the base, slid up against the fences on the base, then spot welded in place. Using our selection criteria for functional datum features, we've identified the cover's three planar mounting features as datum features A, B, and C. Considered individually, the TGC for each datum feature is a full-contact tangent plane. Since the datum feature surfaces are slightly out-of-square to each other, their full-contact TGCs would likewise be out-of-square to each other, as would be the three datum planes derived from them. Together, three out-of-square datum planes cannot yield a unique DRF. We need the three datum (and TGC) planes to be mutually perpendicular. The only way to achieve that is to excuse at least two of the TGC planes from having to make full contact with the cover's datum features.

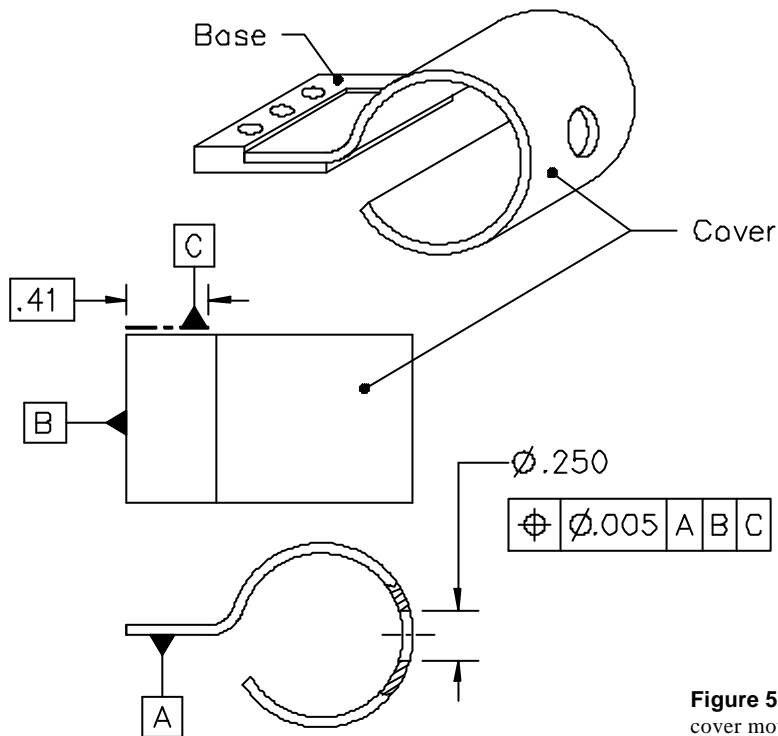


Figure 5-61 Datum precedence for a cover mounted onto a base

On the other hand, if we allow each of the three TGCs to contact only a single high point on its respective datum feature, we permit a wide variety of alignment relationships between the cover and its TGCs. Intuitively, we wouldn't expect the cover to assemble by making only one-point contact with the base. And certainly, this scheme is no good if we want repeatability in establishing DRFs. Instead, we should try to maximize contact between our datum feature surfaces and their TGC planes. Realizing we can't have full contact on all three surfaces, we'll have to prioritize the three datum features, assigning each a different requirement for completeness of contact.

Using the same criteria by which we selected datum features A, B, and C in the first place, we examine the leadership each has over the cover's orientation and location in the assembly. We conclude that datum feature A, being the broad face that will be clamped against the base, is the most influential. The datum feature B and C edges will be pushed up against fences on the base. Datum feature B, being longer, will tend to overpower datum feature C in establishing the cover's rotation in assembly. However, datum feature C will establish a unique location for the cover, stopping against its corresponding fence on the base.

Thus, we establish *datum precedence* for the cover, identifying datum A as the *primary datum*, datum B as the *secondary datum*, and datum C as the *tertiary datum*. We denote datum precedence by placing the datum references sequentially in individual compartments of the feature control frame. The tolerance compartment is followed by the primary datum compartment, followed by the secondary datum compartment, followed by the tertiary datum compartment. In text, we can express the same precedence A|B|C. The specified datum precedence tells us how to prioritize establishment of TGCs, allowing us to fit three mutually perpendicular TGC planes to our out-of-square cover. Here's how it works.

5.9.7 Degrees of Freedom

Let's start with a system of three mutually perpendicular TGC planes as shown in Fig. 5-62(a). For discussion purposes, let's label one plane "A," one "B," and one "C." The lines of intersection between each pair of planes can be thought of as axes, "AB," "BC," and "CA." Remember, this is a system of TGC planes, not a DRF (yet).

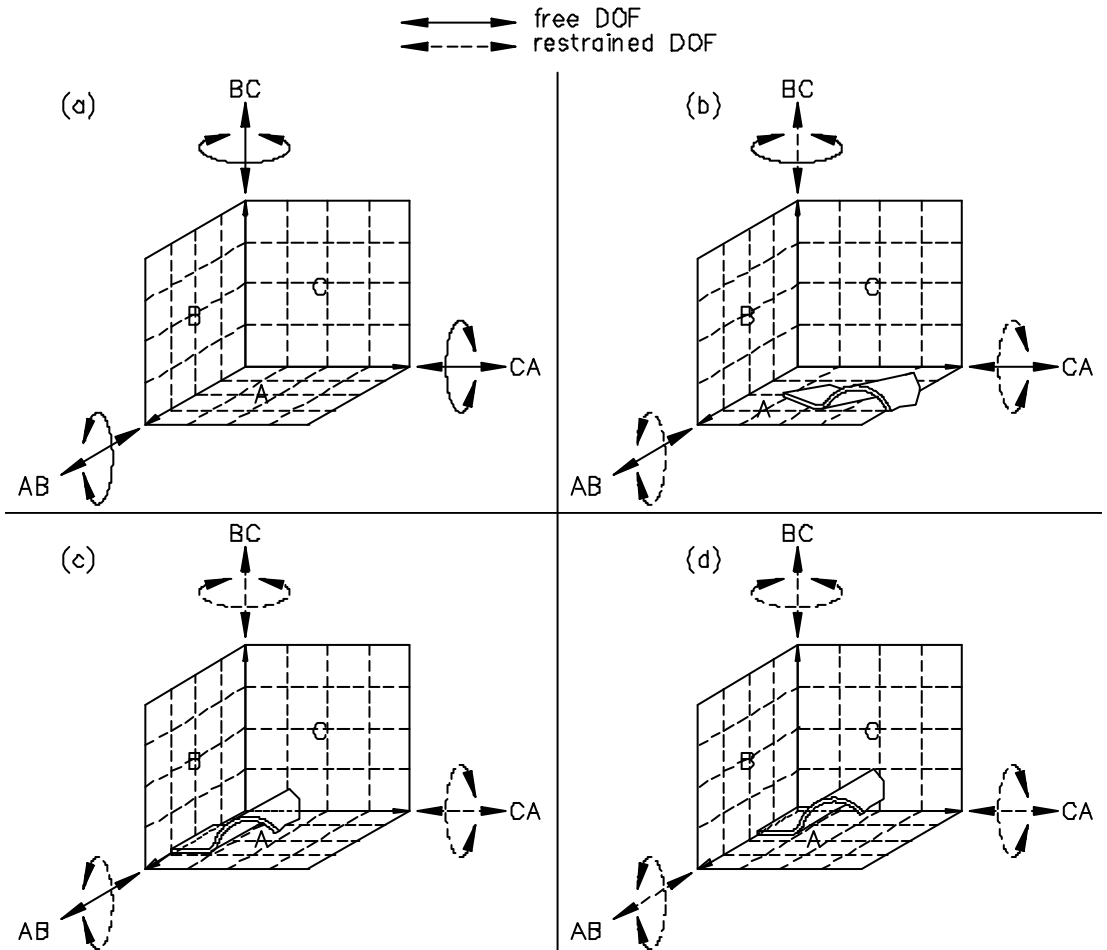


Figure 5-62 Arresting six degrees of freedom between the cover and the TGC system

Imagine the cover floating in space, tumbling all about, and drifting in a randomly winding motion relative to our TGC system. (The CMM users among you can imagine the cover fixed in space, and the TGC system floating freely about—Albert Einstein taught us it makes no difference.) We can describe all the relative free-floating motion between the cover and the TGC system as a combination of *rotation* and *translation* (linear movement) parallel to each of the three TGC axes, AB, BC, and CA. These total *six degrees of freedom*. In each portion of Fig. 5-62, we represent each degree of freedom with a double-headed arrow. To achieve our goal of fixing the TGC system and cover together, we must arrest each one of the six degrees of relative motion between them. Watch the arrows; as we restrain each degree of freedom, its corresponding arrow will become dashed.

Each datum reference in the feature control frame demands a level of congruence (in this case, contact) between the datum feature and its TGC plane. The broad face of the cover is labeled datum feature A, the primary datum feature. That demands maximum congruence between datum feature A and TGC plane A. Fig. 5-62(b) shows the cover slamming up tight against TGC plane A and held there, as if magnetically. Suddenly, the cover can no longer rotate about the AB axis, nor can it rotate about the CA axis. It can no longer translate along the BC axis. Three degrees of freedom arrested, just like that. (Notice the arrows.) However, the cover is still able to twist parallel to the BC axis and translate at will along the AB and CA axes. We'll have to put a stop to that.

The long edge of the cover is labeled datum feature B, the secondary datum feature. Fig. 5-62(c) illustrates the cover sliding along plane A, slamming up tight against plane B and held there. However, this time the maximum congruence possible is limited. As the cover slides, all three degrees of freedom arrested by any higher precedence datum feature—datum feature A in this case—shall remain arrested. Thus, datum feature B can only arrest degrees of freedom left over from datum feature A. This means the cover can't rotate about the BC axis anymore, nor can it translate along the CA axis. Two more degrees of freedom are now arrested. We've reduced the cover to sliding to and fro in a perfectly straight line parallel to axis AB. One more datum reference should finish it off.

The short edge of the cover is labeled datum feature C, the tertiary datum feature. Fig. 5-62(d) now shows the cover sliding along axis AB, slamming up tight against plane C and held there. Again, the maximum congruence possible is even more limited. As the cover slides, all degrees of freedom arrested by higher precedence datum features—three by datum feature A and two by datum feature B—shall remain arrested. Thus, datum feature C can only arrest the last remaining degree of freedom, translation along axis AB. Finally, all six degrees of freedom have been arrested; the cover and its three TGC planes are now totally stuck together.

The next steps are to derive the datum from each TGC, then construct the DRF from the three datums. Since we used such a simple example, in this case, the datums are the same planes as the TGCs, and the three mutually perpendicular planes of the DRF are the very same datum planes. Sometimes, it's just that simple!

Because we were so careful in selecting and prioritizing the cover's datum features according to their assembly functions, the planes of the resulting DRF correspond as closely as possible to the mating surfaces of the base. That's important because it allows us to maximize tolerances for other features controlled to our DRF. Just as importantly, we can unstick the cover, set it toppling and careening all over again, then repeat the above three alignment steps. No matter who tells it, no matter who performs it, no matter which moves, TGCs or cover, the cover's three datum features and their TGC planes will always slam together exactly the same. We'll always get the same useful DRF time after time.

"Always," that is, when datum precedence remains the same, A|B|C. Note that in Fig. 5-63(a), the DRF's orientation was optimized for the primary datum feature, A, first and foremost. The orientation was only partly optimized for the secondary datum feature, B. Orientation was not optimized at all for the tertiary datum feature, C. If we transpose datum precedence to A|C|B, as in Fig. 5-63(b), our first alignment step remains the same. We still optimize orientation of the TGC system to datum feature A. However, now

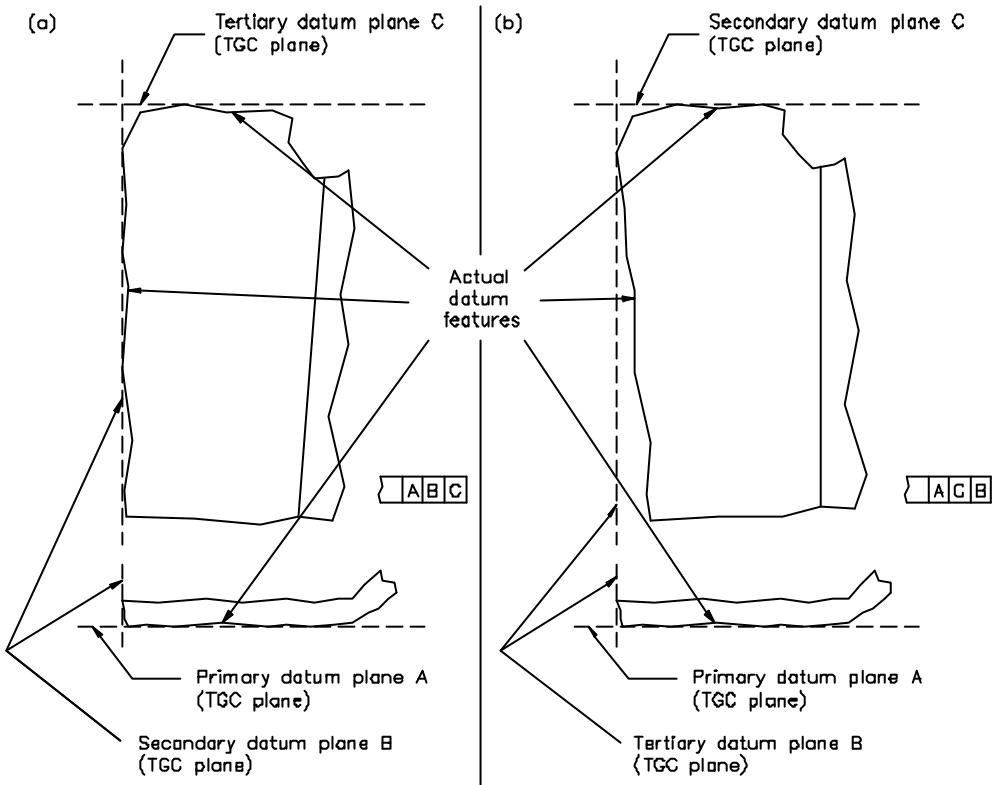


Figure 5-63 Comparison of datum precedence

our second step is to optimize orientation partly for secondary datum C. Datum feature B now has no influence over orientation. Thus, changing datum precedence yields a different DRF. The greater the out-of-squareness between the datum features, the greater the difference between the DRFs.

Our example part needs three datums to arrest all six degrees of freedom. On other parts, all six degrees can be arrested by various pairings of datums, including two nonparallel lines, or by certain types of math-defined contours. Further, it's not always necessary to arrest all six degrees of freedom. Many types of feature control, such as coaxiality, require no more than three or four degrees arrested.

FAQ: *Is there any harm in adding more datum references than necessary in a feature control frame—just to be on the safe side?*

A: Superfluous datum references should be avoided to prevent confusion. A designer must fully understand every datum reference, including the appropriate TGC, the type of datum derived, the degrees of freedom arrested based on its precedence, and that datum's role in constructing the DRF. Doubt is unacceptable.

5.9.8 TGC Types

Table 5-4 shows that each type of datum feature has a corresponding TGC. Each TGC either has no size, adjustable size, or fixed size, depending on the type of datum feature and the referenced material condition. Also, a TGC is either restrained or unrestrained, depending on the datum precedence.

5.9.8.1 Restrained Versus Unrestrained TGC

We saw in our cover example how all the degrees of freedom arrested by higher precedence datum features flowed down to impose limitations, or *restraint*, on the level of congruence achievable between each lower-precedence datum feature and its TGC. As we mentioned, such restraint is necessary in all DRFs to establish mutually perpendicular DRF planes. In the case of a primary datum feature, there is no higher precedence datum, and therefore, no restraint. However, where a secondary TGC exists, it's restrained relative to the primary TGC in all three or four degrees arrested by the primary datum feature. Likewise, where a tertiary TGC exists, it's restrained relative to the primary and secondary TGCs in all five degrees arrested by the primary and secondary datum features.

In our simple cover example, secondary TGC plane B is restrained perpendicular to TGC plane A. The translation arrested by plane A has no effect on the location of plane B. Tertiary TGC plane C is first restrained perpendicular to TGC plane A, then perpendicular to TGC plane B as well. The two degrees of translation arrested by planes A and B have no effect on the location of plane C.

In all cases, the orientation of secondary and tertiary TGCs is restrained. Where a secondary or tertiary datum feature is nominally angled (neither parallel nor perpendicular) to a higher precedence datum, its TGC shall be restrained at the basic angle expressed on the drawing. The planes of the DRF remain normal to the higher precedence datums. If the angled datum arrests a degree of translation, the origin is where the angled datum (not the feature itself) intersects the higher precedence datum. As we'll explain in section 5.9.8.4, there are cases where the location of a TGC is also restrained relative to higher-precedence datums.

5.9.8.2 Nonsize TGC

Look at the "Datum Feature Type" column of Table 5-4. Notice that for a nominally flat plane, the TGC is a tangent plane. For a math-defined (contoured) plane, the TGC is a perfect, tangent, math-defined contour. These TGC planes, whether flat or contoured, have no intrinsic size. As we saw in Fig. 5-62(b), the TGC plane and the datum feature surface are brought together in space to where they just contact at as many high points on the datum feature surface as possible (as many as three for a flat plane, or up to six for a contoured plane). "Tangent" means the TGC shall contact, but not encroach beyond the datum feature surface. In other words, all noncontacting points of the datum feature surface shall lie on the same side of the TGC plane.

Notice under the "Restraint of TGC" column, for a primary flat or contoured tangent plane TGC, no restraint is possible. For a secondary or tertiary tangent plane TGC, orientation is always restrained and location is never restrained to the higher-precedence datum(s). If location were restrained, it might be impossible to achieve contact between the datum feature surface and its TGC.

5.9.8.3 Adjustable-size TGC

Looking again at Table 5-4, we notice that for a feature of size referenced as a datum RFS, the TGC is an actual mating envelope as defined in section 5.6.4.2. An actual mating envelope is either a perfect sphere, cylinder, or pair of parallel planes, depending on the type of datum feature of size. See Fig. 5-64. The actual mating envelope's size shall be adjusted to make contact at two to four high points on the datum feature surface(s) without encroaching beyond it.

According to the Math Standard, for a secondary or tertiary actual mating envelope TGC, orientation is always restrained and location is never restrained to the higher-precedence datum(s). See Fig. 5-65.

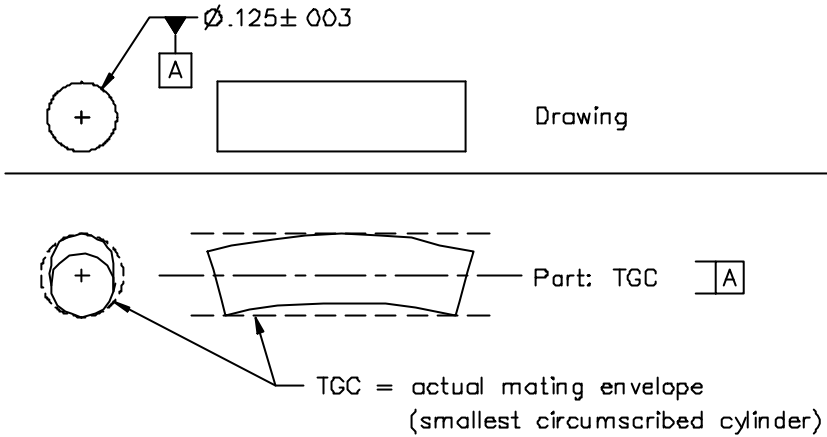


Figure 5-64 Feature of size referenced as a primary datum RFS

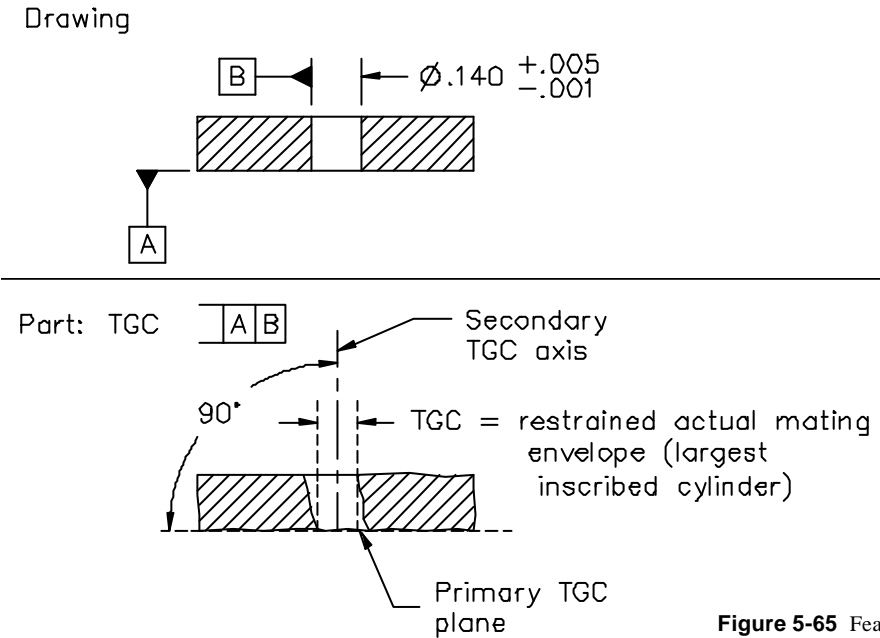


Figure 5-65 Feature of size referenced as a secondary datum RFS

FAQ: *But, if I have a shaft (primary datum A) with a shallow radial anti-rotation hole (secondary datum B), how can the hole arrest the DRF's rotation if its TGC isn't fixed (located) on center with the shaft?*

A: In this example, datum feature B, by itself, can't arrest the rotational degree of freedom satisfactorily. It must work jointly with datum feature A. Both A and B should be referenced as secondary co-datum features, as described in section 5.9.14.2. The DRF would be A|A-B.

5.9.8.4 Fixed-size TGC

According to Table 5-4, for features of size and bounded features referenced as datums at MMC or LMC, the TGCs include MMC and LMC boundaries of perfect form, MMC and LMC virtual condition boundaries, and MMC and LMC profile boundaries. See Figs. 5-66 through 5-71. Each of these TGCs has a fixed size and/or fixed shape. For an MMC or LMC boundary of perfect form, the size and shape are defined by size limits (see section 5.6.3.1 and Figs. 5-66 and 5-68). A virtual condition boundary is defined by a

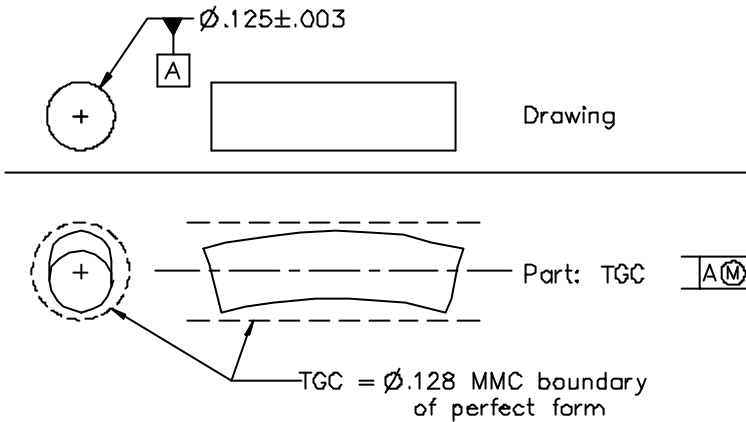


Figure 5-66 Feature of size referenced as a primary datum at MMC

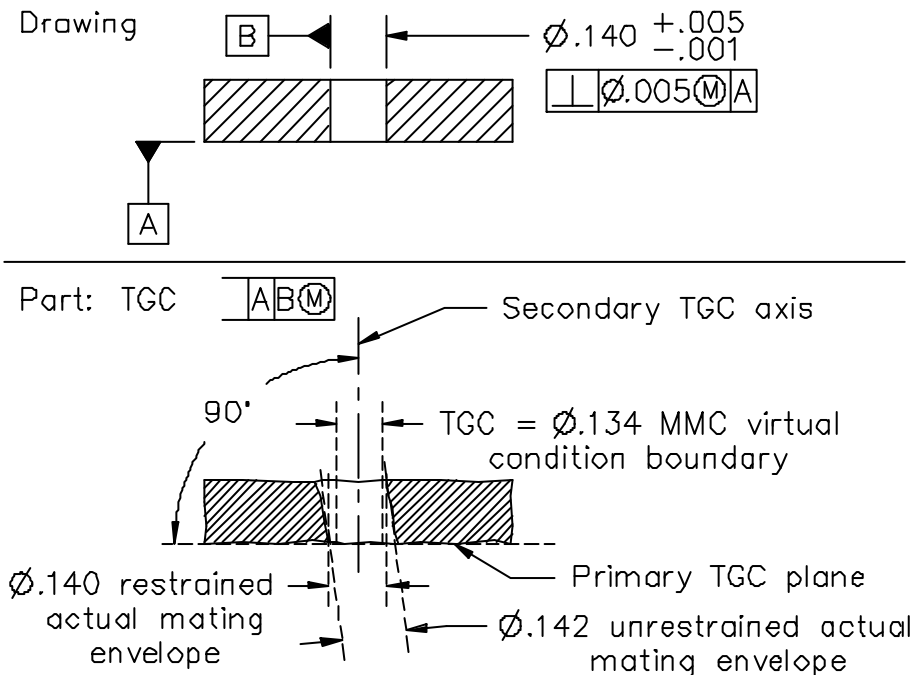


Figure 5-67 Feature of size referenced as a secondary datum at MMC

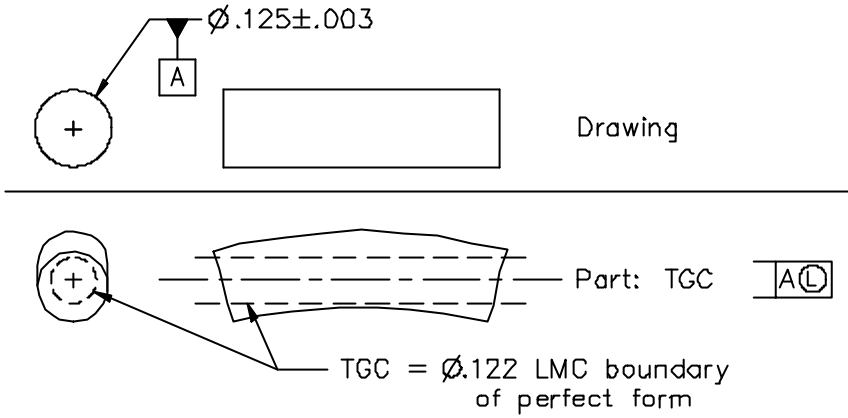


Figure 5-68 Feature of size referenced as a primary datum at LMC

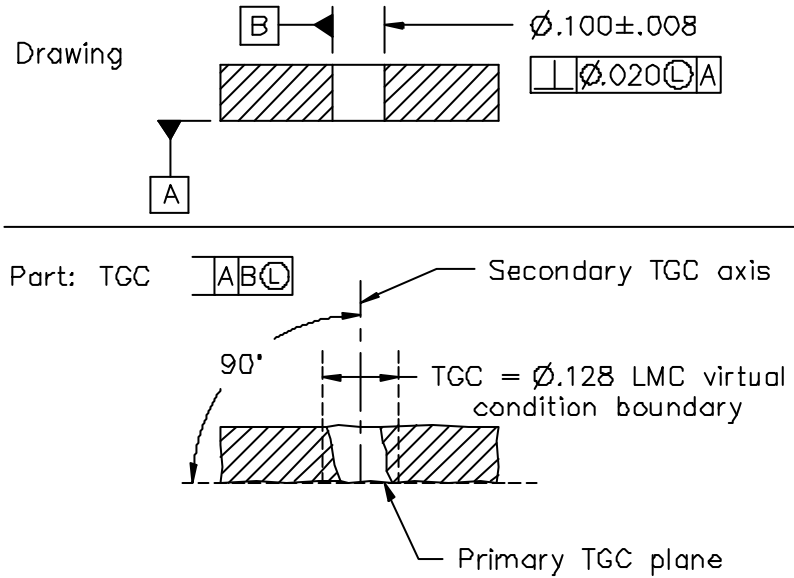


Figure 5-69 Feature of size referenced as a secondary datum at LMC

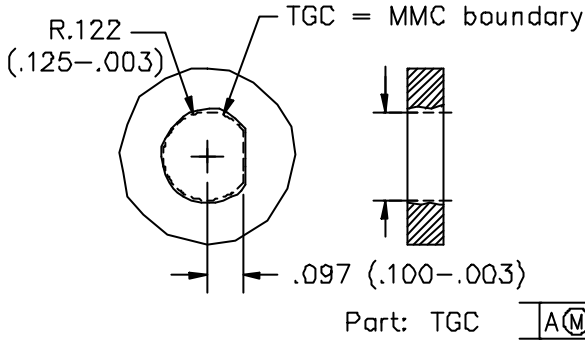
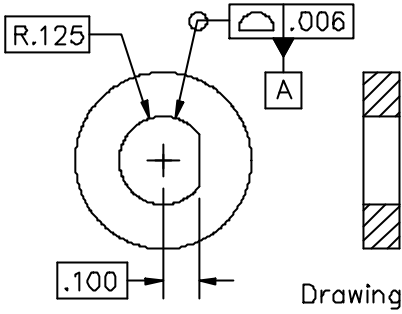


Figure 5-70 Bounded feature referenced as a primary datum at MMC

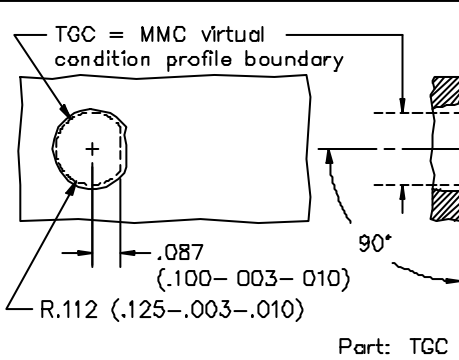
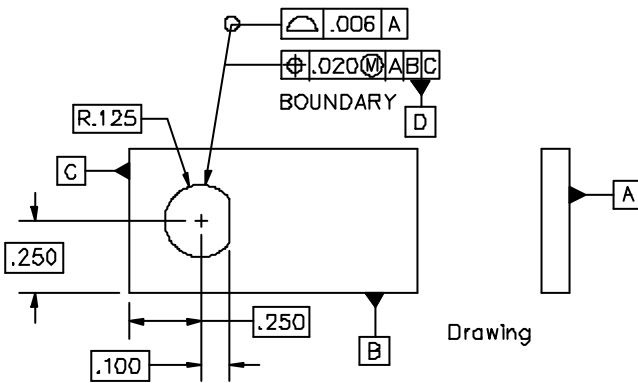


Figure 5-71 Bounded feature referenced as a secondary datum at MMC

combination of size limits and a geometric tolerance (see section 5.6.3.2 and Figs. 5-67 and 5-69). A profile boundary is defined by a profile tolerance (see section 5.13.4 and Figs. 5-70 and 5-71). Thus, none of these boundaries are generated by referencing the feature as a datum feature. It's just that when the feature is referenced, its appropriate preexisting boundary becomes its TGC.

A straightness tolerance at MMC or LMC applied to a primary datum feature cylinder, or a straightness or flatness tolerance at MMC or LMC applied to a primary datum feature width establishes a Level 2 virtual condition boundary for that primary datum feature. See Fig. 5-72. This unrestrained virtual condition boundary becomes the TGC for the datum feature.

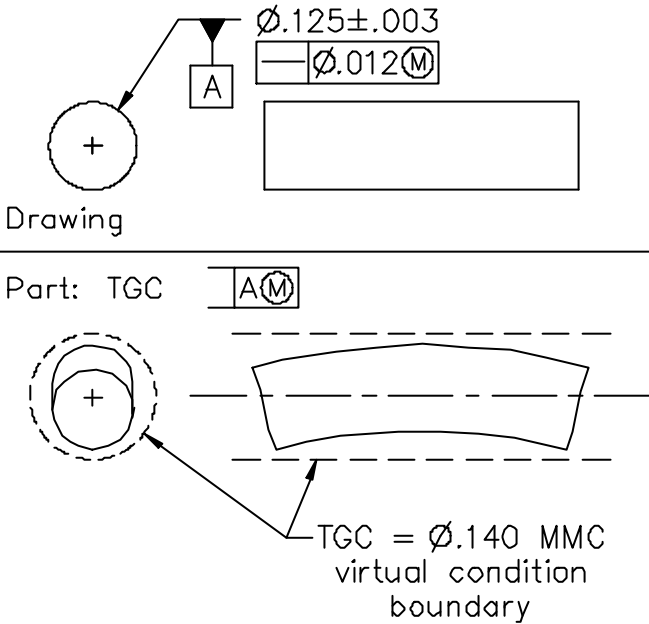


Figure 5-72 Cylindrical feature of size, with straightness tolerance at MMC, referenced as a primary datum at MMC

For a secondary or tertiary datum feature of size or bounded feature referenced at MMC or LMC, the TGC is an MMC or LMC virtual condition boundary. For this virtual condition boundary TGC, orientation is always restrained at the basic angle to the higher-precedence datum(s). Where the virtual condition boundary is also basically located relative to higher precedence datum(s), the TGC's location is always restrained at the basic location as well. In Fig. 5-24, the datum B bore is controlled with a perpendicularity tolerance at MMC, then referenced as a datum at LMC. Such applications should be avoided because the standards don't clearly define the TGC for datum B.

A fixed-size TGC is meant to emulate an assembly interface with a fixed-size feature on the mating part. Since contact may or may not occur between the two mating features, contact is likewise permitted but not required between the datum feature surface and its fixed-size TGC.

5.9.9 Datum Reference Frame (DRF) Displacement

The requirement for maximum contact between a planar surface and its nonsize TGC should yield a unique fit. Likewise, an actual mating envelope's maximum expansion within an internal feature of size or its contraction about an external feature of size ought to assure a repeatable fit. Each of those types of TGC should always achieve a unique and repeatable orientation and location relative to its datum feature. Conversely, a fixed-size TGC is not fitted to the datum feature, and need not even contact the datum

feature surface(s). Rather than achieving a unique and repeatable fit, the fixed-size TGC can achieve a variety of orientations and/or locations relative to its datum feature, as shown in Fig. 5-73. This effect, called *datum reference frame (DRF) displacement*, is considered a virtue, not a bug, since it emulates the variety of assembly relationships achievable between potential mating parts.

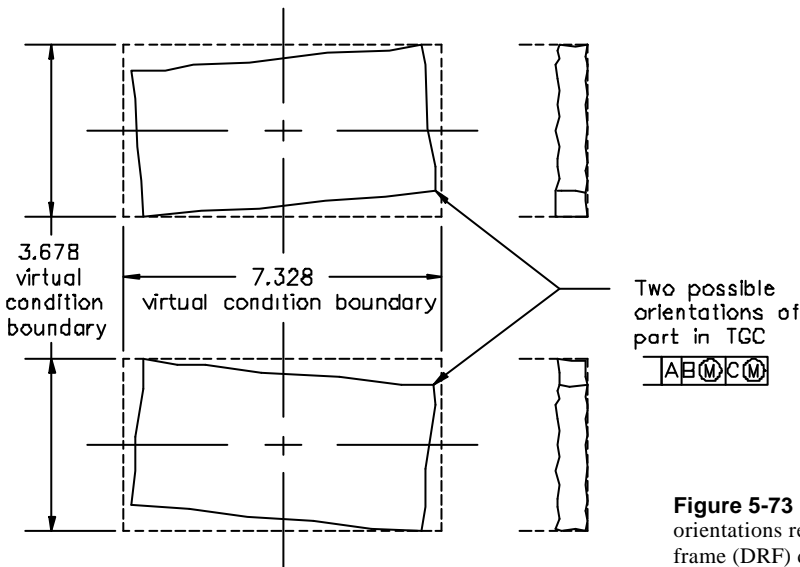
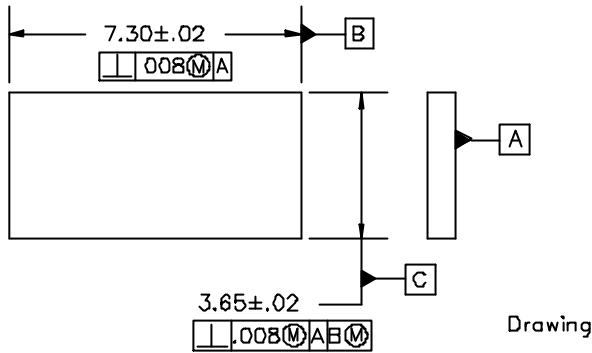


Figure 5-73 Two possible locations and orientations resulting from datum reference frame (DRF) displacement

Usually, a looser fit between two mating parts eases assembly. You may have experienced situations where screws can't seem to find their holes until you jiggle the parts around a little, then the screws drop right through. Where a designer can maximize the assembly clearances between piloting features, those clearances can be exploited to allow greater tolerances for such secondary features as screw holes. This may reduce manufacturing costs without harming assemblability.

5.9.9.1 Relative to a Boundary of Perfect Form TGC

In Fig. 5-74, we have three parts, shaft, collar, and pin. Let's assume our only design concern is that the pin can fit through both the collar and the shaft. We've identified as datum features the shaft's diameter and the collar's inside diameter. Notice that the smaller the shaft is made, the farther its cross-hole can stray from center and the pin will still assemble. Likewise, the larger the collar's inside diameter, the farther off-center its cross-hole can be and the pin will still assemble. On the shaft or the collar, we can make the hole's

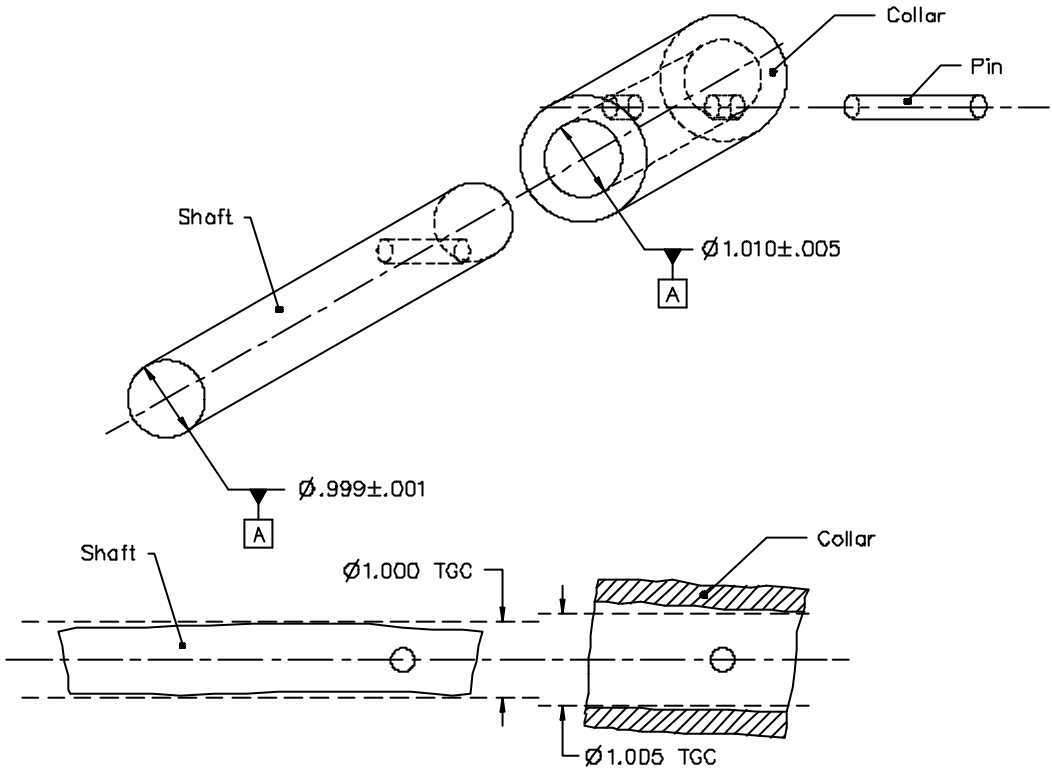


Figure 5-74 DRF displacement relative to a boundary of perfect form TGC

positional tolerance interact with the actual size of the respective datum feature, always permitting the maximum positional tolerance. We'll explain the tolerance calculations in Chapter 22, but right now, we're concerned with how to establish the DRFs for the shaft and the collar.

The shaft's datum feature is a feature of size. According to Table 5-4, if we reference that feature as a primary datum at MMC, its boundary of perfect form at MMC also becomes its TGC. That's a perfect $\varnothing 1.000$ cylinder. Any shaft satisfying its size limits will be smaller than $\varnothing 1.000$ (MMC) and able to rattle around, to some extent, within the $\varnothing 1.000$ TGC cylinder. (Remember, the datum feature surface need not contact the TGC anywhere.) This rattle, or DRF displacement, is relative motion permitted between the datum feature surface and its TGC. You can think of either one (or neither one) as being fixed in space. In the case of the shaft's primary datum, DRF displacement may include any combination of shifting and tilting. In fact, of the six degrees of freedom, none are absolutely restrained. Instead, rotation about two axes, and translation along two axes are merely limited. The limitations are that the TGC may not encroach beyond the datum feature surface. Obviously, the greater the clearance between the datum feature surface and its TGC, the greater the magnitude of allowable DRF displacement.

Similarly, the collar's datum feature is a feature of size. Referenced as a primary datum feature at MMC, its TGC is its $\varnothing 1.005$ boundary of perfect form at MMC. Any collar satisfying its size limits will be larger than $\varnothing 1.005$ (MMC) and able to rattle around about the $\varnothing 1.005$ TGC cylinder.

By extension of principle, an entire bounded feature may be referenced as a datum feature at MMC or LMC. Where the bounded feature is established by a profile tolerance, as in Fig. 5-70, the appropriate MMC or LMC profile boundary also becomes the TGC. As with simpler shapes, DRF displacement derives from clearances between the datum bounded feature surface and the TGC. As always, the TGC may not encroach beyond the datum feature surface.

5.9.9.2 Relative to a Virtual Condition Boundary TGC

A primary datum diameter or width may have a straightness tolerance at MMC, or a feature of size may be referenced as a secondary or tertiary datum at MMC. In these cases, DRF displacement occurs between the datum feature surface and the TGC that is the MMC virtual condition boundary. Table 5-4 reminds us that for a secondary or tertiary datum feature of size at MMC, degrees of rotation (orientation) and/or translation (location) already restrained by higher precedence datums shall remain restrained. Thus, DRF displacement may be further limited to translation along one or two axes and/or rotation about just one axis.

5.9.9.3 Benefits of DRF Displacement

As Fig. 5-52 shows, a TGC defines a datum, which, in turn, defines or helps define a DRF. This DRF, in turn, defines a framework of tolerance zones and/or acceptance boundaries for controlled features. Thus, allowable displacement between a datum feature surface and its TGC equates to identical displacement between the datum feature surface and the framework of tolerance zones. DRF displacement thereby allows freedom and flexibility in manufacturing, commensurate with what will occur in actual assembly. Because DRF displacement is a dynamic interaction, it's often confused with the other type of interaction, "bonus tolerance," described in section 5.6.5.1. Despite what anyone tells you:

Unlike "bonus tolerance," allowable DRF displacement never increases any tolerances. All virtual condition boundaries and/or tolerance zones remain the same size.

5.9.9.4 Effects of All Datums of the DRF

Allowable displacement of the entire DRF is governed by all the datums of that DRF acting in concert. In Fig. 5-75, datum boss B, acting alone as a primary datum, could allow DRF displacement including translation along three axes and rotation about three axes. Where datum A is primary and B is secondary (as shown), DRF displacement is limited to translation in two axes, and rotation only about the axis of B. Addition of tertiary datum C still permits some DRF displacement, but the potential for translation is not equal in all directions. Rotation of the DRF lessens the magnitude of allowable translation, and conversely, translation of the DRF lessens the magnitude of allowable rotation.

5.9.9.5 Effects of Form, Location, and Orientation

The actual form, location, and orientation of each datum feature in a DRF may allow unequal magnitudes for displacement in various directions. In Fig. 5-76, the datum shaft is out-of-round, but is still within its size limits. In Fig. 5-77, the tertiary datum boss deviates from true position, yet conforms to its positional tolerance. In both examples, the potential for DRF translation in the X-axis is significantly greater than in the Y-axis.

5.9.9.6 Accommodating DRF Displacement

In any DRF, the effects described above in sections 5.9.9.4 and 5.9.9.5 may combine to produce a potential for displacement with complex and interactive magnitudes that vary in each direction. As we said, the allowable displacement has no effect on the sizes of any virtual condition boundaries or tolerance zones for controlled features. DRF displacement may be completely and correctly accommodated by softgaging or (in MMC applications) by a functional gage. (See Chapter 19.) (The best way to learn about DRF displacement is to feel with your hands the clearances or "rattle" between a part and its functional gage.)

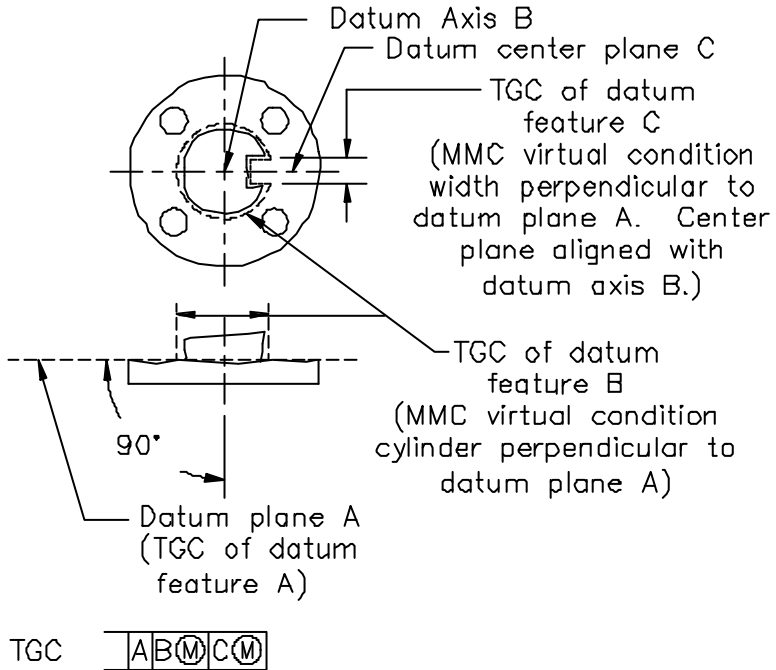
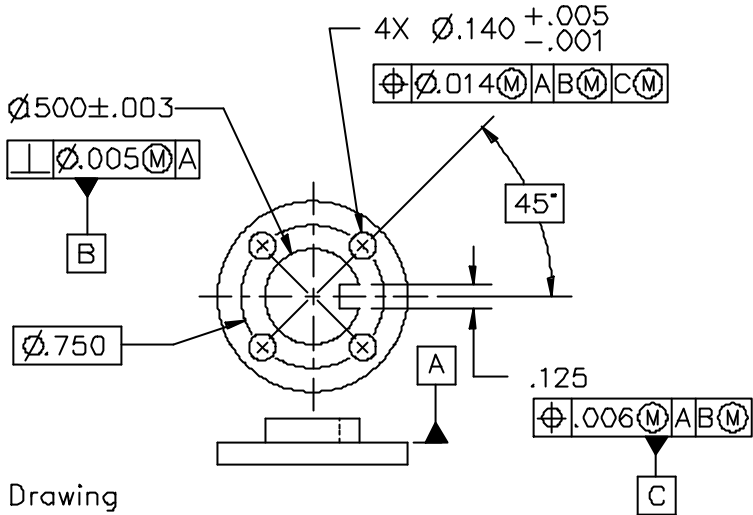


Figure 5-75 DRF displacement allowed by all the datums of the DRF

In DRFs having a single datum feature of size referenced at MMC, allowable displacement may be approximated by calculating the size difference between the datum feature's TGC and its actual mating envelope. Find the appropriate entities to use in Tables 5-3 and 5-4. For a primary datum feature, both the TGC and the actual mating envelope are unrestrained. For a secondary or tertiary datum feature, both entities must be restrained identically for proper results.

For example, in Fig. 5-67, secondary datum feature B's TGC is a cylindrical virtual condition boundary restrained perpendicular to datum A. To calculate allowable DRF displacement, we compare the size of this

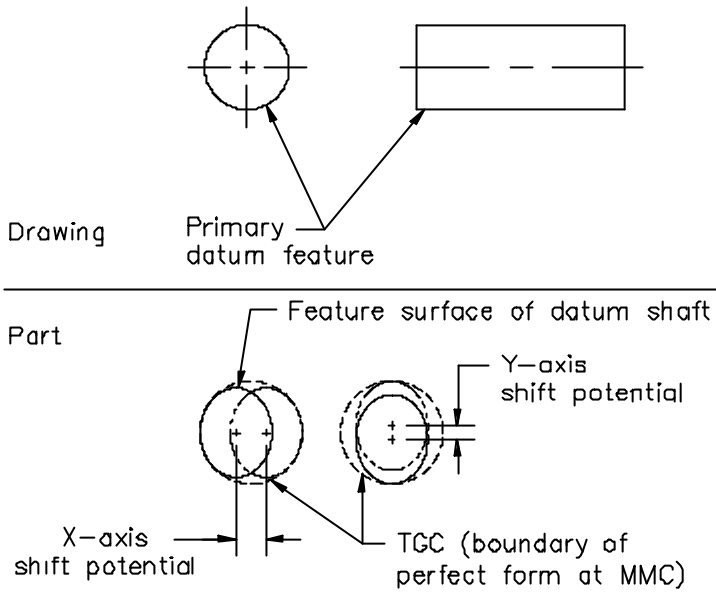


Figure 5-76 Unequal X and Y DRF displacement allowed by datum feature form variation

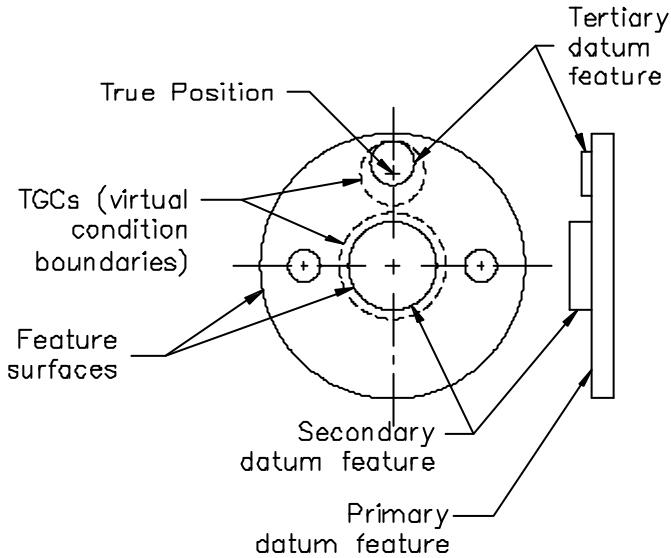


Figure 5-77 Unequal X and Y DRF displacement allowed by datum feature location variation

boundary ($\varnothing.134$) with datum feature B's actual mating size ($\varnothing.140$), derived from the actual mating envelope that is likewise restrained perpendicular to datum A. The calculated size difference ($\varnothing.006$) approximates the total clearance. With the actual mating envelope centered about the virtual condition boundary as shown, the clearance all around is uniform and equal to one-half the calculated size difference ($\varnothing.006 \div 2 = .003$). Thus, the DRF may translate up to that amount (.003) in any direction before the mating envelope and the TGC interfere. In our example, the $\varnothing.142$ unrestrained actual mating envelope is larger than the $\varnothing.140$ restrained envelope. Calculations erroneously based on the larger unrestrained envelope will overestimate the clearance all around, perhaps allowing acceptance of a part that won't assemble.

In using fitted envelopes, this simple approximation method is like the alternative center method described in section 5.6.5 and has similar limitations: It's awkward for LMC contexts, it doesn't accommodate allowable tilting, and the least magnitude for translation in any direction is applied uniformly in all directions. Consequently, it will reject some marginal parts that a proper functional gage will accept. Where used properly, however, this method will never accept a nonconforming part.

5.9.10 Simultaneous Requirements

We mentioned that DRF displacement emulates the variety of orientation and/or location relationships possible between two parts in assembly. In most cases, however, the parts will be fastened together at just one of those possible relationships. Thus, there shall be at least one relationship where all the holes line up, tab A fits cleanly into slot B, and everything works smoothly without binding. Stated more formally, there shall be a single DRF to which all functionally related features simultaneously satisfy all their tolerances. This rule is called *simultaneous requirements*.

By default, the "simultaneous requirements" rule applies to multiple features or patterns of features controlled to a "common" DRF having allowable DRF displacement. Obviously, DRF displacement can only occur where one or more of the datum features is a feature of size or bounded feature referenced at MMC or LMC. Fig. 5-78 demonstrates why "common DRF" must be interpreted as "identical DRF."

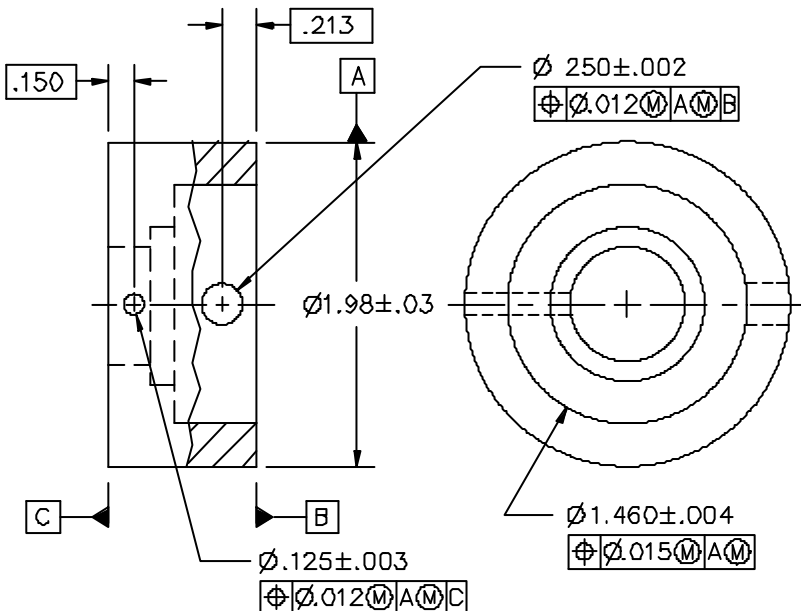


Figure 5-78 "Common DRF" means "identical DRF"

Though primary datum A is “common” to all three feature control frames, we can’t determine whether the DRF of datum A alone should share simultaneous requirements with A|B or with A|C. Thus, no simultaneous requirements exist unless there is a one-to-one match of datum references, in the same order of precedence, and with the same modifiers, as applicable.

The part in Fig. 5-79 will assemble into a body where all the features will mate with fixed counterparts. The designer must assure that all five geometrically controlled features will fit at a single assembly relationship. Rather than identifying the slot or one of the holes as a clocking datum, we have controlled all five features to a single DRF. The angular relationships among the .125 slot and the holes are fixed by 90° and 180° basic angles implied by the crossing center lines, according to Fundamental Rule (j). As a result, all five features share simultaneous requirements, and all five geometric tolerances can be inspected with a single functional gage in just a few seconds.

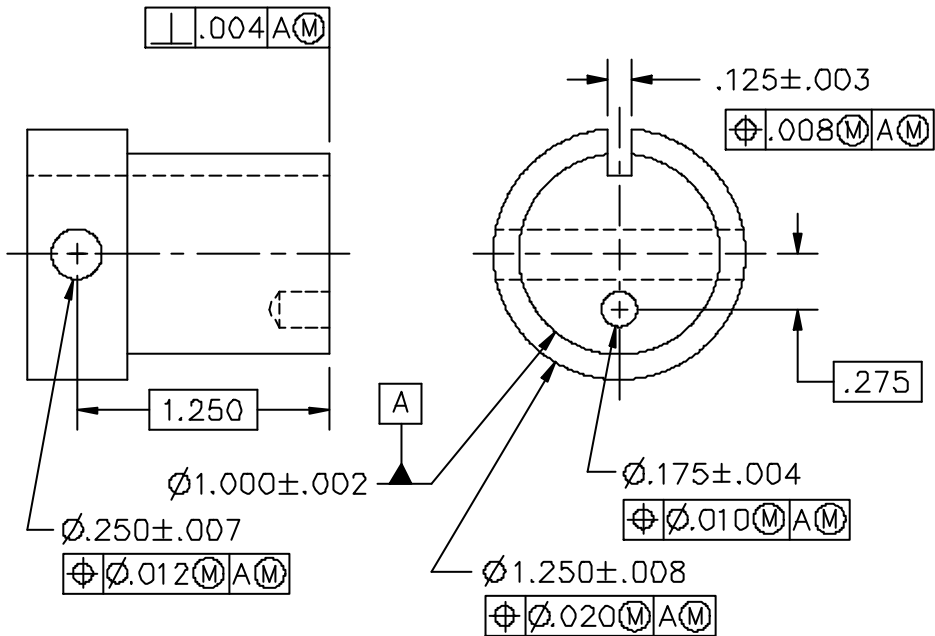


Figure 5-79 Using simultaneous requirements rule to tie together the boundaries of five features

Without such a gage, simultaneous requirements can become a curse. An inspector may be required to make multiple surface plate setups, struggling to reconstruct each time the identical DRF. Older CMMs generally establish all datums as if they were RFS, simply ignoring allowable DRF displacement. That’s fine if all simultaneous requirement features conform to that fixed DRF. More sophisticated CMM software can try various displacements of the DRF until it finds a legitimate one to which all the controlled features conform.

Given the hardships it can impose, designers should nullify the “simultaneous requirements” rule wherever it would apply without functional benefit. Do this by placing the note SEP REQT adjacent to each applicable feature control frame, as demonstrated in Fig. 5-80. Where separate requirements are allowed, a part may still be accepted using a common setup or gage. But a “SEP REQT” feature (or pattern) cannot be deemed discrepant until it has been evaluated separately. For details on how simultaneous or separate requirements apply among composite and stacked feature control frames, see section 5.11.7.3 and Table 5-7.

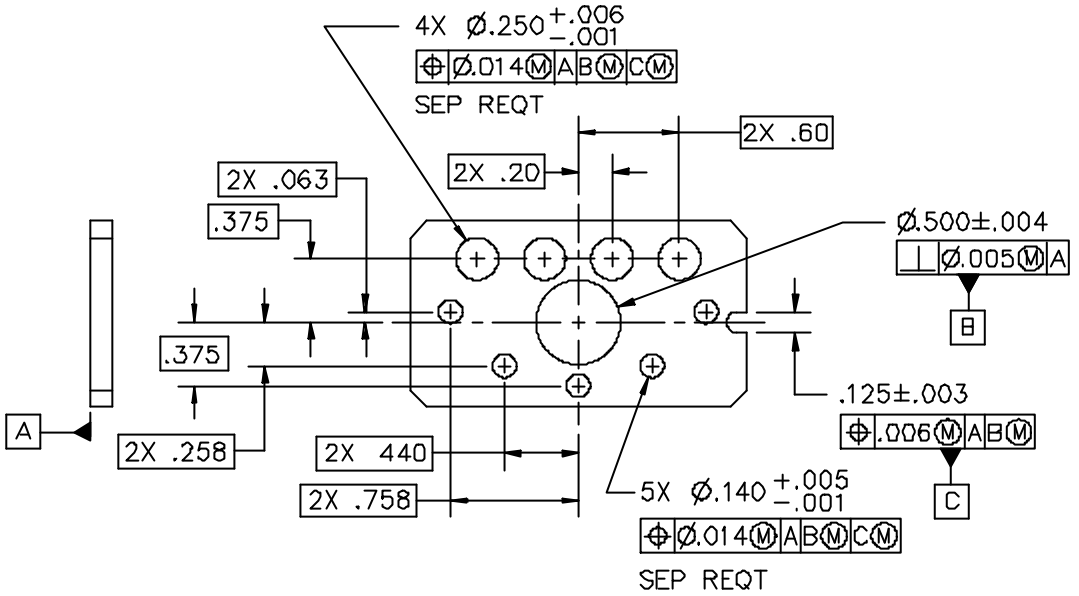


Figure 5-80 Specifying separate requirements

FAQ: *Do simultaneous requirements include profile and orientation tolerances?*

A: Y14.5 shows an example where simultaneous requirements include a profile tolerance, but neither standard mentions the rule applying to orientation tolerances. We feel that, by extension of principle, orientation tolerances are also included automatically, but a designer might be wise to add the note **SIM REQ** adjacent to each orientation feature control frame that should be included, as we have in Fig. 5-81.

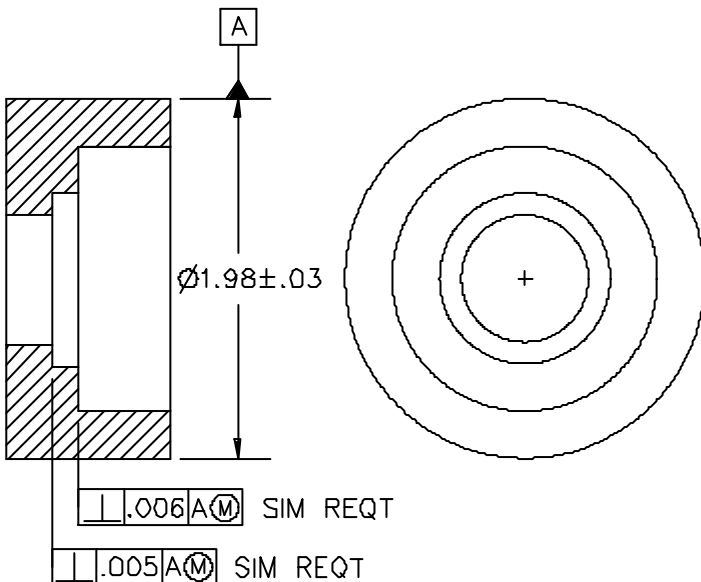


Figure 5-81 Imposing simultaneous requirements by adding a note

5.9.11 Datum Simulation

In sections 5.9.8.1 through 5.9.8.4, we discussed how perfectly shaped TGCs are theoretically aligned, fitted, or otherwise related to their datum features. The theory is important to designers, because it helps them analyze their designs and apply proper geometric controls. But an inspector facing a produced part has no imaginary perfect shapes in his toolbox. What he has instead include the following:

- Machine tables and surface plates (for planar datum features)
- Plug and ring gages (for cylindrical datum features)
- Chucks, collets, and mandrels (also for cylindrical datum features)
- Contoured or offset fixtures (for mathematically defined datum features)

Inspectors must use such high quality, but imperfect tools to derive datums and establish DRFs. The process is called *datum simulation* because it can only simulate the true datums with varying degrees of faithfulness. The tools used, called *datum feature simulators*, though imperfect, are assumed to have a unique tangent plane, axis, center plane, or center point, called the *simulated datum*, that functions the same as a theoretical datum in establishing a DRF.

Fig. 5-52 shows the relationship between the terms Y14.5 uses to describe the theory and practice of establishing datums. Errors in the form, orientation, and/or location of datum simulators create a discrepancy between the simulated datum and the true datum, so we always seek to minimize the magnitude of such errors. “Dedicated” tools, such as those listed above, are preferred as simulators, because they automatically find and contact the surface high points. Alternatively, flexible processing equipment, such as CMMs may be used, but particular care must be taken to seek out and use the correct surface points. The objective is to simulate, as nearly as possible, the theoretical contact or clearance between the TGC and the datum feature’s high or tangent points. Table 5-4 includes examples of appropriate datum feature simulators for each type of datum feature.

5.9.12 Unstable Datums, Rocking Datums, Candidate Datums

Cast and forged faces tend to be bowed and warped. An out-of-tram milling machine will generate milled faces that aren’t flat, perhaps with steps in them. Sometimes, part features distort during machining and heat treating processes. Fig. 5-82 shows a datum feature surface that’s convex relative to its tangent TGC

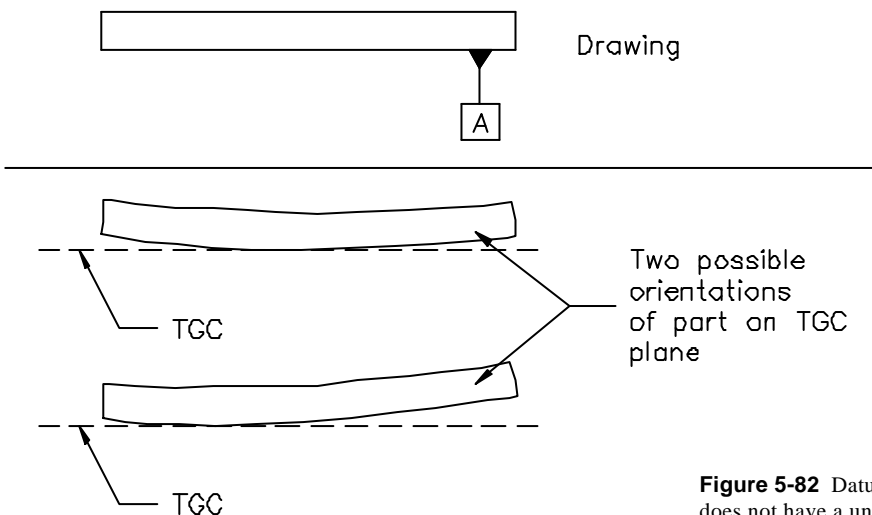


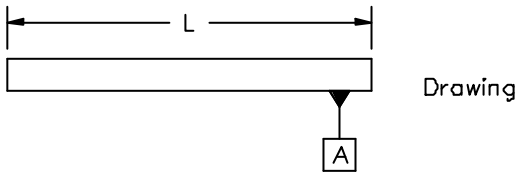
Figure 5-82 Datum feature surface that does not have a unique three-point contact

plane, and can't achieve a unique three-point contact relationship. In fact, contact may occur at just one or two high points. This is considered an "unstable" condition and produces what's called a *rocking datum*. In other words, there are a variety of tangent contact relationships possible, each yielding a different *candidate datum* and resulting *candidate datum reference frame*. These terms derive from the fact that each "candidate" is qualified to serve as the actual datum or DRF. The standards allow a user to elect any single expedient candidate datum.

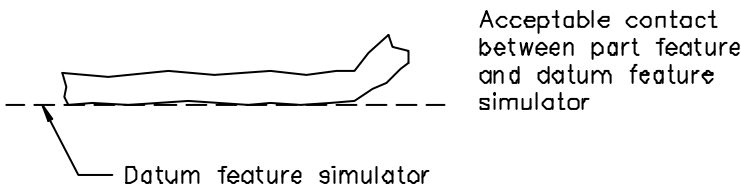
Let's suppose an inspector places a part's primary datum face down on a surface plate (a datum simulator) and the part teeters under its own weight. The inspector needs the part to hold still during the inspection. Y14.5 states the inspector may "adjust" the part "to an optimum position," presumably a position where all features that reference that DRF conform to their tolerances. The prescribed "adjustment" usually involves placing some shims or clay strategically between the part and the surface plate.

The only way a CMM can properly establish a usable candidate datum from a rocking surface is by collecting hundreds or even thousands of discrete points from the surface and then modeling the surface in its processor. It must also have data from all features that reference the subject DRF. Then, the processor must evaluate the conformance of the controlled features to various candidate DRFs until it finds a candidate DRF to which all those features conform.

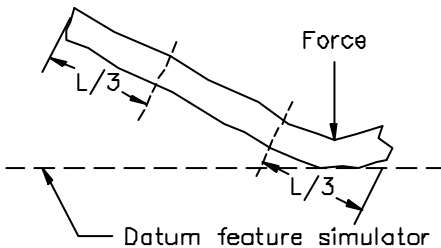
We mentioned an example part that "teeters under its own weight," but really, neither standard cites gravity as a criterion for candidate datums. A part such as that shown in Fig. 5-83 may be stable under its own weight, but may rock on the surface plate when downward force is applied away from the center of gravity. In fact, one side of any part could be lifted to a ludicrous angle while the opposite edge still makes one- or two-point contact with the simulator. Recognizing this, the Math Standard added a restriction saying (roughly simplified) that for a qualified candidate datum, the TGC's contact point(s) cannot all lie on one "side" of the surface, less than one-third of the way in from the edge. (One-third is the default; the drawing can specify any fraction.) This restriction eliminates, at least in most cases, "optimizations," such as shown at the bottom of Fig. 5-83, that might be functionally absurd.



Drawing



Acceptable contact between part feature and datum feature simulator



Unacceptable contact between part feature and datum feature simulator

Figure 5-83 Acceptable and unacceptable contact between datum feature and datum feature simulator

This entire “adjusting to an optimum position” scheme is fraught with pitfalls and controversy. Depending on the inspection method, the optimization may not be repeatable. Certainly, the part will not achieve the same artificially optimized orientation in actual assembly. For example, a warped mounting flange might flatten out when bolted down, not only invalidating the DRF to which the part conformed in inspection, but possibly physically distorting adjacent features as well. It’s fairly certain the designer didn’t account for a rocking datum in his tolerance calculations.

FAQ: *Can’t we come up with a standard method for deriving a unique and repeatable datum from a rocker?*

A: A variety of methods have been proposed, each based on different assumptions about the form, roughness, rigidity, and function of typical features. But this debate tends to eclipse a larger issue. A rocking datum feature betrays a failure in the design and/or manufacturing process, and may portend an even larger disaster in the making. Rather than quarrel over how to deal with rocking datums, we believe engineers should direct their energies toward preventing them. Designers must adequately control the form of datum features. They should consider datum targets (explained below) for cast, forged, sawed, and other surfaces that might reasonably be expected to rock. Manufacturing engineers must specify processes that will not produce stepped or tottering datum features. Production people must be sure they produce surfaces of adequate quality. Inspectors finding unstable parts should report to production and help correct the problem.

5.9.13 Datum Targets

So far, we’ve discussed how a datum is derived from an entire datum feature. TGC (full-feature) datum simulation demands either a fixture capable of contacting any high points on the datum feature, or sampling the entire datum feature with a probe. These methods are only practicable, however, where the datum feature is relatively small and well formed with simple and uniform geometry. Few very large datum features, such as an automobile hood or the outside diameter of a rocket motor, mate with other parts over their entire length and breadth. More often, the assembly interface is limited to one or more points, lines, or small areas. Likewise, **non-planar or uneven surfaces produced by casting, forging, or molding; surfaces of weldments; and thin-section surfaces subject to bowing, warping, or other inherent or induced distortions** rarely mate or function on a full-feature basis. More than just being impracticable and cost prohibitive in such cases, full-feature simulation could yield erroneous results. The obvious solution is to isolate only those pertinent points, lines, and/or limited areas, called *datum targets*, to be used for simulation. The datum thus derived can be used the same as a datum derived from a TGC. It can be referenced alone, or combined with other datums to construct a DRF.

5.9.13.1 Datum Target Selection

For each “targeted” datum feature, the type of target used should correspond to the type of mating feature or to the desired simulator and the necessary degree of contact, according to the following table.

Multiple target types may be combined to establish a single datum. However, the type(s), quantity, and placement of datum targets on a feature shall be coordinated to restrain the same degrees of freedom as would a full-feature simulator. For example, a targeted primary datum plane requires a minimum of three noncolinear points, or a line and a noncolinear point, or a single area of sufficient length and breadth. While the number of targets should be minimized, additional targets may be added as needed to simulate assembly, and/or to support heavy or nonrigid parts. For example, the bottom side of an automobile hood

Table 5-6 Datum target types

MATING FEATURE OR SIMULATOR TYPE	TARGET TYPE
spherical or pointed	POINT (0-dimensional contact)
“side” of a cylinder or “knife” edge	LINE (1-dimensional contact)
flat or elastic “pad” area	AREA (2-dimensional contact)

may need six or more small target areas. Unless target locations correspond to mating interfaces, multiple targets for a single datum should be spread as far apart as practicable to provide maximum stability.

5.9.13.2 Identifying Datum Targets

First, wherever practicable, the datum feature itself should be identified in the usual way with a “datum feature” symbol to clarify the DRF origin. As detailed in the following paragraphs, each datum target is shown on or within the part outline(s) in one or more views. Outside the part outline(s), one “datum target” symbol is leader directed to each target point, line, and area. Where the target is hidden in the view, perhaps on the far side of the part, the leader line shall be dashed. The “datum target” symbol is a circle divided horizontally into halves. See Figs. 5-8 and 5-84. The lower half always contains the target label, consisting of the datum feature letter, followed by the target number, assigned sequentially starting with 1 for each datum feature. The upper half is either left blank, or used for defining the size of a target area, as described below.

Datum Target Point—A datum target point is indicated by the “target point” symbol, dimensionally located on a direct view of the surface or on two adjacent views if there’s no direct view. See Fig. 5-85.

Datum Target Line—A datum target line is indicated by the “target point” symbol on an edge view of the surface, a phantom line on the direct view, or both. See Fig. 5-85. The location (in one or two axes) and length of the datum target line shall be directly dimensioned as necessary.

Datum Target Area—A datum target area is indicated on a direct view of the surface by a phantom outline of the desired shape with section lines inside. The location (in one or two axes) and size of the datum target area shall be dimensioned as necessary. See Fig. 5-84(a) and (b). Notice that the diameter value of the target area is either contained within the upper half of the “datum target” symbol (space permitting) or leader directed there. Where it’s not practicable to draw a circular phantom outline, the “target point” symbol may be substituted, as in Fig. 5-84(c).

FAQ: *Can the upper half of the “datum target” symbol be used to specify a noncircular area?*

A: Nothing in the standard forbids it. A size value could be preceded by the “square” symbol instead of the “diameter” symbol. A rectangular area, such as .25 X .50, could also be specified. The phantom outline shall clearly show the orientation of any noncircular target area.

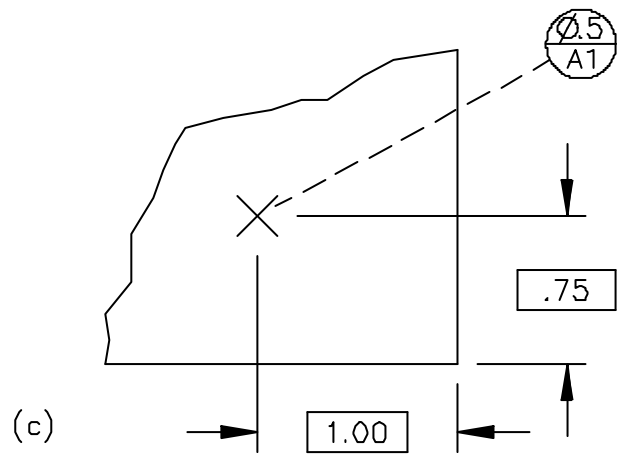
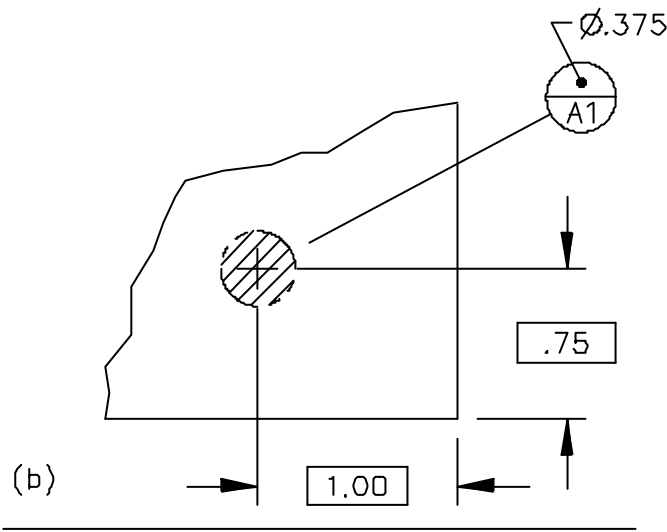
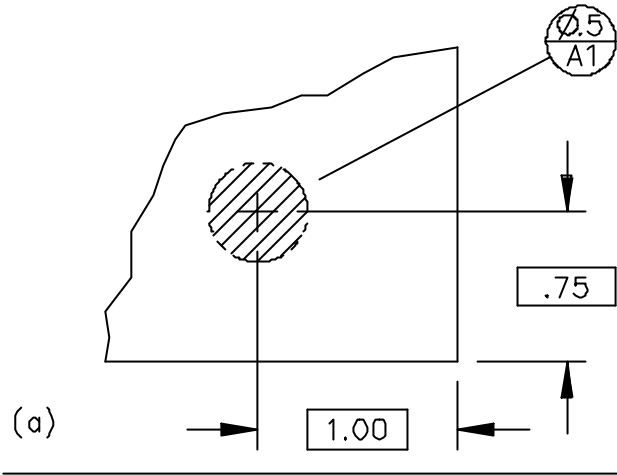


Figure 5-84 Datum target identification

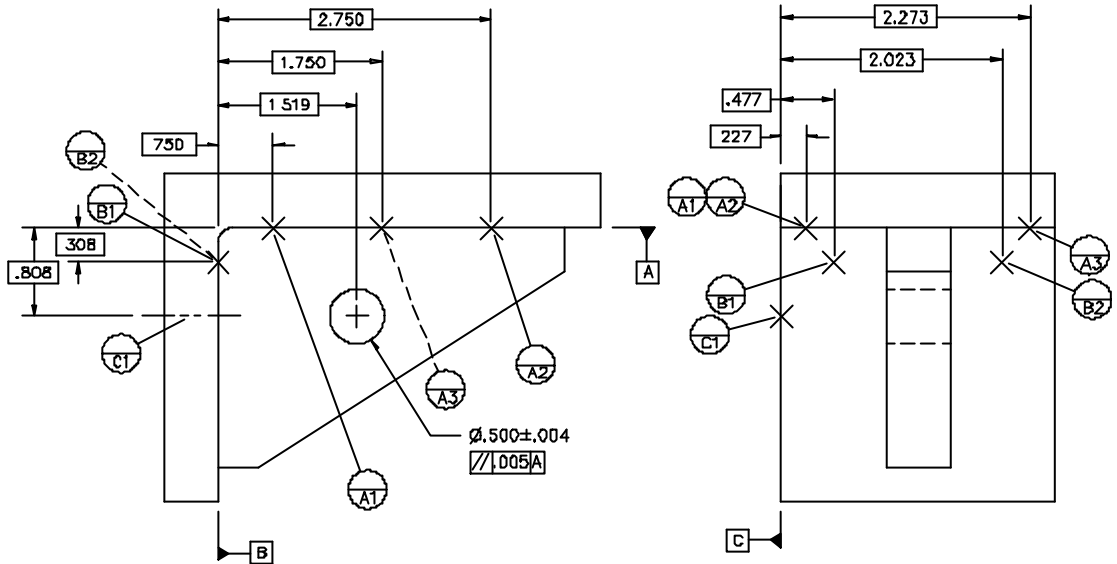


Figure 5-85 Datum target application on a rectangular part

5.9.13.3 Datum Target Dimensions

The location and size, where applicable, of datum targets are defined with either basic or toleranced dimensions. If defined with basic dimensions, established tooling or gaging tolerances apply. Such dimensions are unconventional in that they don't pertain to any measurable attribute of the part. They are instead specifications for the process of datum simulation, in effect saying, "Simulation for this datum feature shall occur here."

On any sample part, the datum simulation process may be repeated many times with a variety of tools. For example, the part could be made in multiple machines, each having its own fixture using the datum targets. The part might then be partially inspected with a CMM that probes the datum feature only at the datum targets. Final inspection may employ a functional gage that uses the datum targets. Thus, dimensions and tolerances for a datum target actually apply directly to the location (and perhaps, size) of the simulator (contacting feature) on each tool, including CMM probe touches. Variations within the applicable tolerances contribute to discrepancies between the DRFs derived by different tools.

FAQ: *Where can I look up "established tooling or gaging tolerances" for locating simulators?*

A: We're not aware of any national or military standard and it's unlikely one will emerge. The traditional rule of thumb—5% or 10% of the feature tolerance—is quite an oversimplification in this context. (And to which feature would it refer?) While tolerances of controlled features are certainly a factor in determining target tolerances, there are usually many other factors, including the form and surface roughness of the datum feature, and the type and size of the simulator. For example, on a forged surface, the point of contact of a \varnothing 1mm spherical simulator is usually more critical than that of a \varnothing 4mm simulator. (Both are common CMM styli.)

5.9.13.4 Interdependency of Datum Target Locations

In Fig. 5-85, three targeted datum features establish a DRF. Notice that targets A1, A2, and A3 are located relative to datums B and C. Targets B1 and B2 are located relative to datums A and C. Likewise, target C1 is located relative to datums A and B. This interdependency creates no problem for hard tooling that simulates all three datums simultaneously. However, methods that simulate the datums sequentially encounter a paradox: The targets for any one datum cannot be accurately found until the other two datums have been properly established. A CMM, for example, may require two or three iterations of DRF construction to achieve the needed accuracy in probing the targets. Even for the simple parallelism callout that references only datum A, all three datums must be simulated and the entire A|B|C DRF properly constructed.

FAQ: *Should the parallelism callout in Fig. 5-85 reference all three datums, then, A|B|C?*

A: No. Referencing datum B would add an unnecessary degree of restraint to the parallelism tolerance. An excellent solution is to extend positional tolerancing principles (RFS) to datum targets. See section 11. A feature control frame complete with datum references may be placed beneath the #1 “datum target” symbol for each datum (for example, A1, B1, and C1). This method overcomes all the shortcomings of plus and minus coordinate tolerancing, and unambiguously controls the locations of all six targets to a common and complete DRF. (In our example, A|B|C should be referenced for each of the three target sets.) The standard neither prohibits nor shows this method, so a drawing user might welcome guidance from a brief general note.

5.9.13.5 Applied to Features of Size

Datum targets may be applied to a datum feature of size for RFS simulation. The simulators shall be adjustable to contact the feature at all specified targets. Simulators on hard tools shall expand or contract uniformly while maintaining all other orientation and location relationships relative to each other and to other datums in the subject DRF.

Width-Type Feature—In the tertiary datum slot in Fig. 5-86, simulators C1 and C2 shall expand apart. Proper simulation is achieved when each simulator contacts the slot, each is equidistant from datum plane BY, and each is the specified distances from datum planes A and BX.

Cylindrical Feature—A datum target line or area may be wrapped around a cylindrical feature, specifying what amounts to a TGC of zero or limited length. Alternatively, datum target points or lines (longitudinal) may be equally spaced around the feature. For the secondary datum boss in Fig. 5-86, simulators B1, B2, and B3 shall contract inward to trap the feature. A hard tool, perhaps a precision chuck, shall have a set of three equally spaced simulators (jaws) capable of moving radially at an equal rate from a common axis. Proper simulation is achieved when each simulator contacts the boss and each is equidistant from the datum axis.

Poor feature form, orientation, or location may prevent one or more simulators from making contact, despite obeying all the rules. Where, for example, we need to derive a primary datum from a forged rod, we may specify target points A1, A2, and A3 around one end, and A4, A5, and A6 around the other end. This requires all six simulators to contract uniformly. The larger rod end will be trapped securely, while at the smaller end, never more than two simulators can touch. This yields a rocking datum. One solution is to relabel A4, A5, and A6 as B1, B2, and B3, and then establish co-datum A-B. This allows the two simulator sets, A and B, to contract independently of each other, thereby ensuring contact at all six targets.

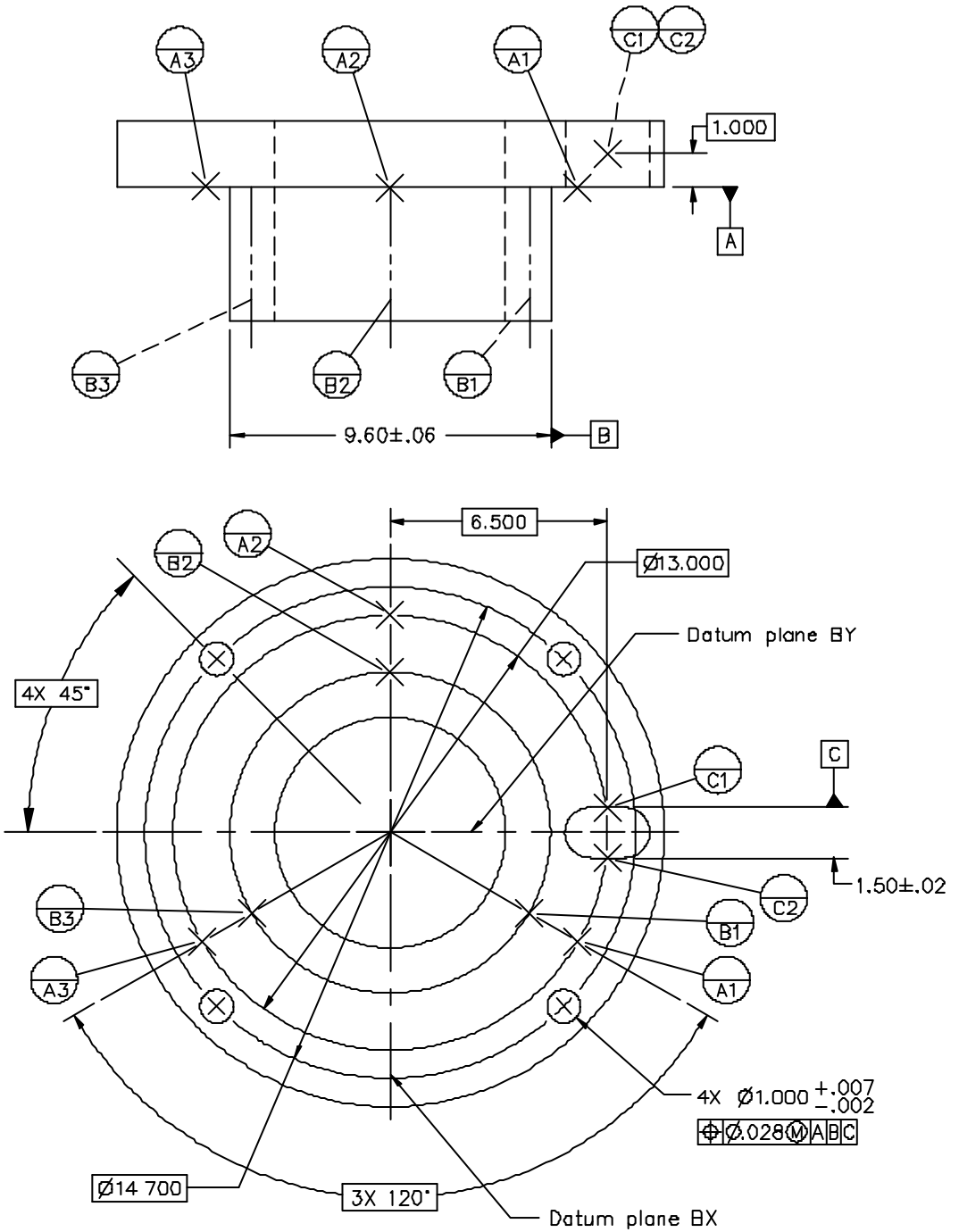


Figure 5-86 Datum target application on a cylindrical part

FAQ: *Can datum targets be applied to a feature of size on an MMC basis?*

A: Nothing in the standard precludes it. We've been careful to emphasize that datum targets are targets for simulation, not necessarily contact. For MMC, a typical hard tool would have simulators at a fixed diameter or width based on the datum feature's MMC. With advanced software, a CMM can easily accommodate MMC and LMC applications. All the DRF displacement principles of section 5.9.9 apply, except that the target set does not comprise a TGC.

5.9.13.6 Applied to Any Type of Feature

Datum targets provide the means for simulating a usable datum from any imaginable type and shape of feature. With irregular datum features, the designer must carefully assure that all nonadjustable relationships between targets are dimensioned, preferably using just one coordinate system. Any relationship between targets left undimensioned shall be considered adjustable.

Particularly with a complex drawing, a drawing user may have trouble identifying a datum plane or axis derived and offset from a stepped or irregularly shaped datum feature. In such cases only, it's permissible to attach a "datum feature" symbol to a center line representing the datum.

Stepped Plane—A datum plane can be simulated from multiple surfaces that are parallel but not coplanar. Datum targets should be defined such that at least one target lies in the datum plane. Offset distances of other targets are defined with dimensions normal to the datum plane. This also permits convenient application of profile tolerancing to the part surfaces.

Revolutes—A revolute is generated by revolving a 2-D spine (curve) about a coplanar axis. This can yield a cone (where the spine is a straight nonparallel line), a toroid (where the spine is a circular arc), or a vase or hourglass shape. It may be difficult or impossible to define TGCs for such shapes. Further, full-feature datum simulation based on nominal or basic dimensions may not achieve the desired fit or contact. Where a revolute must be referenced as a datum feature, it's a good idea to specify datum targets at one or two circular elements of the feature. At each circular element, a triad of equally spaced datum target points or lines, or a single circular target "line" may be used.

Fig. 5-87 shows a datum axis derived from a chicken egg. Targets A1, A2, and A3 are equally spaced on a fixed $\varnothing 1.250$ basic circle. These simulators neither expand nor contract relative to each other. Targets B1, B2, and B3 are likewise equally spaced on a fixed $\varnothing 1.000$ basic circle. The drawing implies basic coaxiality and clocking between the two target sets. However, the distance between the two sets is undimensioned and therefore, adjustable. This distance shall close until contact occurs at all six targets and the egg is immobilized. In the positional tolerance feature control frame for the egg's $\varnothing .250$ observation port (peephole), co-datum axis A-B is referenced RFS (see section 5.9.14.2). The $.500$ basic dimension for the observation hole originates from the plane of the datum A target set.

Fig. 5-88 shows one possible setup for drilling the observation hole. Despite the egg's frailty, we've chosen pointed simulators over spherical ones to assure that contact always occurs at the specified basic diameters. Simulators A1, A2, and A3 are affixed to the "stationary" jaw of a precision vise. Simulators B1, B2, and B3 are attached to the "movable" vise jaw. "Stationary" and "movable" are always relative terms. In this case, mobility is relative to the machine spindle.

To simulate the egg's datum axis at MMC, a basic or toleranced dimension shall be added for the distance between the two triads of targets. The targets are labeled A1 through A6 and establish datum axis A (where A is any legal identifying letter). Since none of the simulators would be adjustable in any direction, the egg can rattle around between them. (On a hard tool, one or more simulators would have to be removable to let the egg in and out.)

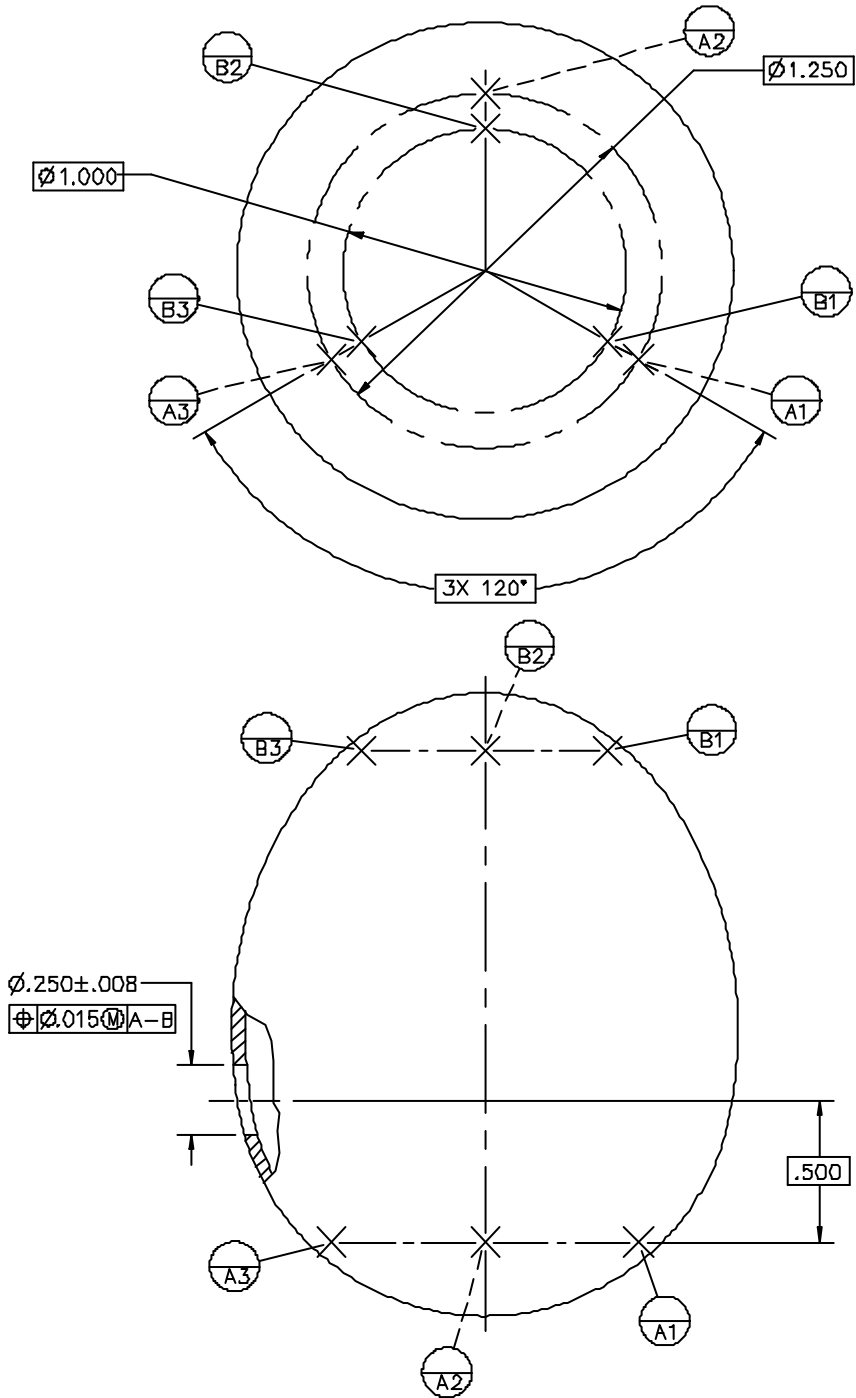


Figure 5-87 Using datum targets to establish a primary axis from a revolute

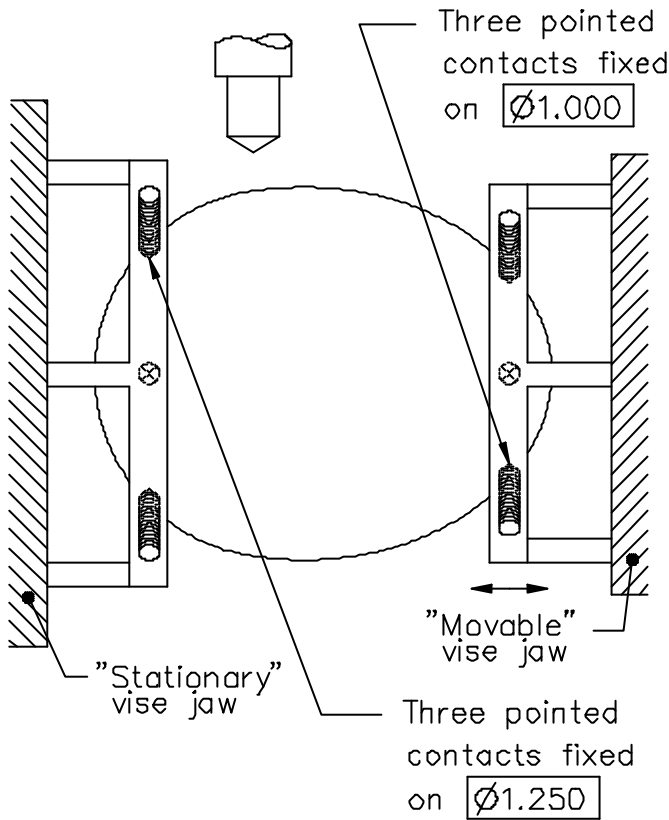


Figure 5-88 Setup for simulating the datum axis for Fig. 5-87

Math-Defined Feature—Datum targets can be placed on radii, spherical radii, and any type of nominally warped planar surface. The desired datum planes can establish a coordinate system for defining the location of each target in 3-D space. In some cases, it may be simpler if every target is offset from the datum planes.

Bounded Feature—All the above principles can apply.

5.9.13.7 Target Set with Switchable Precedence

In Fig. 5-89, datum B is the primary datum for a parallelism tolerance, so we’ve identified the minimum necessary target points, B1, B2, and B3. However, in the other DRF, A|B|C, datum B is the secondary datum. Here, we only need and want to use points B1 and B2. On a very simple drawing, such as ours, a note can be added, saying, “IN DATUM REFERENCE FRAME A|B|C, OMIT TARGET B3.” On a more complex drawing, a table like the one below could be added. The right column can list either targets to use or targets to omit, whichever is simpler.

IN DATUM REFERENCE FRAME	OMIT TARGET(S)
A B C	B3
B A	A3
D E F	E3, F2, F3

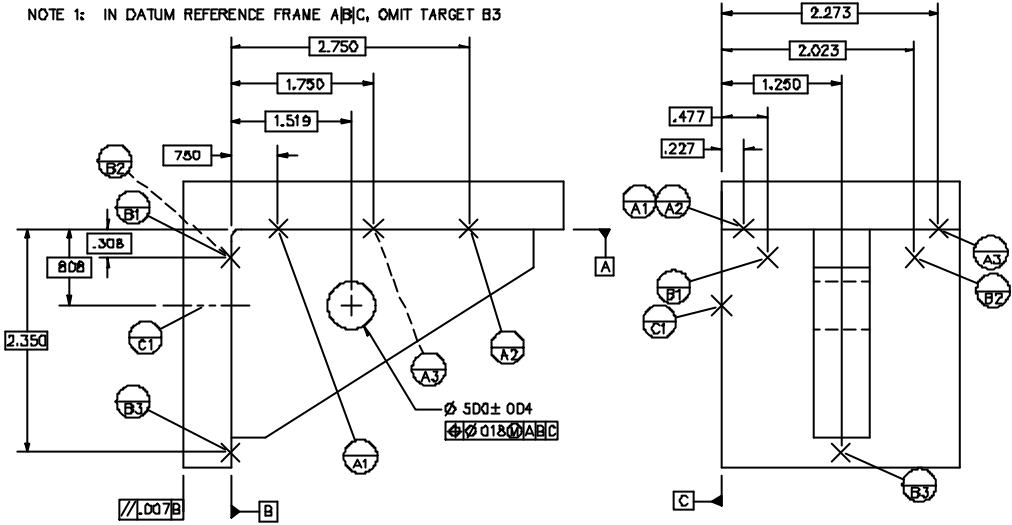


Figure 5-89 Target set with switchable datum precedence

5.9.14 Multiple Features Referenced as a Single Datum Feature

In some cases, multiple features can be teamed together and treated as a single datum feature. This is a frontier of datuming, not fully developed in the standards. When referencing multiple features in this way, designers must be extremely careful to understand the exact shapes, sizes (where applicable), and interrelationships of the TGC(s); simulation tools that might be used; and the exact degrees of freedom arrested. If any of these considerations won't be obvious to drawing users, the designer must explain them in a drawing note or auxiliary document.

5.9.14.1 Feature Patterns

While discussing Fig. 5-54, we said the cylinder head's bottom face is an obvious choice for the primary datum feature. The two dowel holes are crucial in orienting and locating the head on the block. One hole could be the secondary datum feature and the other tertiary, but the holes would then have unequal specifications requiring unequal treatment. Such datum precedence is counterintuitive, since both holes play exactly equal roles in assembly. This is an example where a pattern of features can and should be treated as a single datum feature. Rather than a single axis or plane, however, we can derive two perpendicular datum planes, both oriented and located relative to the holes.

Fig. 5-90 shows just three of many options for establishing the origin from our pattern of dowel holes. The designer must take extra care to clarify the relationship between a datum feature pattern and the origins of the coordinate system derived therefrom.

Fig. 5-91 shows a feature pattern referenced as a single datum feature at MMC. Rather than a single TGC, the datum B reference establishes a pattern or *framework* of multiple, identical, fixed-size TGCs. Within this framework, the orientation and location of all the TGCs are fixed relative to one another according to the basic dimensions expressed on the drawing. As the figure's lower portion shows, two perpendicular planes are derived, restricting all three remaining degrees of freedom. For discussion pur-

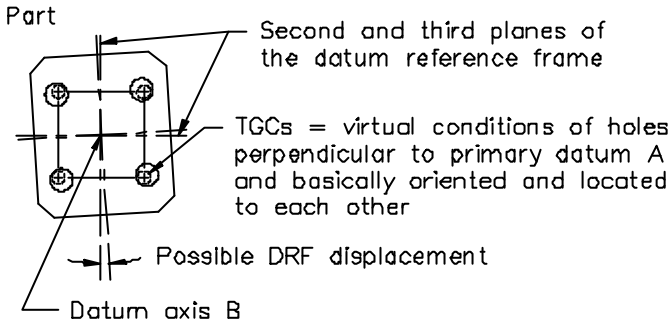
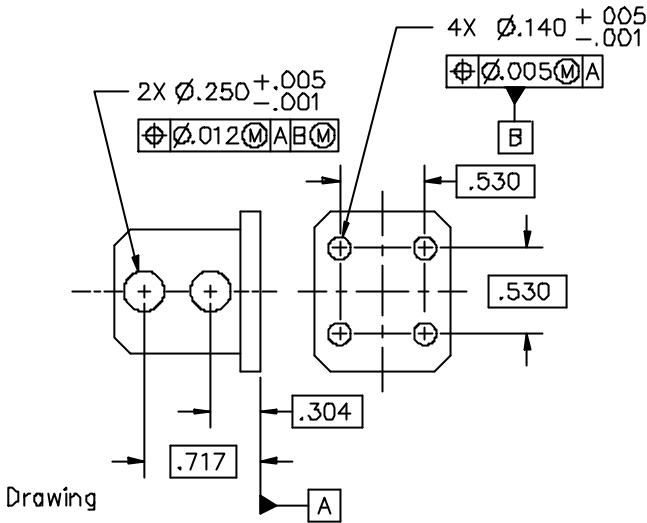


Figure 5-91 Pattern of holes referenced as a single datum at MMC

poses, we've labeled the intersection of these planes "datum axis B." Since each individual feature in the pattern clears its respective TGC, DRF displacement is possible, including rotation about datum axis B, and translation in any direction perpendicular to datum axis B. The rules for simultaneous requirements are the same as if datum feature B were a single feature.

FAQ: *Can a datum feature pattern be referenced at LMC or RFS?*

A: At LMC, yes, but this will require softgaging. The datum feature simulator is a set of virtual fixed-size TGCs. For RFS, the simulator should be a set of adjustable TGCs, each expanding or contracting to fit its individual feature. But differences among the size, form, orientation, and location of individual features raise questions the standards don't address. Must the TGCs adjust simultaneously and uniformly? Must they all end up the same size? In such a rare application, the designer must provide detailed instructions for datum simulation, because the standards don't.

5.9.14.2 Coaxial and Coplanar Features

Fig. 5-131 shows another example of separate features—this time, two bearing journals—that have exactly equal roles in orienting and locating the shaft in assembly. Again, to give one feature precedence over the other seems inappropriate. Here, however, the features are not the same size, and can't be considered a feature pattern.

The solution is to identify each datum feature separately, but include both identifying letters in a single datum reference, separated by a hyphen. It doesn't matter which letter appears first in the compartment, since neither datum feature has precedence over the other.

Rather than a single TGC, a hyphenated co-datum reference establishes a pair of perfectly coaxial or coplanar TGCs (depending on the feature types). In our example, datum features A and B are both referenced RFS. Their TGCs are coaxial actual mating envelopes that shall contract independently until each makes a minimum of two-point contact, jointly arresting four degrees of freedom. Hyphenated co-datum features are usually the same type of feature, with matching material conditions, and thus, matching TGC types. But not necessarily. The principle is equally applicable at MMC, LMC, or any pairing of material conditions.

FAQ: *How can this simulation scheme work if the two datum features are badly eccentric?*

A: The simulation will still work, but the part might not. Deriving meaningful datums (and DRFs, for that matter) from multiple features always demands careful control (using GD&T) of the orientation and location relationships between the individual datum features. For our example shaft, section 5.12.4 and Fig. 5-132 describe an elegant way to control coaxiality between the two bearing journals.

5.9.15 Multiple DRFs

On larger and/or more complicated parts, it may be impractical to control all features to a single DRF. Where features have separate functional relationships, relating them to the same DRF might be unnecessarily restrictive. Multiple DRFs may be used, but only with great care. Designers typically use too many datums and different DRFs, often without realizing it. Remember that any difference in datum references, their order of precedence, or their material conditions, constitutes a separate DRF. The tolerances connecting these DRFs start stacking up to where the designer quickly loses control of the part's overall integrity. A good way to prevent this and to unify the design is to structure multiple DRFs as a tree. That means controlling the datum features of each "branch" DRF to a common "trunk" DRF.

5.10 Orientation Tolerance (Level 3 Control)

Orientation is a feature's angular relationship to a DRF. An *orientation tolerance* controls this relationship without meddling in location control. Thus, an orientation tolerance is useful for relating one datum feature to another and for refining the orientation of a feature already controlled with a positional tolerance.

5.10.1 How to Apply It

An orientation tolerance is specified using a feature control frame displaying one of the three orientation characteristic symbols. See Fig. 5-92. The symbol used depends on the basic orientation angle, as follows.

0° or 180°—“parallelism” symbol
 90° or 270°—“perpendicularity” symbol
 any other angle—“angularity” symbol

All three symbols work exactly the same. The only difference is that where the “angularity” symbol is used, a basic angle shall be explicitly specified. Where the “parallelism” or “perpendicularity” symbol is used, the basic angle is implied by a drawing view that shows the parallel or perpendicular relationship. Though a single generic “orientation” symbol has been proposed repeatedly, most users prefer separate symbols for parallelism and perpendicularity because each tells the whole story at a glance. The feature control frame includes the orientation tolerance value followed by one or two datum references.

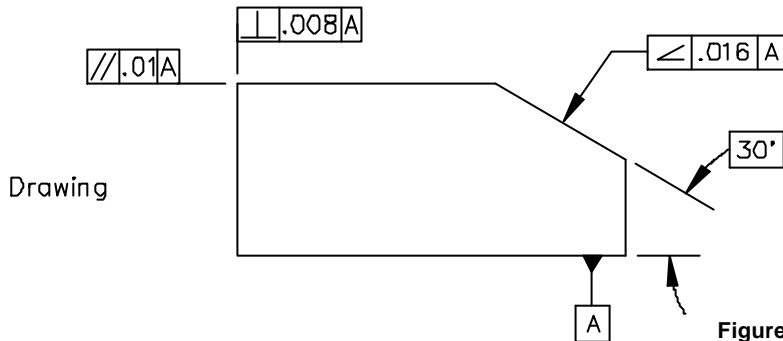


Figure 5-92 Application of orientation tolerances

5.10.2 Datums for Orientation Control

Orientation control requires a DRF. A primary datum plane or axis always establishes rotation about two axes of the DRF and is usually the only datum reference needed for orientation control. There are cases where it's necessary to establish rotation about the third axis as well and a secondary datum reference is needed. Sometimes, a secondary datum is needed to orient and/or locate a tolerance zone plane for controlling line elements of a feature. In other cases, hyphenated co-datums (see section 5.9.14.2) may be used to arrest rotation. Since all three rotational degrees of freedom can be arrested with just two datums, a tertiary datum is usually meaningless and confusing.

5.10.3 Applied to a Planar Feature (Including Tangent Plane Application)

Any nominally flat planar feature can be controlled with an orientation tolerance. Fig. 5-93 shows the tolerance zone bounded by two parallel planes separated by a distance equal to the tolerance value. The surface itself shall be contained between the two parallel planes of the tolerance zone. Form deviations including bumps, depressions, or waviness in the surface could prevent its containment. Thus, an orientation tolerance applied to a plane also controls flatness exactly the same as an equal flatness tolerance. In a mating interface, however, depressions in the surface may be inconsequential. After all, only the surface's three highest points are likely to contact the mating face (assuming the mating face is perfectly flat). Here, we may want to focus the orientation control on only the three highest or *tangent points*, excluding all other points on the surface from the tolerance. We do this by adding the “tangent plane” symbol (a circled T) after the tolerance value in the feature control frame. See Fig. 5-94. Now, only the perfect plane constructed tangent to the surface's three highest points shall be contained within the tolerance zone. Since it's acceptable for lower surface points to lie outside the zone, there's no flatness control.

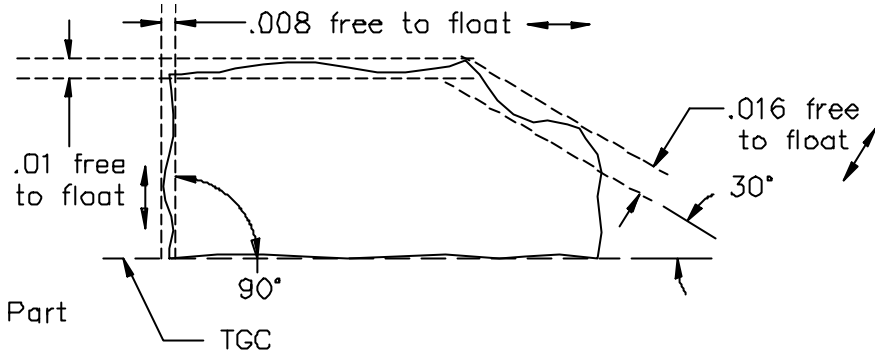


Figure 5-93 Tolerance zones for Fig. 5-92

The validity of “tangent plane” orientation control depends on the surface having exactly three noncolinear points that rise above the rest, allowing construction of exactly one tangent plane. Any other condition allows multiple candidate tangent planes to be constructed—a catastrophe not addressed by any standard. The method also assumes the mating face will be perfectly flat. If it too has three outstanding points, it’s unlikely that contact will occur in either surface’s tangent plane. Be careful with the “tangent plane” symbol.

For a width-type feature of size, Rule #1 automatically limits the parallelism of each surface to the other. Thus, a separate orientation tolerance meant to control parallelism between the two surfaces won’t have any effect unless it’s less than the total size tolerance.

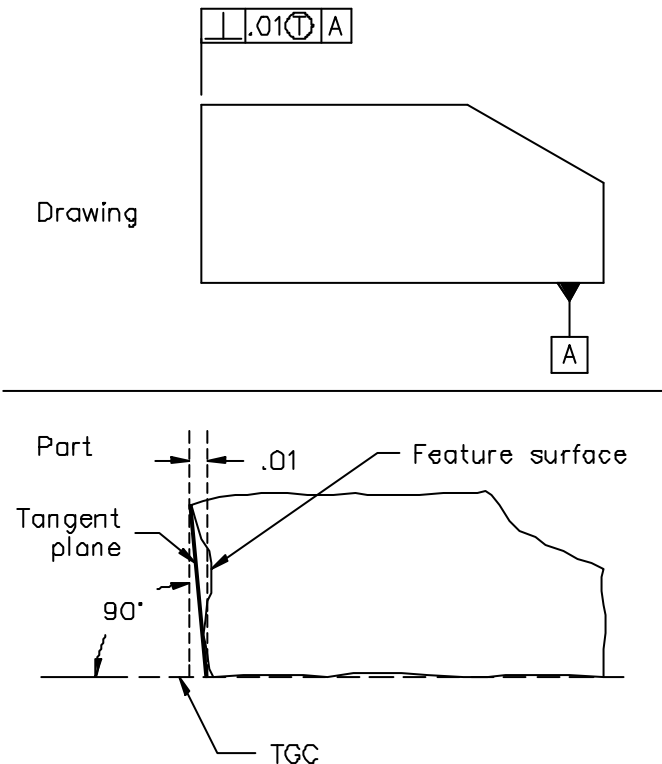


Figure 5-94 Application of tangent plane control

5.10.4 Applied to a Cylindrical or Width-Type Feature

Where an orientation tolerance feature control frame is placed according to options (a) or (d) in Table 5-1 (associated with a diameter or width dimension), the tolerance controls the orientation of the cylindrical or width-type feature. Where the tolerance is modified to MMC or LMC, it establishes a Level 3 virtual condition boundary as described in section 5.6.3.2 and Figs. 5-17(c) and 5-18(c). Alternatively, the “center method” described in section 5.6.5.1 may be applied to an orientation tolerance at MMC or LMC. Unmodified, the tolerance applies RFS and establishes a central tolerance zone as described in section 5.6.4.1, within which the feature’s axis or center plane shall be contained. See Fig. 5-95. Applied to a feature of size, the orientation tolerance provides no form control beyond Level 2.

Fig. 5-95 shows the center plane of a slot contained within a central parallel-plane tolerance zone (“center method”). Y14.5 also allows the orientation of an axis to be controlled within a parallel-plane tolerance zone. Since this would not prevent the axis from revolving like a compass needle between the two parallel planes, such an application usually accompanies a larger positional tolerance. In Fig. 5-96, a “diameter” symbol precedes the angulation tolerance value. Here, the central tolerance zone is bounded by a cylinder having a diameter equal to the tolerance value. This control is more like a positional tolerance, except the orientation zone is not basically located from the datums.

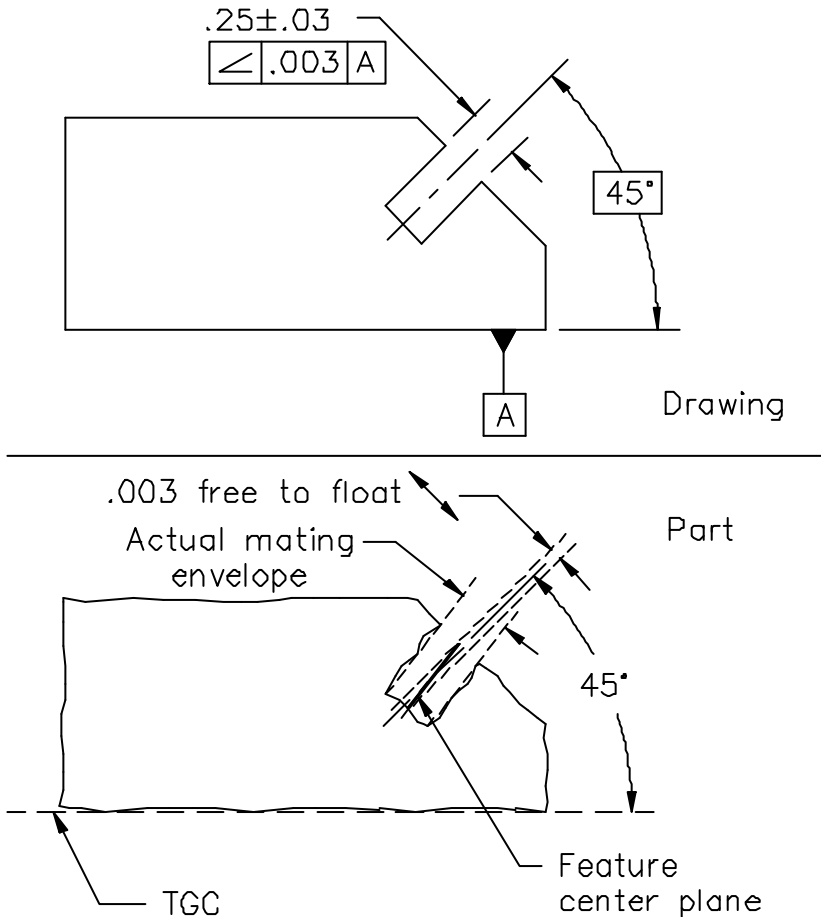


Figure 5-95 Applying an angularity tolerance to a width-type feature

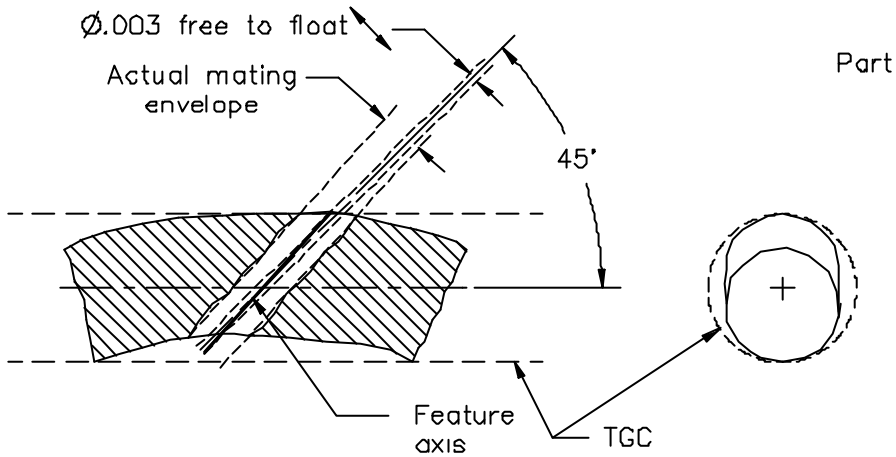
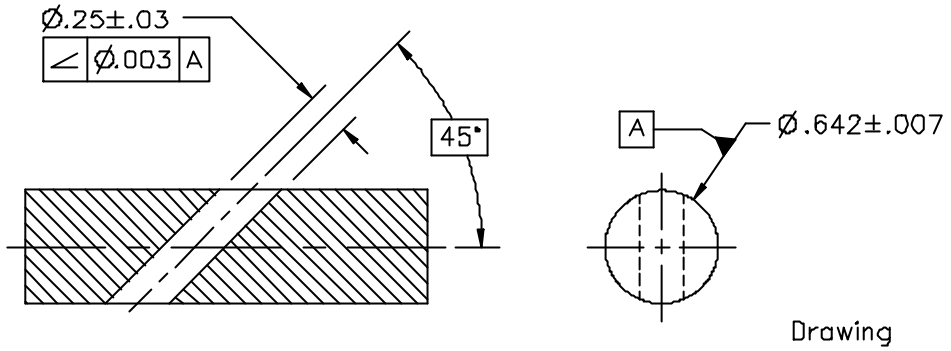


Figure 5-96 Applying an angularity tolerance to a cylindrical feature

A positional tolerance also controls orientation for a feature of size to the same degree as an equal orientation tolerance. Thus, for any feature of size, an orientation tolerance equal to or greater than its positional tolerance is meaningless. Conversely, where the designer needs to maximize positional tolerance while carefully protecting orientation, a generous positional tolerance can be teamed up with a more restrictive orientation tolerance.

5.10.4.1 Zero Orientation Tolerance at MMC or LMC

Where the only MMC design consideration is a clearance fit, there may be no reason for the feature’s MMC size limit to differ from its Level 3 virtual condition. In such a case, we recommend stretching the MMC size limit to equal the MMC virtual condition size and reducing the orientation tolerance to zero as described in section 5.6.3.4. In LMC applications, as well, a zero orientation tolerance should be considered.

5.10.5 Applied to Line Elements

Where a profiled surface performs a critical function, it’s sometimes necessary to control its orientation to a DRF. For the cam surface shown in Fig. 5-97, the 3-D control imposed by a parallel-planes tolerance zone is inappropriate because the surface isn’t supposed to be flat. Here, we want to focus the orientation

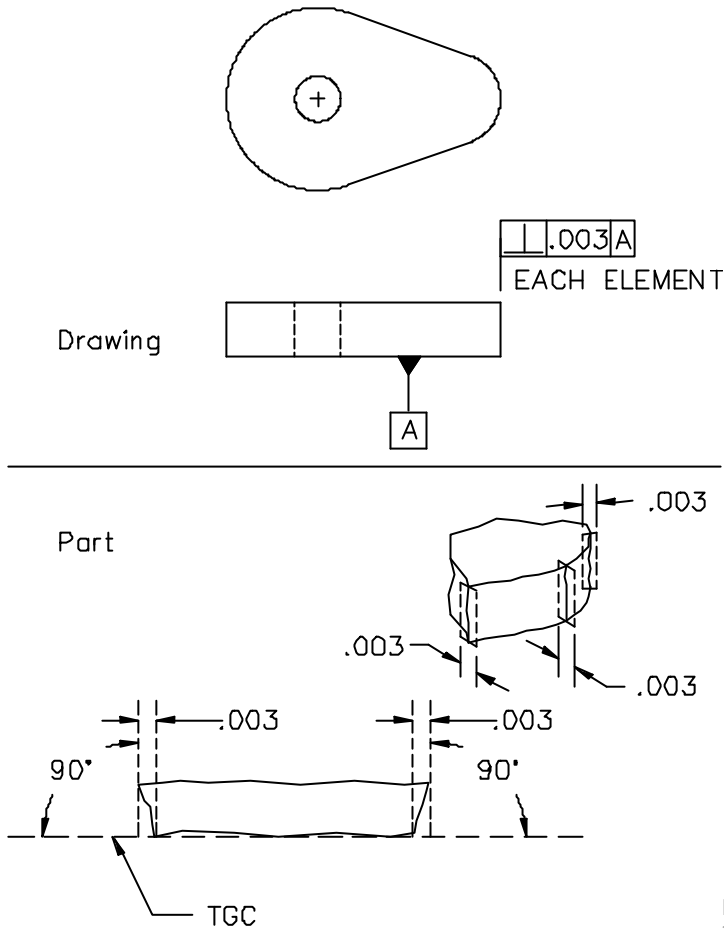


Figure 5-97 Controlling orientation of line elements of a surface

tolerance only on individual cross sections of the surface, one at a time. We do this by adding a note such as **EACH ELEMENT** or **EACH RADIAL ELEMENT** adjacent to the orientation feature control frame. This specifies a tolerance zone plane containing a tolerance zone bounded by two parallel lines separated by a distance equal to the tolerance value. As the tolerance zone plane sweeps the entire surface, the surface's intersection with the plane shall everywhere be contained within the tolerance zone (between the two lines). Within the plane, the tolerance zone's location may adjust continuously to the part surface while sweeping, but its orientation shall remain fixed at the basic angle relative to the DRF. This type of 2-D control allows unlimited surface undulation in only one direction.

Of a Surface Constructed About a Datum Axis—The note **EACH RADIAL ELEMENT** adjacent to the feature control frame means the tolerance zone plane shall sweep radially about a datum axis, always containing that axis. If the orienting (primary) datum doesn't provide an axis of revolution for the tolerance zone plane, a secondary datum axis shall be referenced. Note that within the rotating tolerance zone plane, the tolerance zone's location may adjust continuously.

Of a Profiled Surface—Where only a primary datum is referenced, as in Fig. 5-97, the tolerance zone plane shall sweep all around the part, always basically oriented to the datum, and always normal (perpendicular) to the controlled surface at each location. Where a secondary datum is referenced, the tolerance zone plane shall instead remain basically oriented to the complete DRF as it sweeps.

5.10.6 The 24 Cases

So far, in this section we've described the following:

- Four different types of orientation tolerance zone containments (“center method”)
 - Plane (feature surface, tangent, or center) between two parallel planes
 - Axis between two parallel planes
 - Axis within a cylinder
 - Line element between two parallel lines
- Two types of primary datums for orientation
 - Plane
 - Axis
- Three orientation tolerance symbols
 - Parallelism (0° or 180°)
 - Perpendicularity (90° or 270°)
 - Angularity (any other angle)

These components can be combined to create 24 ($4 \times 2 \times 3$) different fundamental applications (or “cases”) of orientation tolerance, illustrated in Fig. 5-98. In many cases, a secondary datum may be added for additional control. The illustrated parts are simplified abstracts, meant to show only the orientation control. On real parts, the orientation tolerances often accompany positional or profile tolerances.

5.10.7 Profile Tolerance for Orientation

As we'll see in Section 13, a single profile tolerance can control the size, form, orientation, and location of any feature, depending on the feature's type and the completeness of the referenced DRF. Where a profile tolerance already establishes the “size” and shape of a feature, incorporating orientation control may be as simple as adding another datum reference or expanding the feature control frame for composite profile control. Otherwise, it's better to use one of the dedicated orientation symbols.

5.10.8 When Do We Use an Orientation Tolerance?

Most drawings have a tolerance block or a general note that includes default plus and minus tolerances for angles. This default tolerance applies to any angle explicitly dimensioned without a tolerance. The angle between the depicted features shall be within the limits established by the angle dimension and the default angle tolerance. The default tolerance can be overridden by attaching a greater or lesser tolerance directly to an angle dimension. Either way, since neither feature establishes a datum for the other, the angular control between the features is reciprocal and balanced. The same level of control occurs where center lines and/or surfaces of part features are depicted on a drawing intersecting at right angles. Here, an implied 90° angle is understood to apply along with the default plus and minus angle tolerances. As before, there is no datum hierarchy, so all affected angular relationships are mutual.

The type of plus and minus angle tolerances just described does not establish a tolerance zone, wedge shaped or otherwise, to control the angulation of either feature. Be careful not to misinterpret Y14.5's Fig. 2-13, which shows a wedge-shaped zone controlling the location of a planar surface. Because it's still possible for the surface to be angled out of tolerance within the depicted zone, the “MEANS THIS” portion of the figure adds the note, **its angle shall not be less than $29^\circ 30'$ nor more than $30^\circ 30'$.**

Datum Axis

Parallel Planes

Cylindrical

Parallel Lines

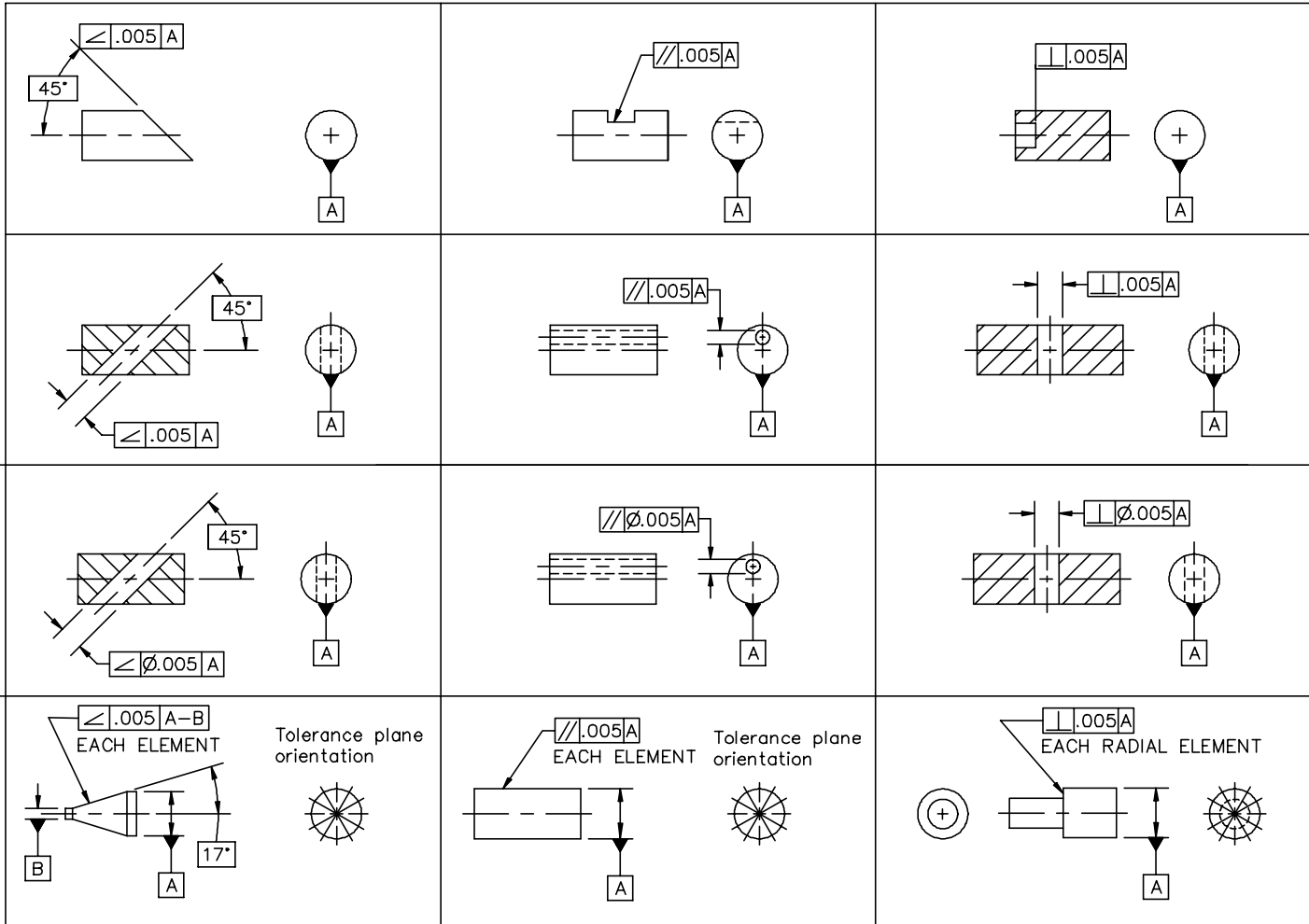


Figure 5-98 Applications of orientation tolerances

Datum Plane

<p>Diagram showing a 45-degree chamfered edge with a surface texture symbol $\sqrt{\text{.005}}\text{A}$ and datum A.</p>	<p>Diagram showing a flat surface with a surface texture symbol $\sqrt{\text{.005}}\text{A}$ and datum A.</p>	<p>Diagram showing a flat surface with a surface texture symbol $\sqrt{\text{.005}}\text{A}$ and datum A.</p>	Surf/Cen Pl/Tan Pl
<p>Diagram showing a 45-degree chamfered edge with a surface texture symbol $\sqrt{\text{.005}}\text{A}$ and datum A.</p>	<p>Diagram showing a cylindrical feature with a surface texture symbol $\sqrt{\text{.005}}\text{A}$ and datum A.</p>	<p>Diagram showing a cylindrical feature with a surface texture symbol $\sqrt{\text{.005}}\text{A}$ and datum A.</p>	Feature Axis
<p>Diagram showing a 45-degree chamfered edge with a surface texture symbol $\sqrt{\text{.005}}\text{A}$ and datum A.</p>	<p>Diagram showing a cylindrical feature with a surface texture symbol $\sqrt{\text{.005}}\text{A}$ and datum A.</p>	<p>Diagram showing a cylindrical feature with a surface texture symbol $\sqrt{\text{.005}}\text{A}$ and datum A.</p>	Feature Axis
<p>Diagram showing a chamfered edge with a surface texture symbol $\sqrt{\text{.005}}\text{A}$ and datum A.</p>	<p>Diagram showing a cylindrical feature with a surface texture symbol $\sqrt{\text{.005}}\text{AB}$ and datum A, B, and a tolerance plane orientation symbol.</p>	<p>Diagram showing a cylindrical feature with a surface texture symbol $\sqrt{\text{.005}}\text{AB}$ and datum A, B, and a tolerance plane orientation symbol.</p>	Surface Line Element

Figure 5-98 continued Applications of orientation tolerances

Now, let's consider a different case, illustrated in Fig. 5-99, where two planar features intersect at an angle controlled with plus and minus tolerances and location is not an issue. For the sake of discussion, we'll attach the "dimension origin" symbol to the extension line for one surface, ostensibly making it a "quasi-datum" feature and the other a "controlled" feature. We'll suppose the "controlled" feature shall be contained within some wedge-shaped tolerance zone. Without a rule for locating its vertex (a line), such a zone would be meaningless. For example, if we could locate the vertex a mile away from the part, the zone could easily contain the "controlled" feature, the whole part, and probably the whole building! Since the standards are mute on all this, let's be reasonable and suppose the vertex can be located anywhere in our supposed "datum plane," as we've shown in the lower portion of the figure.

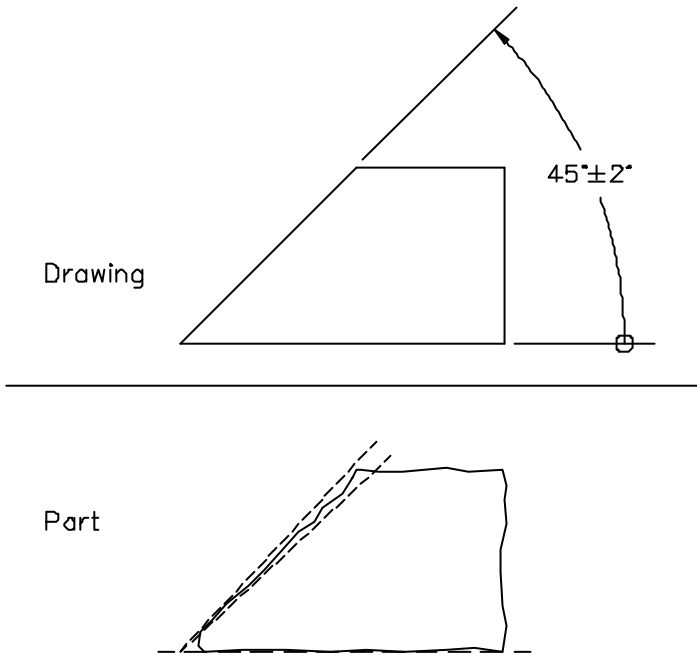


Figure 5-99 Erroneous wedge-shaped tolerance zone

Now here's the problem: Approaching the vertex, the width of our wedge-shaped tolerance zone approaches zero. Of course, even a razor edge has a minute radius. So we can assume that because of an edge radius, our "controlled" feature won't quite extend all the way to the vertex of the tolerance zone. But depending on the "size" of the radius and the angular tolerance, the zone could be only a few microns wide at the "controlled" feature's edge. Thus, the "controlled" feature's line elements parallel to the vertex shall be straight within those few microns, and angularity of the feature shall likewise approach perfection. Those restrictions are absurd.

Thus, even with a "dimension origin" symbol, a plus and minus angle tolerance establishes no defensible or usable tolerance zone for angulation. Instead, the tolerance applies to the angle measured between the two features. Imperfections in feature form complicate the measurement, and different alignments of the measuring scale yield different measurements. Unfortunately, the standards provide no guidance in either area. Despite these limitations, plus and minus angle tolerances are often sufficient for noncritical relationships where inspectors can be trusted to come up somehow with adequately repeatable and reproducible measurements.

Where a feature's orientation is more critical and the above methods are too ambiguous, an orientation tolerance feature control frame should be applied. In theory, datum simulation methods can accommodate out-of-squareness between datum features in a DRF. However, datum simulation will be more repeatable and error free where squareness of the secondary and tertiary datum features has been carefully and directly controlled to the higher-precedence datum(s).

As we'll see in the following sections, positional and profile tolerances automatically control feature orientation. But often, a generous positional or profile tolerance must be accompanied by a more strict orientation tolerance to assure functionality.

5.11 Positional Tolerance (Level 4 Control)

In the past, it was customary to control the location of a feature on a part by specifying for each direction a nominal dimension accompanied by plus and minus tolerances. In Fig. 5-100, the measured hole location shall be $1.625 \pm .005$ from the end of the shaft. Since the hole is drawn on the center line of the shaft, we know it must be well centered. But plus or minus how much? Let's assume the tolerance for centrality should match that for the 1.625 length. In effect, then, the axis of the hole shall lie within a $.010" \times .010"$ square box. Such a "square box" tolerance zone rarely represents the true functional requirements. Chapter 3 further elaborates on the shortcomings of plus and minus tolerances for location. The standards neither explain nor prohibit this method, but Y14.5 expresses a clear preference for its own brand of *positional tolerance* to control the orientation and location of one or more features of size, or in some cases, bounded features, relative to a DRF. A positional tolerance provides no form control beyond Level 2.

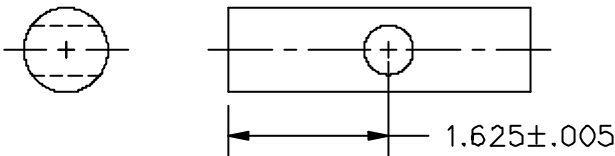


Figure 5-100 Controlling the location of a feature with a plus and minus tolerance

5.11.1 How Does It Work?

A positional tolerance may be specified in an RFS, MMC, or LMC context.

At MMC or LMC—Where modified to MMC or LMC, the tolerance establishes a Level 4 virtual condition boundary as described in section 5.6.3.3 and Figs. 5-17(d) and 5-18(d). Remember that the virtual condition boundary and the corresponding size limit boundary differ in size by an amount equal to the positional tolerance. In section 5.6.3.4, we discuss the advantages of unifying these boundaries by specifying a positional tolerance of zero. A designer should always consider this option, particularly in fastener applications.

At RFS—Unmodified, the tolerance applies RFS and establishes a central tolerance zone as described in section 5.6.4.1, within which the feature's center point, axis, or center plane shall be contained.

Alternative "Center Method" for MMC or LMC—Where the positional tolerance applies to a feature of size at MMC or LMC, the alternative "center method" described in section 5.6.5.1 may be applied.

For any feature of size, including cylindrical, spherical, and width-type features, a virtual condition boundary and/or derived center element is easily defined, and positional tolerancing is readily applicable.

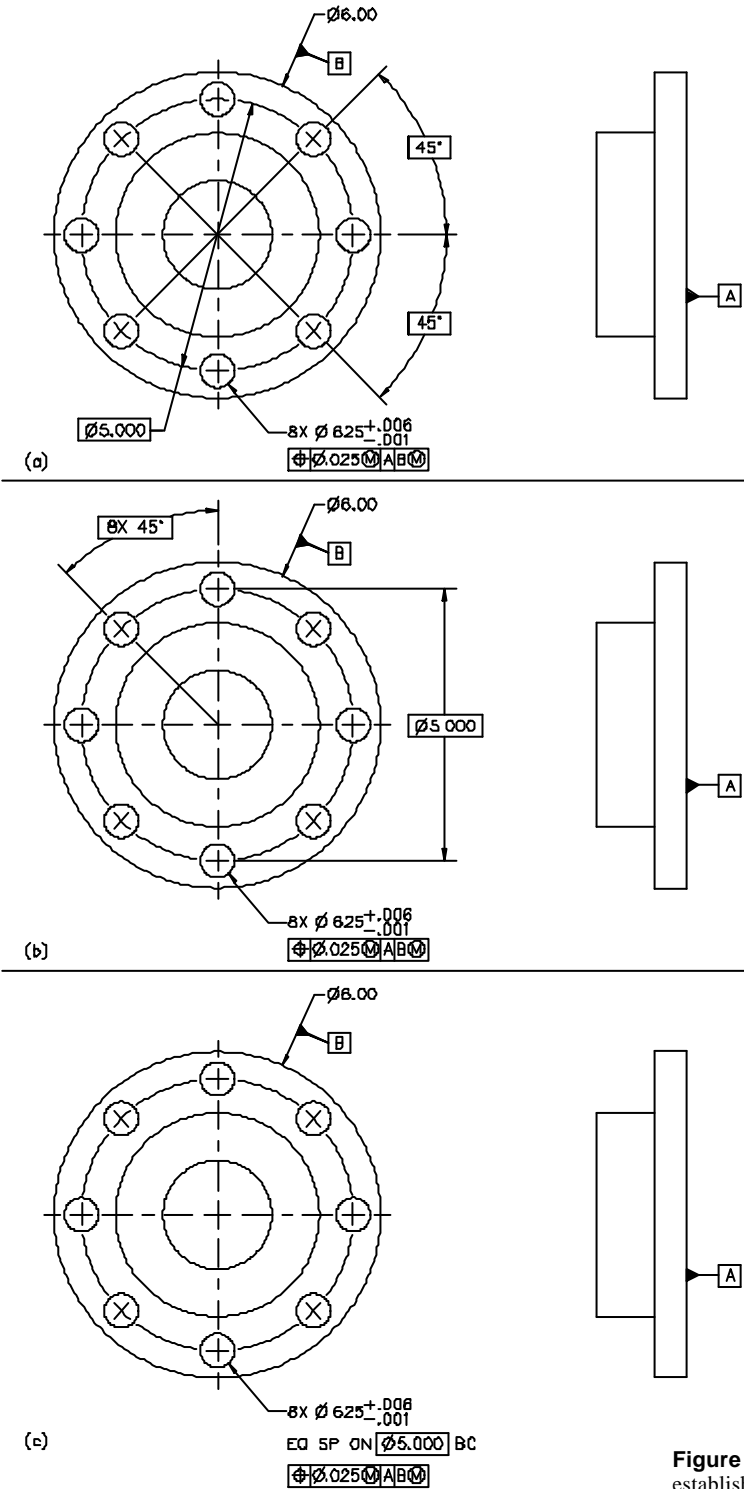


Figure 5-102 Alternative methods for establishing true positions using coordinate dimensioning

Base line dimensioning—For each of the two $\varnothing.125$ holes shown in Fig. 5-101, a basic dimension originates from each plane of the DRF. Manufacturers prefer this method because it directly provides them the coordinates for each true position relative to the datum origin. CMM inspection is simplified, using a single 0,0 origin for both holes.

Chain dimensioning—In Fig. 5-101, a basic dimension of 1.565 locates the upper $\varnothing.250$ hole directly from the center plane. However, the lower $\varnothing.250$ hole is located with a 3.000 basic dimension from the true position of the upper hole. People often confuse the 3.000 basic as originating from the actual axis of the upper hole, rather than from its true position. A manufacturer needing the coordinate of the lower hole will have to calculate it: $1.565 - 3.000 = -1.445$. Or is it -1.435 ?

Implied symmetry dimensioning—In many cases, the applicable basic dimensions are implied by drawing views. In Fig. 5-101, the true positions of the two $\varnothing.375$ holes have a single 2.000 basic dimension between them, but no dimension that relates either hole to the planes of the DRF. Since the holes appear symmetrical about the center plane of the DRF, that symmetrical basic relationship is implied.

Implied zero-basic dimensions—The view implies the relationship of the $\varnothing.500$ hole to the planes of the DRF as represented by the view’s center lines. Obviously, the hole’s basic orientation is 0° and its basic offset from center is 0. These implied zero-basic values need not be explicated.

Polar coordinate dimensioning—Rather than by “rectangular coordinates” corresponding to two perpendicular axes of the DRF, the true positions of the eight $\varnothing.625$ holes shown in Fig. 5-102(a) are defined by *polar coordinates* for angle and diameter. The $\varnothing 5.000$ “bolt circle” is basically centered at the intersection of the datum planes, and the two 45° basic angles originate from a plane of the DRF. Figs. 5-102(b) and (c) show alternative approaches that yield equivalent results, based on various methods and fundamental rules we’ve presented.

All the above methods are acceptable. Often, a designer can choose between base line and chain dimensioning. While both methods yield identical results, we prefer base line dimensioning even if the designer has to make some computations to express all the dimensions originating from the datum origin. Doing so once will preclude countless error-prone calculations down the road.

5.11.3 Datums for Positional Control

One of the chief advantages of a GD&T positional tolerance over plus and minus coordinate tolerances is its relationship to a specific DRF. Every positional tolerance shall reference one, two, or three datum features. The DRF need not restrain all six degrees of freedom, only those necessary to establish a unique orientation and location for true position. (Degrees of freedom are explained in section 5.9.7.) For example, the DRF established in Fig. 5-103 restrains only four degrees of freedom. The remaining two degrees, rotation about and translation along the datum axis, have no bearing on the controlled feature’s true position. Thus, further datum references are meaningless and confusing.

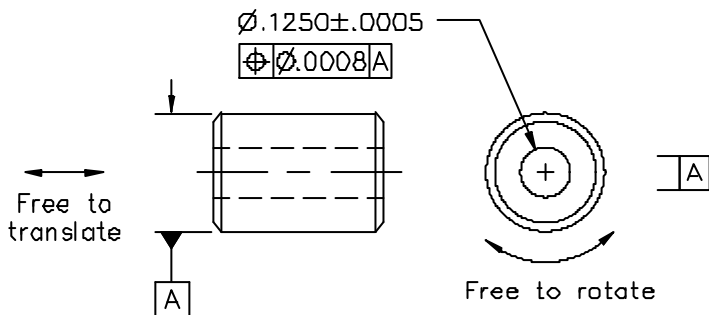


Figure 5-103 Restraining four degrees of freedom

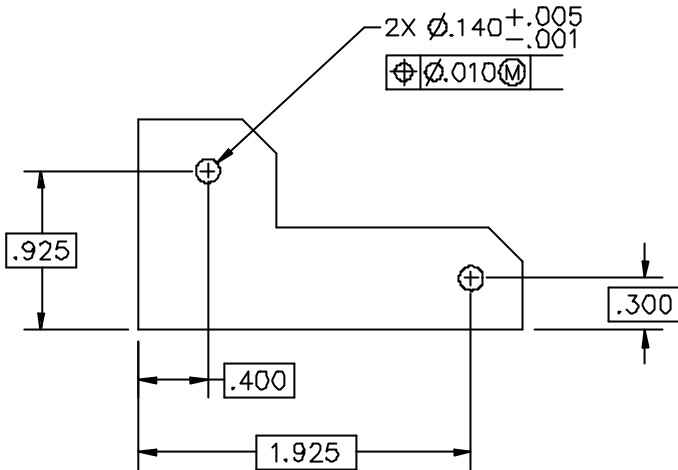


Figure 5-104 Implied datums are not allowed

For many positional tolerances, such as those shown in Fig. 5-104, the drawing view makes it quite obvious which part features are the origins, even if they weren't identified as datum features and referenced in the feature control frame. Before the 1982 revision of Y14.5, *implied datums* were recognized and not required to be explicitly referenced in such cases. In Fig. 5-104, although we all may agree the part's left and lower edges are clearly datum features, we might disagree on their precedence in establishing the orientation of the DRF. In another example, where a part has multiple coaxial diameters, it might be obvious to the designer, but very unclear to the reader, which diameter is supposed to be the datum feature. For these reasons, Y14.5 no longer allows implied datums; the savings in plotter ink aren't worth the confusion.

A datum feature of size can be referenced RFS (the default where no modifier symbol appears), at MMC, or at LMC. Section 5.6.7 discusses modifier choices. When MMC or LMC is selected, the DRF is not fixed to the part with a unique orientation and location. Instead, the DRF can achieve a variety of orientations and/or locations relative to the datum feature(s). The stimulating details of such allowable "DRF displacement" are bared in section 5.9.9.

5.11.4 Angled Features

Positional tolerancing is especially suited to angled features, such as those shown in Fig. 5-105. Notice how the true position for each angled feature is carefully defined with basic lengths and angles relative only to planes of the DRF. In contrast, Fig. 5-106 shows a common error: The designer provided a basic dimension to the point where the hole's true position axis intersects the surrounding face. Thus, the true position is established by a face that's not a datum feature. This is an example of an implied datum, which is no longer allowed.

5.11.5 Projected Tolerance Zone

A positional tolerance, by default, controls a feature over its entire length (or length and breadth). This presumes the feature has no functional interface beyond its own length and breadth. However, in Fig. 5-107, a pin is pressed into the controlled hole and expected to mate with another hole in a cover plate. The mating feature is not the pin hole itself, but rather the pin, which represents a projection of the hole. Likewise, the mating interface is not within the length of the pin hole, but above the hole, within the thickness of the cover plate.

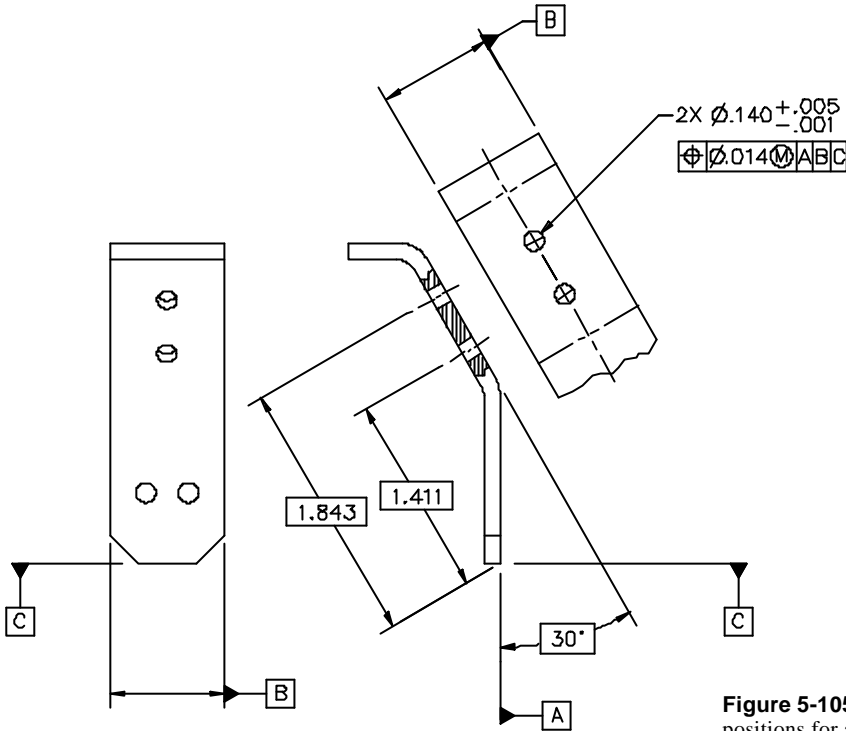


Figure 5-105 Establishing true positions for angled features—one correct method

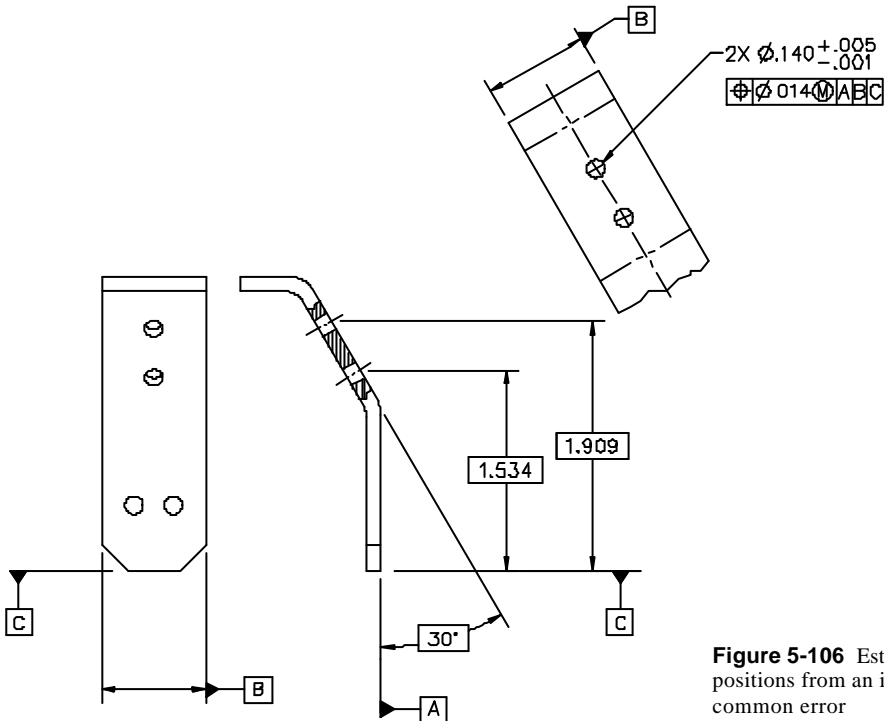


Figure 5-106 Establishing true positions from an implied datum—a common error

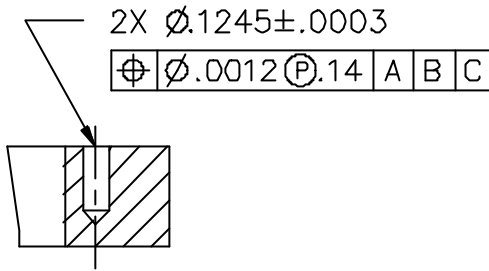


Figure 5-107 Specifying a projected tolerance zone

If the pin hole were perfectly perpendicular to the planar interface between the two parts, there would be no difference between the location of the hole and the pin. Any angulation, however, introduces a discrepancy in location. This discrepancy is proportional to the length of projection. Thus, directly controlling the location of the pin hole itself is inadequate to assure assemblability. Instead, we need to control the location of the hole's projection, which could be thought of as a phantom pin. This is accomplished with a positional tolerance modified with a projected tolerance zone.

A projected tolerance zone is specified by placing the “projected tolerance zone” symbol (a circled P) after the tolerance value in the position feature control frame. This establishes a constant-size central tolerance zone bounded either by two parallel planes separated by a distance equal to the specified tolerance, or by a cylinder having a diameter equal to the specified tolerance. For blind holes and other applications where the direction of projection is obvious, the length of projection may be specified after the symbol in the feature control frame. This means the projected tolerance zone terminates at the part face and at the specified distance from the part face (away from the part, and parallel to the true position axis or center plane). The projection length should equal the maximum extension of the mating interface. In our pin and cover plate example, the projection length must equal the cover plate's maximum thickness, .14. Where necessary, the extent and direction of the projected tolerance zone are shown in a drawing view as a dimensioned value with a heavy chain line drawn next to the center line of the feature, as in Fig. 5-108.

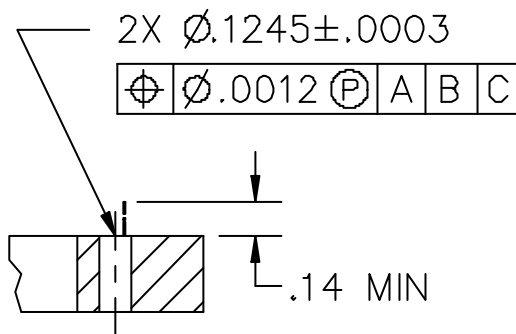


Figure 5-108 Showing extent and direction of projected tolerance zone

At RFS—The extended axis or center plane of the feature's actual mating envelope (as defined in section 5.6.4.2) shall be contained within the projected tolerance zone.

At MMC—The extended axis or center plane of the feature's applicable Level 2 MMC perfect form boundary (as defined in section 5.6.3.1) shall be contained within the projected tolerance zone. See Fig. 5-109. As the feature's size departs from MMC, the feature fits its MMC perfect form boundary more loosely. This permits greater deviation in the feature's orientation and/or location. A hole's departure from MMC permits assembly with a mating pin having its axis anywhere within a conical zone. The alternative

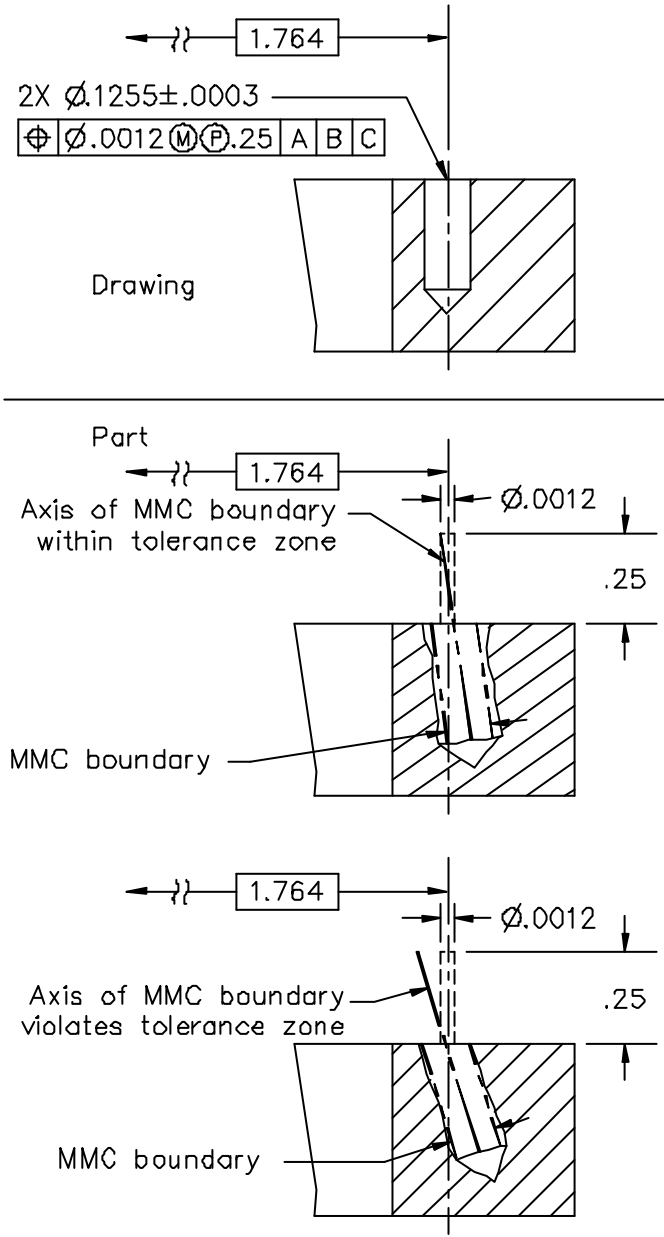


Figure 5-109 Projected tolerance zone at MMC

“center method” described in section 5.6.5.1 cannot be used for a projected tolerance zone. Its “bonus tolerance” would simply enlarge the projected tolerance zone uniformly along its projected length, failing to emulate the feature’s true functional potential.

At LMC—The extended axis or center plane of the feature’s Level 2 LMC perfect form boundary (as defined in section 5.6.3.1) shall be contained within the projected tolerance zone. As the feature’s size departs from LMC, the feature fits its LMC perfect form boundary more loosely. This permits greater deviation in the feature’s orientation and/or location. The alternative “center method” described in section 5.6.5.1 cannot be used for a projected tolerance zone.

5.11.6 Special-Shaped Zones/Boundaries

We stated that a “square box” tolerance zone rarely represents a feature’s true functional requirements, and that the shape of a positional tolerance zone usually corresponds to the shape of the controlled feature. There are exceptions, however, and GD&T has been made flexible enough to accommodate them.

5.11.6.1 Tapered Zone/Boundary

Where a relatively long or broad feature of size has different location requirements at opposite extremities, a separate positional tolerance can be specified for each extremity. This permits maximization of both tolerances. “Extremities” are defined by nominal dimensions. Thus, for the blind hole shown in Fig. 5-110, the $\varnothing.010$ tolerance applies at the intersection of the hole’s true position axis with the surrounding part face (Surface C). The $\varnothing.020$ tolerance applies .750 (interpreted as basic) below that.

At MMC or LMC—The tolerances together establish a Level 4 virtual condition boundary as described in section 5.6.3.3 and Figs. 5-17(d) and 5-18(d), except that in this case, the boundary is a frustum (a cone or wedge with the pointy end chopped off). The virtual condition size at each end derives from the regular applicable formula and applies at the defined extremity.

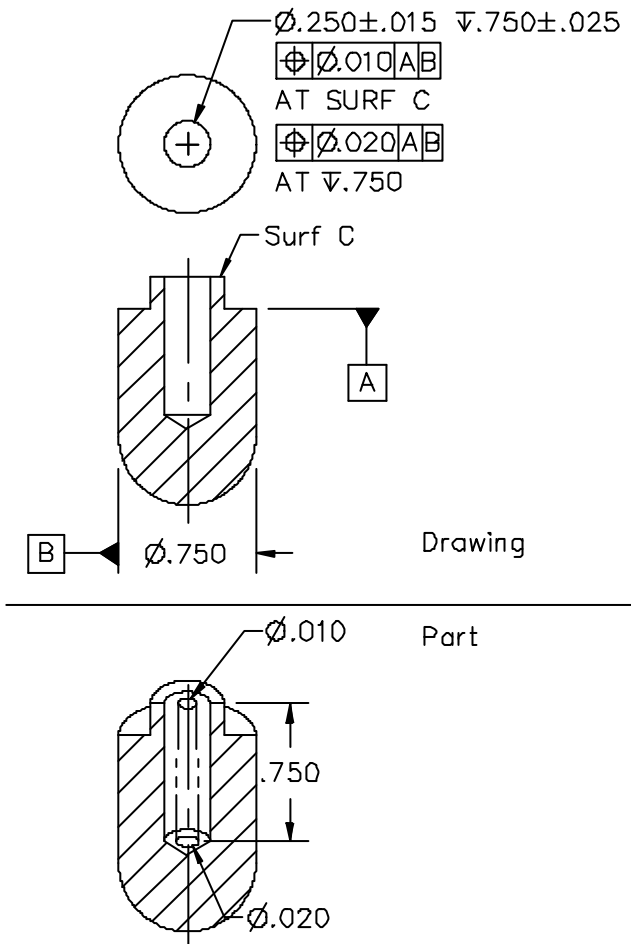


Figure 5-110 Different positional tolerances (RFS) at opposite extremities

At RFS—Unmodified, the tolerances apply RFS and establish a central tolerance zone bounded by a conical or wedge-shaped frustum, within which the feature’s axis or center plane shall be contained. The specified tolerance zone sizes apply at the defined extremities. See Fig. 5-110.

Alternative “Center Method” for MMC or LMC—Where modified to MMC or LMC, the tolerances may optionally be interpreted as in an RFS context—that is, they establish a central tolerance zone bounded by a conical or wedge-shaped frustum, within which the feature’s axis or center plane shall be contained. However, unlike in the RFS context, the size of the MMC or LMC tolerance zone shall be enlarged at each defined extremity by a single “bonus tolerance” value, derived according to section 5.6.5.1.

5.11.6.2 Bidirectional Tolerancing

A few features have different positional requirements relative to different planes of the DRF. Where these differences are slight, or where even the lesser tolerance is fairly generous, the more restrictive value can be used in an ordinary positional tolerance. In most cases, the manufacturing process will vary nearly equally in all directions, so an extra .001" of tolerance in just one direction isn’t much help. However, where the difference is significant, a separate feature control frame can be specified for each direction. Y14.5 calls this practice *bidirectional tolerancing*. It can be used with a cylindrical feature of size located with two coordinates, or with a spherical feature of size located with three coordinates.

Each bidirectional feature control frame may be evaluated separately, just as if each controls a separate feature of size. However, as with separate features, rules for simultaneous or separate requirements apply (see section 5.9.10). By convention, the “diameter” symbol (∅) is not used in any bidirectional feature control frames. The exact meanings of bidirectional tolerances are deceptively complex. They depend on whether true position is defined in a rectangular or polar coordinate system, and on whether the tolerances apply in an RFS, MMC, or LMC context.

In a Rectangular Coordinate System—Fig. 5-111 shows a coupling ball located with rectangular coordinates in three axes. Each of the three separate feature control frames constrains the ball’s location

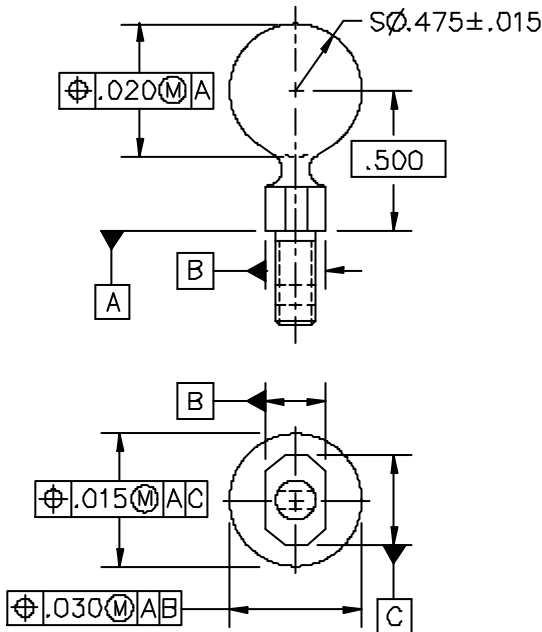


Figure 5-111 Bidirectional positional tolerancing, rectangular coordinate system

relative only to the DRF plane that is perpendicular to the dimension line. The .020 tolerance, for example, applies only to the .500 BASIC coordinate, relative to the horizontal plane of the DRF.

At MMC or LMC (Rectangular)—Each positional tolerance establishes a tolerance plane perpendicular to its dimension line. Each tolerance plane contains the center point (or axis, for a cylinder) of a Level 4 virtual condition boundary as described in section 5.6.3.3. However, within this plane, the location (and for a cylinder, orientation) of the boundary center is unconstrained. Thus, by itself, each tolerance would permit the controlled feature to spin and drift wildly within its tolerance plane. But, the combined restraints of three (or two, for a cylinder) perpendicular tolerance planes are usually adequate to control the feature’s total location (and orientation, for a cylinder).

The virtual condition boundaries for a shaft at MMC are external to the shaft. As each cylindrical boundary spins and drifts within its tolerance plane, it generates an effective boundary of two parallel planes. The intersection of these parallel-plane boundaries is a fixed size rectangular box at true position. See Fig. 5-112. Thus, a single functional gage having a fixed rectangular cutout can gauge both bidirectional positional tolerances in a single pass. The same is not true where the virtual condition boundaries are internal to a hole at MMC, since a hole cannot contain parallel-plane boundaries.

At RFS (Rectangular)—Unmodified, each positional tolerance applies RFS and specifies a central tolerance zone bounded by two parallel planes separated by a distance equal to the specified tolerance. The intersection of these parallel-plane tolerance zones is a rectangular box centered at true position, within which the feature’s axis or center point shall be contained. See Fig. 5-113.

Alternative “Center Method” for MMC or LMC (Rectangular)—Where modified to MMC or LMC, both tolerances may optionally be interpreted as in an RFS context—that is, each establishes a central

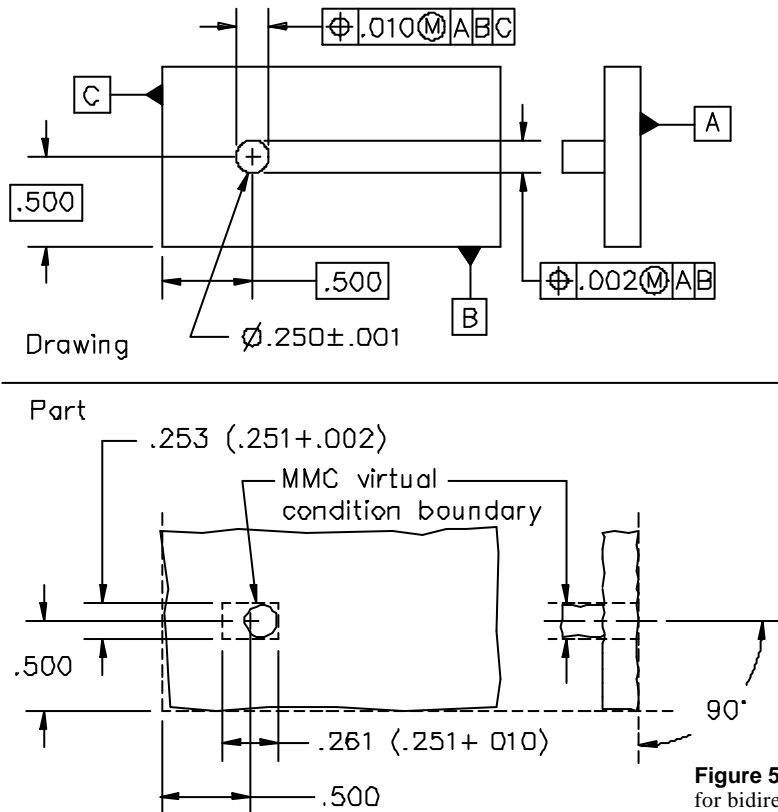


Figure 5-112 Virtual condition boundaries for bidirectional positional tolerancing at MMC, rectangular coordinate system

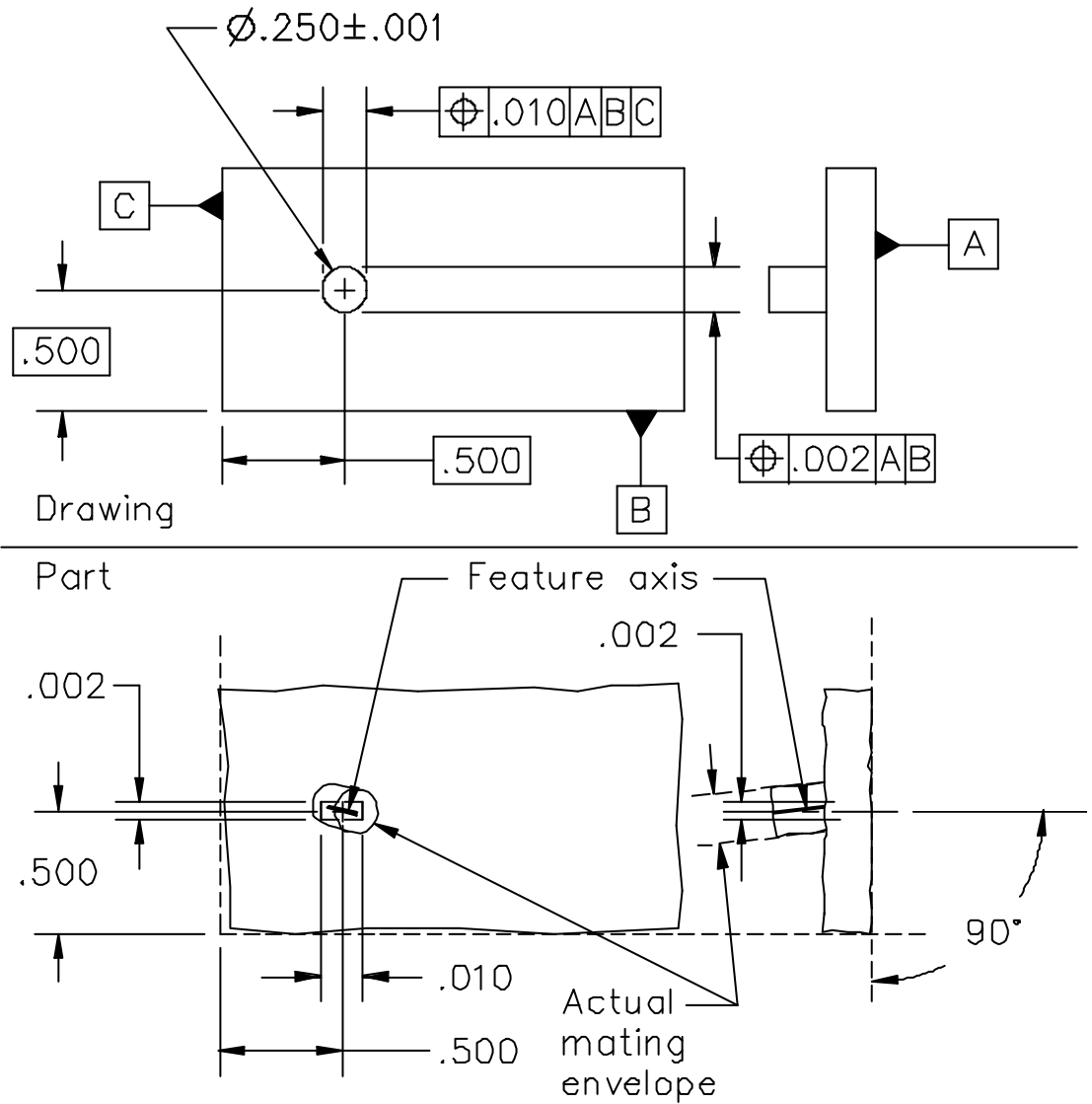
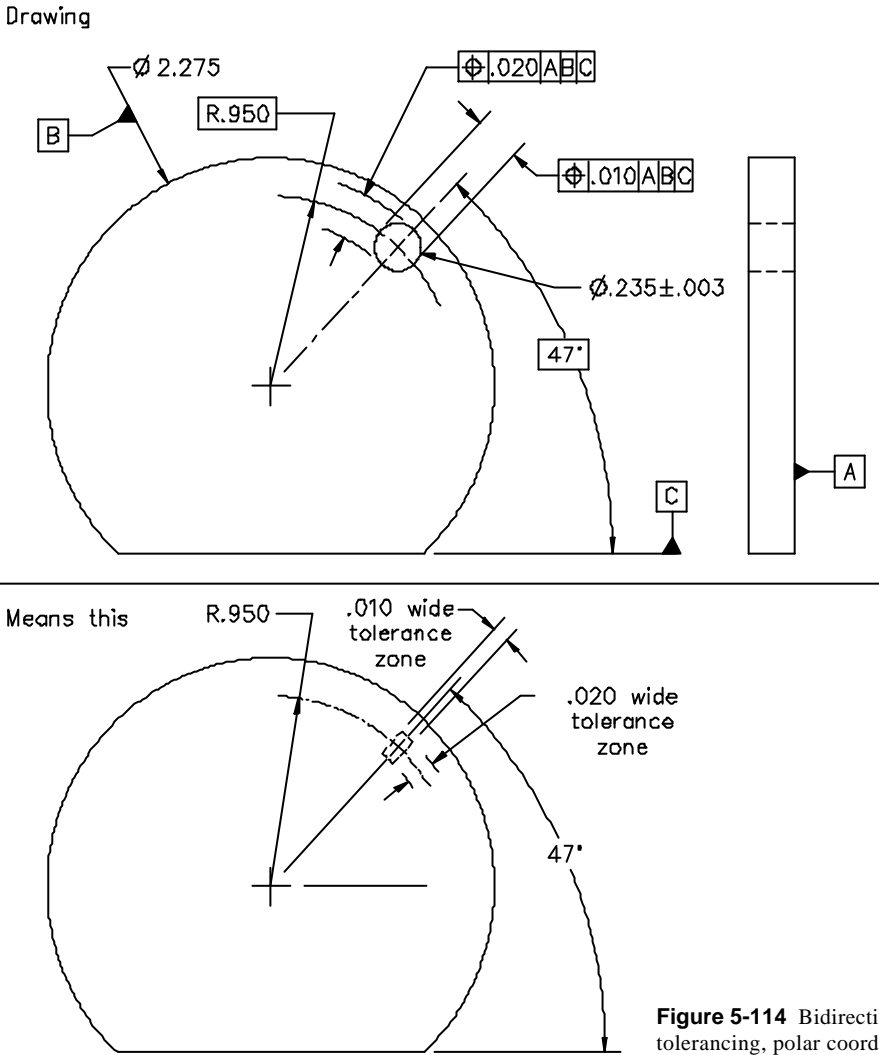


Figure 5-113 Tolerance zone for bidirectional positional tolerancing applied RFS, rectangular coordinate system

tolerance zone bounded by a pair of parallel planes, within which the feature’s axis or center point shall be contained. However, unlike in the RFS context, the size of each MMC or LMC tolerance zone shall be enlarged by a single “bonus tolerance” value, derived according to section 5.6.5.1.

In a Polar Coordinate System—Fig. 5-114 shows a hole located with polar coordinates, one for radius and one for angle. The .020 tolerance constrains the hole’s location relative only to the R.950 basic coordinate—in effect, its radial distance from the DRF origin point. The .010 tolerance constrains the hole relative only to a center plane rotated 47° basic relative to the DRF plane.

At MMC or LMC (Polar)—In this type of application, no virtual condition boundary is defined, due to problems in defining its restraint. The “center method,” described on the next page, shall be used instead.



At RFS (Polar)—Unmodified, each positional tolerance applies RFS. One tolerance specifies a central tolerance zone bounded by two parallel planes separated by a distance equal to the specified tolerance. The other tolerance specifies a tolerance zone bounded by two concentric cylinders radially separated by a distance equal to the specified tolerance. The intersection of these tolerance zones is an arc-shaped space (shown in the lower portion of Fig. 5-114) centered at true position, within which the feature’s axis or center point shall be contained.

“Center Method” for MMC or LMC (Polar)—Where modified to MMC or LMC, both tolerances shall be interpreted as in an RFS context—that is, each establishes a central tolerance zone bounded by a pair of parallel planes and a pair of concentric cylinders, within which the feature’s axis or center point shall be contained. However, unlike in the RFS context, the size of each MMC or LMC tolerance zone shall be enlarged by a single “bonus tolerance” value, derived according to section 5.6.5.1.

5.11.6.3 Bounded Features

Positional tolerance can be applied judiciously to bounded features having opposing elements that partly or completely enclose a space.

At MMC or LMC—If the positional tolerance is modified to MMC, the bounded feature shall have a defined and discernible MMC size/form boundary. This can derive from multiple size dimensions or profile tolerance(s) (see Section 13). In an LMC context, an LMC size/form boundary shall be defined. The tolerance establishes a Level 4 virtual condition boundary uniformly offset from the applicable MMC or LMC size/form limit boundary by an amount equal to one-half the specified positional tolerance. For clarification, the term **BOUNDARY** is placed beneath the feature control frames.

At RFS—RFS is not applicable unless the designer specifies a detailed procedure for deriving unique and repeatable center elements. Then, the tolerance establishes one or more central tolerance zones within which the derived center element(s) shall be contained.

Fig. 5-115 shows a bounded feature controlled with two different positional tolerances. In this example, the concept is identical to that for bidirectional tolerancing described in section 5.11.6.2, except the controlled feature is noncircular with a separate size dimension corresponding to each positional tolerance. Where bidirectional control is not necessary, we recommend using instead composite profile tolerancing, as detailed in section 5.13.13.

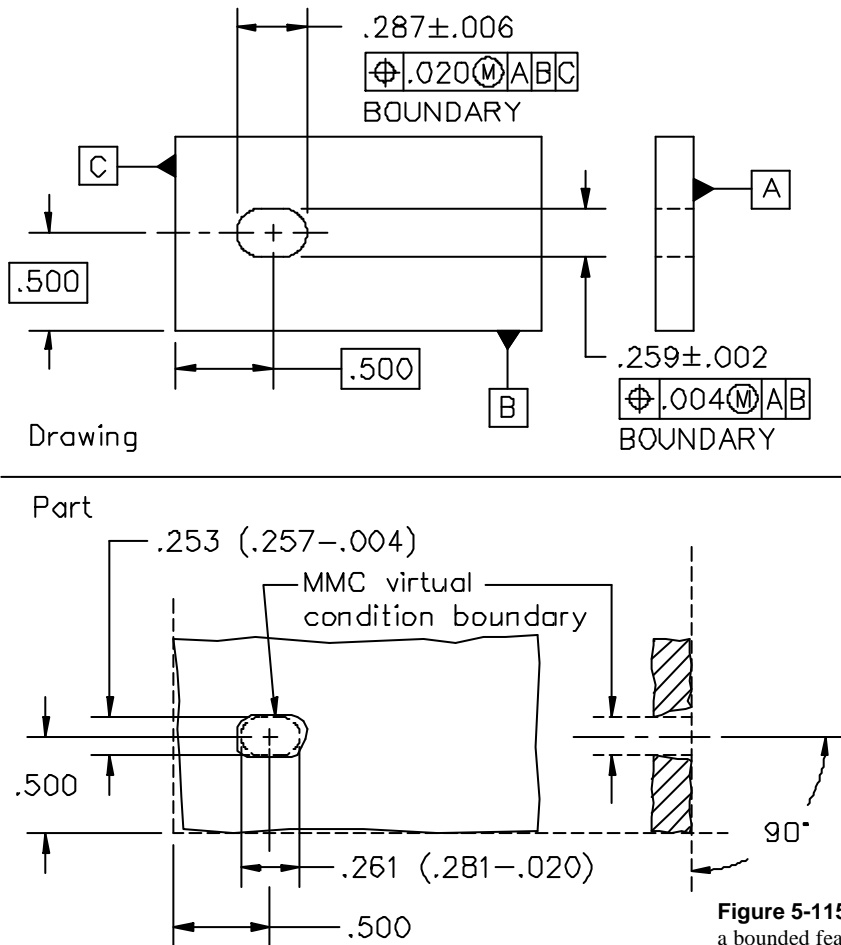


Figure 5-115 Positional tolerancing of a bounded feature

5.11.7 Patterns of Features

In many assemblies, two parts are attached to each other through a pattern of (multiple) features of size. For example, a closure cover may be bolted to a pump body with 24 3/8" bolts. A positional tolerance may be applied to the entire pattern, controlling the orientation and location of each individual feature relative to a DRF, and relative to every other feature in the pattern. Rather than a single boundary or tolerance zone, a positional tolerance applied to a feature pattern establishes a pattern (*framework*) of multiple boundaries or tolerance zones. Within this framework, the orientation and location of all the boundaries (or zones) are fixed relative to one another according to the basic dimensions expressed on the drawing.

At MMC or LMC—Where modified to MMC or LMC, the tolerance establishes a framework of Level 4 virtual condition boundaries as described in section 5.6.3.3.

At RFS—Unmodified, the tolerance applies RFS and establishes a framework of central tolerance zones as described in section 5.6.4.1.

Alternative “Center Method” for MMC or LMC—Where the positional tolerance applies to features of size at MMC or LMC, the alternative “center method” described in section 5.6.5.1 may be applied. The size of each tolerance zone adjusts independently according to the actual size of its corresponding feature.

In the following discussion, we’re going to focus on cylindrical mating features and their Level 4 MMC virtual condition boundaries. However, pattern controls are equally effective for width-type features, and just as usable in LMC and RFS contexts. The few simplified calculations we’ll be making are just to illustrate the concepts of pattern control. Subsequent chapters, particularly 22 and 24, present a more thorough discussion of positional tolerance calculations.

5.11.7.1 Single-Segment Feature Control Frame

The handle shown in Fig. 5-116 is for lifting an avionics “black box” out of a plane. It will be attached to a die-cast aluminum box using six 8-32 machine screws into blind tapped holes. The handle is a standard catalog item, chosen partly for its ready availability and low cost. Had it been a custom design, we might have specified tighter tolerances for the mounting holes. Nevertheless, through careful use of GD&T, we can still specify a pattern of tapped holes that will always allow hassle-free mounting of any sample handle.

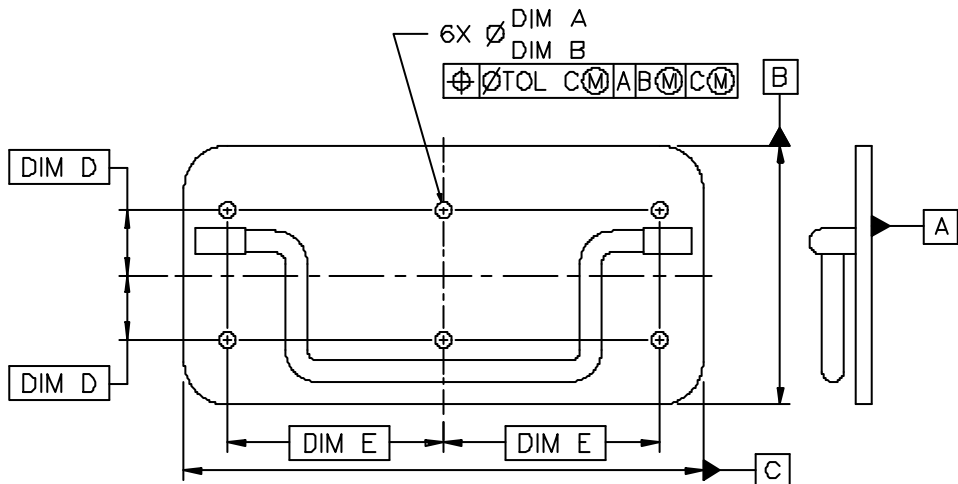


Figure 5-116 Standard catalog handle

For ease of assembly, we primarily need to assure a clearance fit between each of the handle's holes and the major diameter of its corresponding 8-32 screw. Worst-case assemblability is therefore represented by the MMC virtual conditions of the holes and the MMC virtual conditions of the screws. The handle's Technical Bulletin (Fig. 5-117) tells us the mounting holes can be as small as $\varnothing.186$. At that MMC size, a hole's positional deviation can be as much as $\varnothing.014$ (likely a conversion from $\pm.005$ coordinate tolerances). According to the formula in section 5.6.3.1, the MMC virtual condition for each hole (internal feature) is $\varnothing.186 - \varnothing.014 = \varnothing.172$.

Dash #	Screw Size	DIM A	DIM B	TOL C	DIM D	DIM E
-1	#4					
-2	#6					
-3	#8	.191	.186	.014	750	2.500
-4	#10					

Figure 5-117 Handle Technical Bulletin

To assure a clearance fit, then, we must establish for each screw a Level 4 virtual condition boundary no larger than $\varnothing.172$. While we can't apply a positional tolerance directly to the screws, we can apply a tolerance to the pattern of tapped holes. The most difficult assembly would result from a screw with its pitch diameter at MMC and its major diameter at MMC ($\varnothing.1640$), torqued into a tapped hole that's also at MMC. Functionally, this is only slightly more forgiving than a simple $\varnothing.164$ boss. For our tapped holes, then, if we model our virtual conditions on a substitute $\varnothing.164$ boss, our tolerances will be slightly conservative, which is fine.

For a $\varnothing.164$ boss, the maximum allowable positional tolerance is found by simply reversing our virtual condition formula—that is, by starting with the desired MMC virtual condition size and subtracting the feature's MMC size: $\varnothing.172 - \varnothing.164 = \varnothing.008$. In Fig. 5-118, we've specified a single positional tolerance of $\varnothing.008$ for the entire pattern of six tapped holes. The tolerance controls the location of each hole to the DRF A|B|C, and at the same time, the spacings between holes. Assemblability is assured. Problem solved.

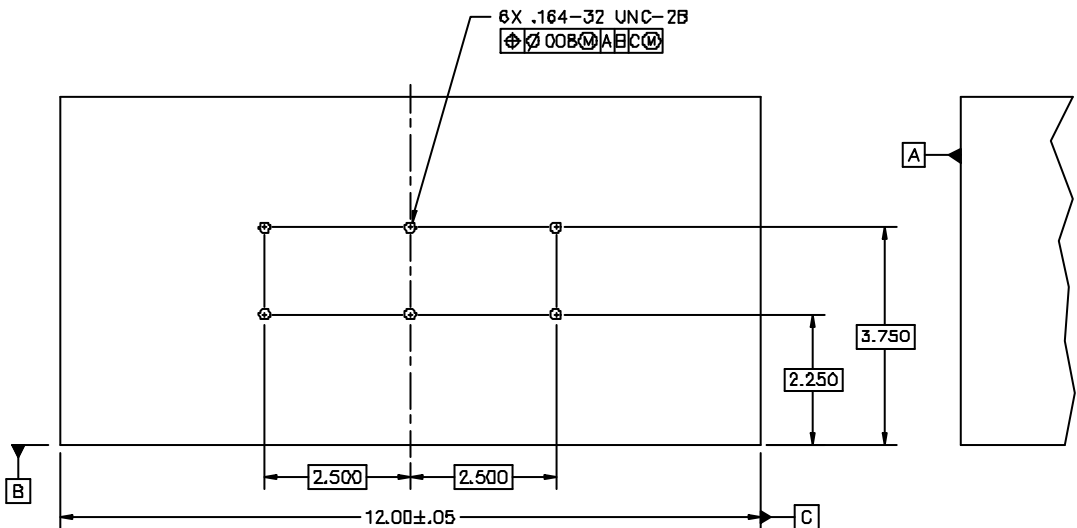


Figure 5-118 Avionics “black box” with single positional tolerance on pattern of holes

“Problem solved,” that is, until we discover that about half the boxes made have one or more tapped holes exceeding their $\varnothing.008$ positional tolerance. On closer analysis, we find the same problem on every rejected box: Though the hole-to-hole spacings are excellent and handles can assemble easily, the entire pattern of holes is shifted relative to the datum C width. We often find that processes can make hole-to-hole spacings more precise than the overall location of the pattern. Fortunately, most designs can afford a significantly greater tolerance for overall location. In our example, $\varnothing.008$ is necessary for the hole-to-hole spacings, but we could actually allow the entire pattern (the handle itself) to shift around on the box $1/8$ " or so in any direction.

5.11.7.2 Composite Feature Control Frame

In Fig. 5-119, we’ve applied a *composite positional tolerance* feature control frame to our pattern of tapped holes. As does the more common single-segment frame already described, the composite frame has a single “position” symbol. Unlike the single-segment frame, the composite frame has two segments, upper and lower, each establishing a distinct framework of virtual condition boundaries or central tolerance zones. Notice the difference in tolerance values and datum references between the two segments. The intent of a composite feature control frame is for the upper segment to provide a complete overall location control, then for the lower segment to provide a specialized refinement within the constraints of the upper segment. Here’s how it works.

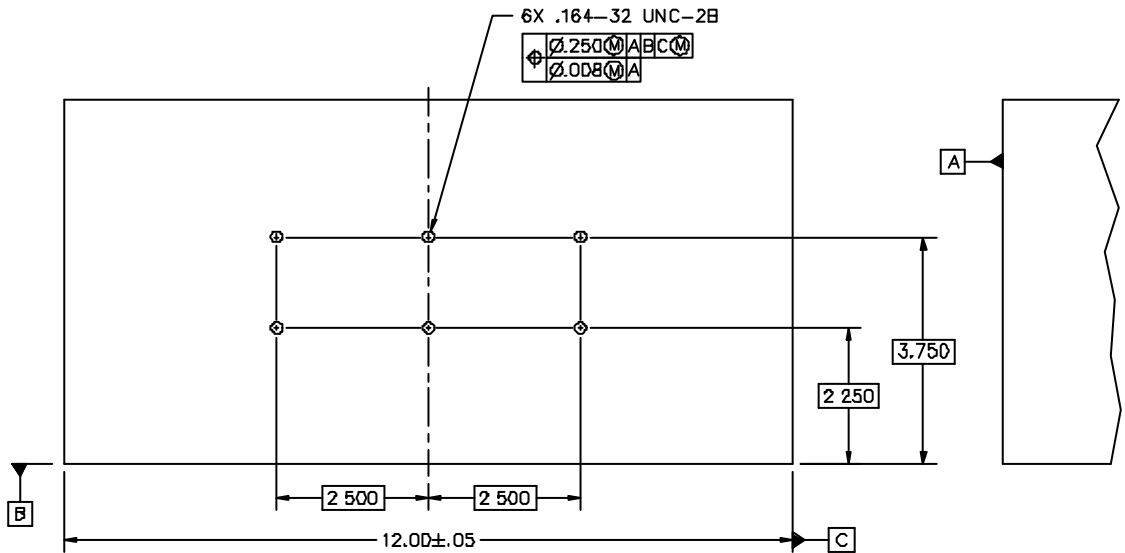
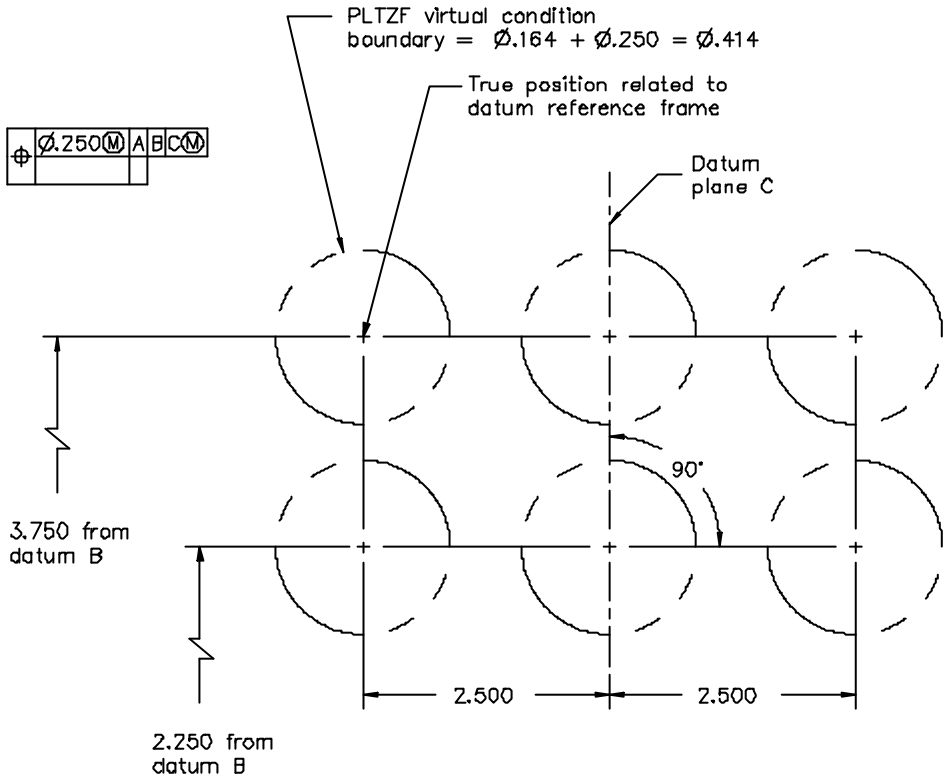


Figure 5-119 Avionics “black box” with composite positional tolerance on pattern of holes

The *upper segment* means the same as a single-segment positional tolerance feature control frame. In our Fig. 5-119 example, positional tolerance of $\varnothing.250$ is permitted for each hole, relative to the DRF A|B|C. This establishes a *Pattern Locating Tolerance Zone Framework (PLTZF)* (pronounced “Plahtz”) comprising six virtual condition boundaries for the holes, all basically parallel and basically located to each other. In addition, the orientation and location of the entire PLTZF is restrained relative to the referenced DRF A|B|C. In this case, the tapped holes would have negative virtual conditions. Fig. 5-120 shows instead the PLTZF virtual condition boundaries for our substitute $\varnothing.164$ bosses.



Note All boundaries are oriented perpendicular to primary datum A.

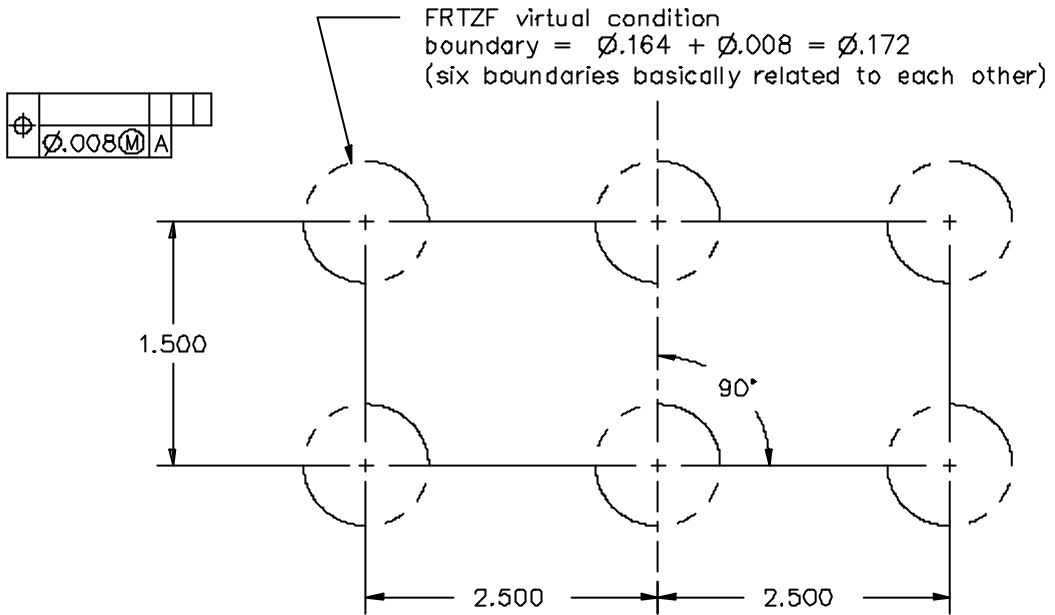
Figure 5-120 PLTZF virtual condition boundaries for Fig. 5-119

Compared with the single-segment positional tolerance of Fig. 5-118, the upper segment tolerance in our example affords much more freedom for the overall location of the handle on the box. However, $\varnothing.250$ allows too much feature-to-feature variation to assure assemblability. That’s where the lower segment kicks in.

The lower segment establishes the *Feature Relating Tolerance Zone Framework (FRTZF)* (pronounced “Fritz”). This segment may have zero, one, two, or three datum references. Where datums are referenced, they restrain only the orientation of the FRTZF, never its location. Fig. 5-121 shows the FRTZF virtual condition boundaries for our substitute bosses at work. Notice that datum A restrains the orientation of the FRTZF. This is crucial to the handle’s fitting flush. However, datum A couldn’t possibly restrain the location of the FRTZF, since the holes are perpendicular to datum A. In our example, then, the rule against location restraint is moot. In a moment, we’ll show how the difference can become relevant.

Compared with the single-segment positional tolerance of Fig. 5-118, the lower segment tolerance in our example has the same tolerance value, and affords exactly the same feature-to-feature control. However, the lower segment’s entire FRTZF is able to translate freely relative to the DRF, affording no restraint at all for the overall location of the handle on the box.

To summarize, we’ve solved our handle mounting problem with a composite positional tolerance that’s really two tolerances in one: a larger tolerance to control the overall location of the handle on the box; and a smaller tolerance to control the orientation (perpendicularity) of the holes to the mounting face, as well as the hole-to-hole spacings. Assemblability is assured. Problem solved.



Note: All boundaries are oriented perpendicular to primary datum A.

Figure 5-121 FRTZF virtual condition boundaries for Fig. 5-119

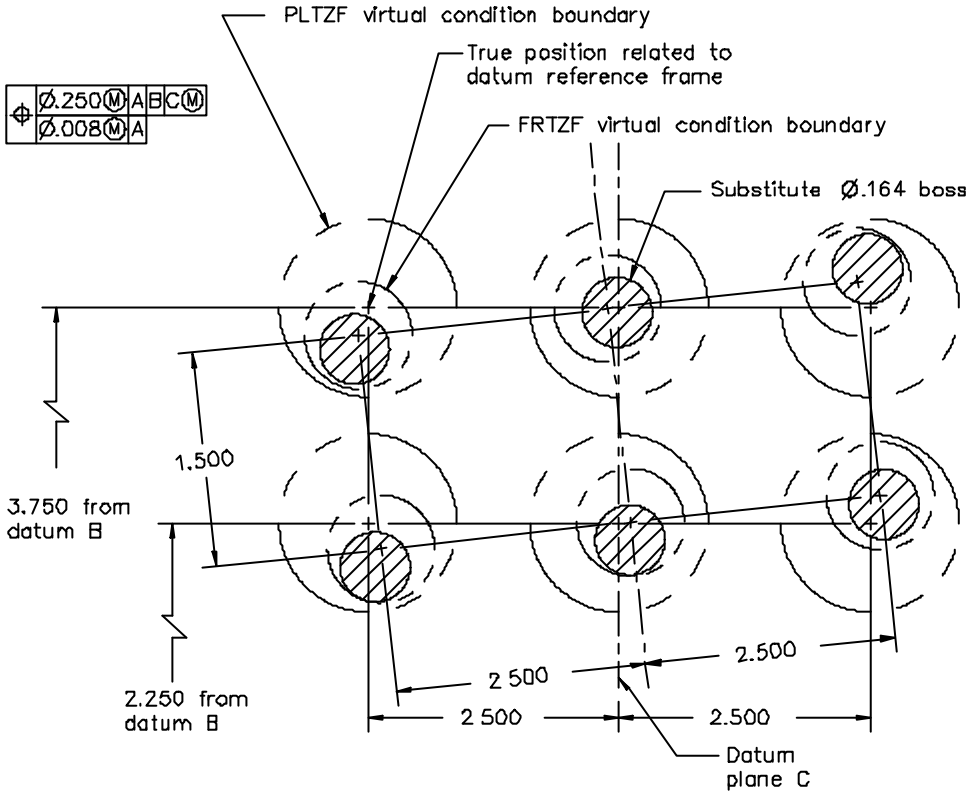
With a Secondary Datum in the Lower Segment—With composite control, there's no explicit congruence requirement between the PLTZF and the FRTZF. But, if features are to conform to both tolerances, the FRTZF will have to drift to where its virtual condition boundaries (or central tolerance zones) have enough overlap with those of the PLTZF. Fig. 5-122 shows for our example one possible valid relationship between the PLTZF and FRTZF. Again, the virtual condition boundaries are based on a substitute $\varnothing.164$ boss. Notice that the PLTZF virtual conditions are so large, they allow considerable rotation of the pattern of tapped holes. The FRTZF offers no restraint at all of the pattern relative to datums B or C. This could allow a handle to be visibly crooked on the box.

In Fig. 5-123, we've corrected this limitation by simply referencing datum B as a secondary datum in the lower segment. Now, the orientation (rotation) of the FRTZF is restrained normal to the datum B plane. Although datum B could also restrain the basic location of the FRTZF, in a composite control such as this, it's not allowed to. Thus, while the pattern of tapped holes is now squared up, it can still shift around nearly as much as before.

5.11.7.3 Rules for Composite Control

Datum References—Since the lower segment provides specialized refinement only within the constraints of the upper segment, the lower segment may never reference any datum(s) that contradicts the DRF of the upper segment. Neither shall there be any mismatch of material condition modifier symbols. This leaves four options for referencing datums in the lower segment.

1. Reference no datums.
2. Copy only the primary datum and its modifier (if any).
3. Copy the primary and secondary datums and their modifiers, in order.
4. Copy the primary, secondary, and tertiary datums and their modifiers, in order.



Note: All boundaries are oriented perpendicular to primary datum A.

Figure 5-122 One possible relationship between the PLTZF and FRTZF for Fig. 5-119

Only datums needed to restrain the orientation of the FRTZF may be referenced. The need for two datum references in a lower segment is somewhat rare, and for three, even more uncommon.

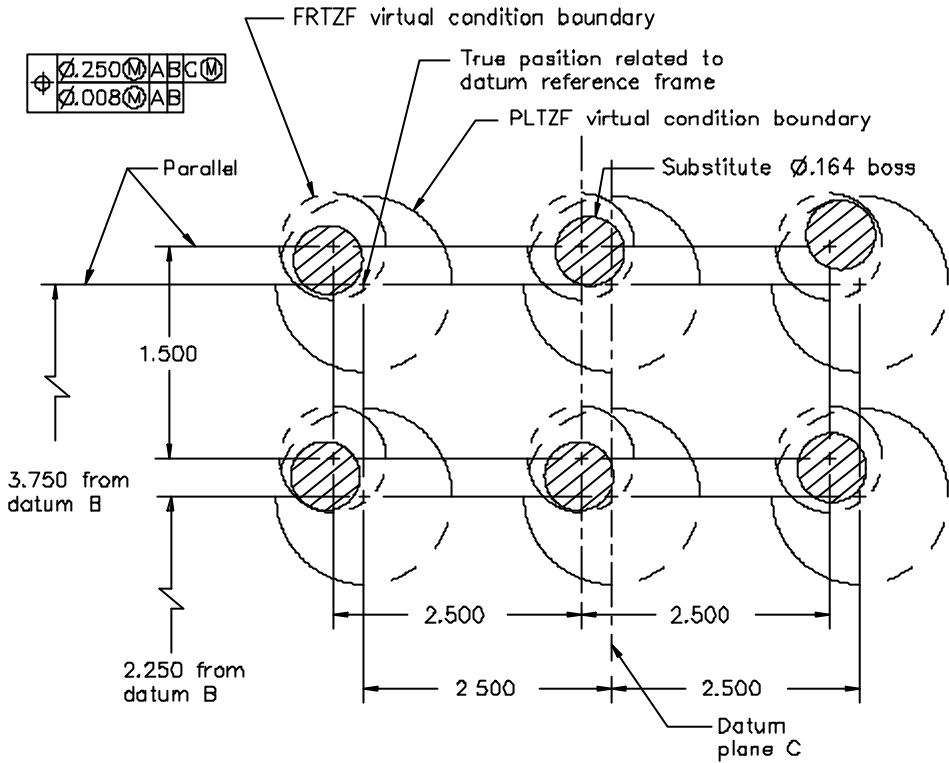
Tolerance Values—The upper-segment tolerance shall be greater than the lower-segment tolerance. Generally, the difference should be enough to make the added complexity worthwhile.

Simultaneous Requirements—The upper and lower segments may be verified separately, perhaps using two different functional gages. Thus, where both upper and lower segments reference a datum feature of size at MMC or at LMC, each segment may use a different datum derived from that datum feature. Table 5-7 shows the defaults for simultaneous requirements associated with composite control. Simultaneous requirements are explained in section 5.9.10.

FAQ: The Table 5-7 defaults seem somewhat arbitrary. Can you explain the logic?

A: No, it escapes us too.

Notice that the lower segments of composite feature control frames default to separate requirements. Placing the note SIM REQ adjacent to a lower segment that references one or more datums overrides the default and imposes simultaneous requirements. If the lower segment references no datums, functionally related features of differing sizes should instead be grouped into a single pattern of features controlled



Note: All boundaries are oriented perpendicular to primary datum A.

Figure 5-123 One possible relationship between the PLTZF and FRTZF with datum B referenced in the lower segment

Table 5-7 Simultaneous/separate requirement defaults

Between	Default	Modifiable?
Upper and lower segments within a single composite feature control frame	SEP REQTS	NO
Upper segments (only) of two or more composite feature control frames	SIM REQTS	YES
Lower segments (only) of two or more composite feature control frames	SEP REQTS	YES
Upper segment of a composite and a single-segment feature control frame	SIM REQTS	YES
Lower segment of a composite and a single-segment feature control frame	SEP REQTS	YES

with a single composite feature control frame. This can be done with a general note and flags, or with a note such as THREE SLOTS or TWO COAXIAL HOLES placed adjacent to the shared composite feature control frame.

5.11.7.4 Stacked Single-Segment Feature Control Frames

A composite positional tolerance cannot specify different location requirements for a pattern of features relative to different planes of the DRF. This is because the upper segment allows equal translation in all directions relative to the locating datum(s) and the lower segment has no effect at all on pattern translation. In section 5.11.6.2, we explained how bidirectional positional tolerancing could be used to specify different location requirements relative to different planes of the DRF. This works well for an individual feature of size, but applied to a pattern, the feature-to-feature spacings would likewise have a different tolerance for each direction.

Fig. 5-124 shows a sleeve with four radial holes. In this design, centrality of the holes to the datum A bore is critical. Less critical is the distance of the holes from the end of the sleeve, datum B. Look closely at the feature control frames. The appearance of two “position” symbols means this is not a composite positional feature control frame. What we have instead are simply two single-segment positional tolerance feature control frames stacked one on top of the other (with no space between). Each feature control frame, upper and lower, establishes a distinct framework of Level 4 virtual condition boundaries or central tolerance zones.

Fig. 5-125 shows the virtual condition boundaries for the upper frame. The boundaries are basically oriented and located to each other. In addition, the framework of boundaries is basically oriented and located relative to the referenced DRF A|B. The generous tolerance in the upper frame adequately locates the holes relative to datum B, but not closely enough to datum A.

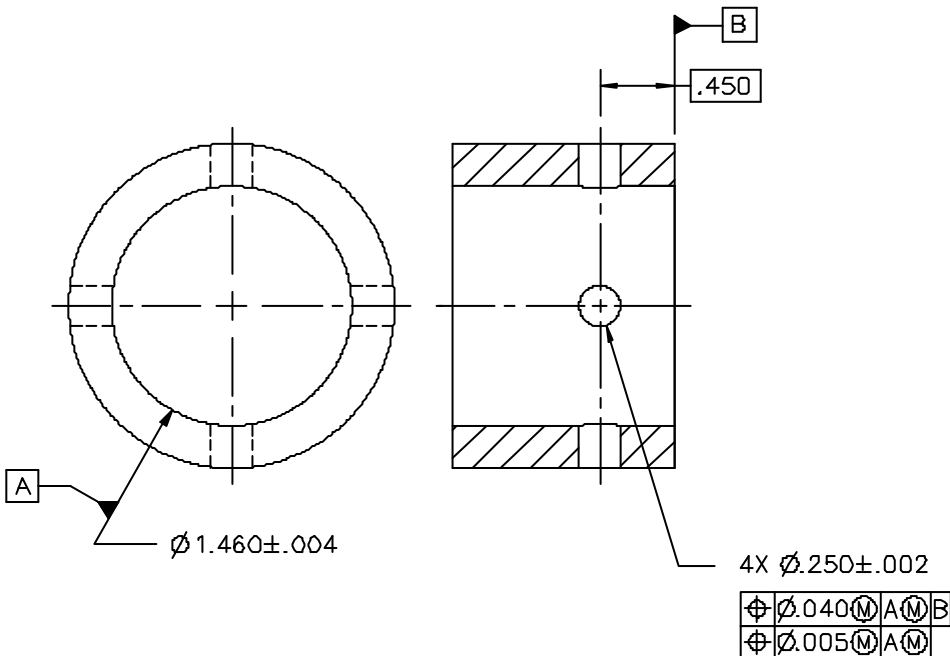


Figure 5-124 Two stacked single-segment feature control frames

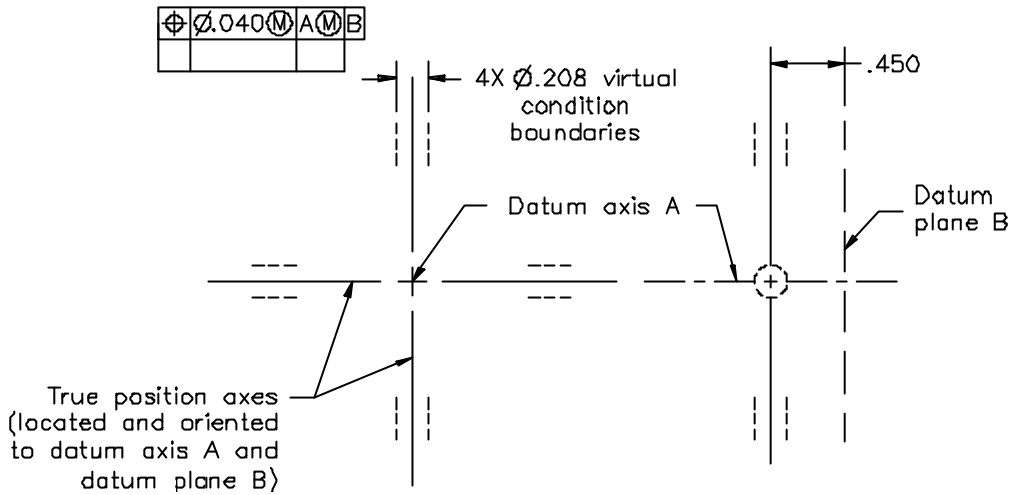


Figure 5-125 Virtual condition boundaries of the upper frame for Fig. 5-124

Fig. 5-126 shows the virtual condition boundaries for the lower frame. The boundaries are basically oriented and located to each other. In addition, the framework of boundaries is basically oriented and located relative to the referenced datum A. The comparatively close tolerance adequately centers the holes to the bore, but has no effect on location relative to datum B.

There is no explicit congruence requirement between the two frameworks. But, if features are to conform to both tolerances, virtual condition boundaries (or central tolerance zones) must overlap to some extent.

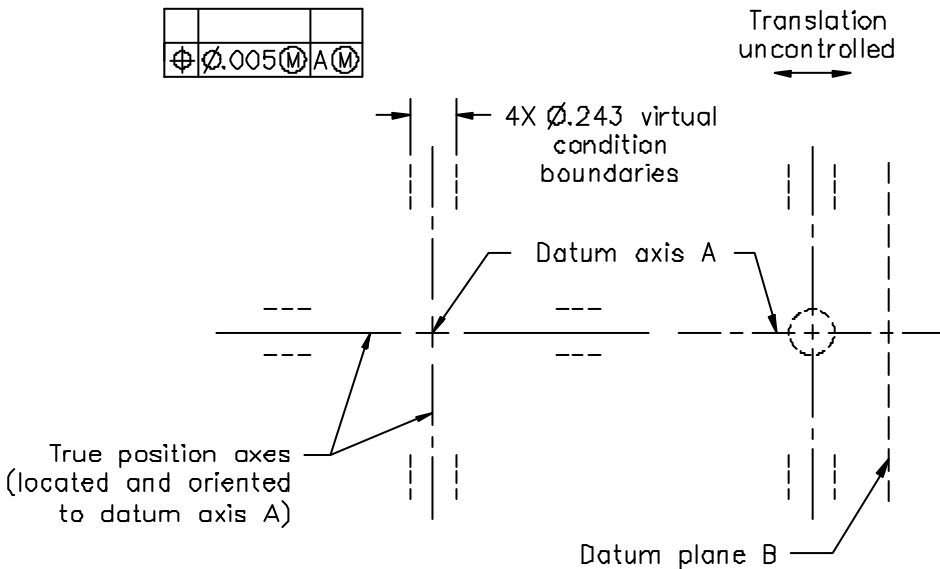


Figure 5-126 Virtual condition boundaries of the lower frame for Fig. 5-124

5.11.7.5 Rules for Stacked Single-Segment Feature Control Frames

Datum References—As with any pair of separate feature control frames, each may reference whatever datum(s), in whatever precedence, and with whatever modifiers are appropriate for the design, provided the DRFs are not identical (which would make the larger tolerance redundant). Since one frame’s constraints may or may not be contained within the constraints of the other, the designer must carefully assure that the feature control frames together provide the necessary controls of feature orientation and location to the applicable datums.

Tolerance Values—Generally, the tolerances should differ enough to justify the added complexity. It’s customary to place the frame with the greater tolerance on top.

Simultaneous Requirements—Since the two frames reference non-matching DRFs, they shall be evaluated separately, perhaps using two different functional gages. As explained in section 5.9.10, each feature control frame defaults to sharing simultaneous requirements with any other feature control frame(s) that references the identical DRF, as applicable.

FAQ: *I noticed that the 1994 revision of Y14.5 has much more coverage for pattern location than the 1982 revision. Is that just because the principles are so complicated, or does it mean I should make more use of composite and stacked feature control frames?*

A: Y14.5M-1982 was unclear about composite control as to whether the lower segment affects pattern location. Perhaps because most users assumed it did, Y14.5M-1994 includes dozens of figures meant to clarify that it does not and to introduce the method of using stacked frames. Don’t interpret the glut of coverage as a sign that composite tolerancing is extremely complicated or that it’s underused. The next revision might condense pattern location coverage.

FAQ: *How should I interpret composite tolerancing on drawings made before the 1994 revision? Does the lower segment control pattern location or not?*

A: That remains a huge controversy. Here’s what ASME Y14.5M-1982 says (in section 5.4.1.4) about an example lower segment: “The axes of individual holes must also lie within 0.25 diameter feature-relating tolerance zones basically related to each other and basically oriented to datum axis A.” Though it would have been very pertinent in the example, basic location to datum A is not mentioned. If we interpret this as an error of omission, we can likewise interpret anything left out of the standard as an error and do whatever we please. Thus, we feel the “not located” interpretation is more defensible. Where an “oriented and located” interpretation is needed on an older drawing, there’s no prohibition against “retrofitting” stacked single-segment frames.

5.11.7.6 Coaxial and Coplanar Features

All the above principles for locating patterns of features apply as well to patterns of cylindrical features arranged in-line on a common axis, or width-type features arranged on a common center plane. Fig. 5-127 shows a pattern of two coaxial holes controlled with a composite positional tolerance. Though we’ve added a third segment to our composite feature control frame, the meaning is consistent with what we described in section 5.11.7.2. The upper segment’s PLTZF controls the location and orientation of the pair of holes to the referenced DRF. The middle segment refines only the orientation (parallelism) of a FRTZF relative to datum A. The lower segment establishes a separate free-floating FRTZF that refines only the feature-to-feature coaxiality of the individual holes. Child’s play. Different sizes of in-line features can share a common positional tolerance if their size specifications are stacked above a shared feature control frame.

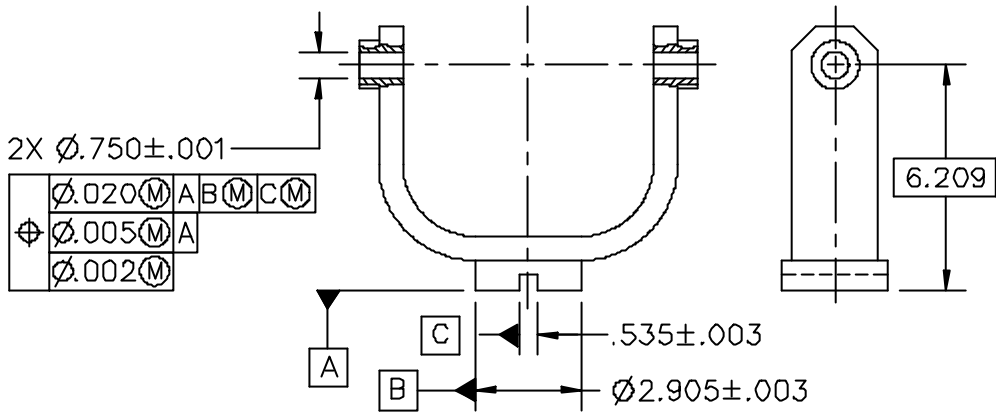


Figure 5-127 Three-segment composite feature control frame

5.11.8 Coaxiality and Coplanarity Control

Coaxiality is the relationship between multiple cylindrical or revolute features sharing a common axis. Coaxiality can be specified in several different ways, using a runout, concentricity, or positional tolerance. As Section 12 explains, a runout tolerance controls surface deviations directly, without regard for the feature's axis. A concentricity tolerance, explained in section 5.14.3, controls the midpoints of diametrically opposed points.

The standards don't have a name for the relationship between multiple width-type features sharing a common center plane. We will extend the term *coplanarity* to apply in this context. Coplanarity can be specified using either a symmetry or positional tolerance. A symmetry tolerance, explained in section 5.14.4, controls the midpoints of opposed surface points.

Where one of the coaxial or coplanar features is identified as a datum feature, the coaxiality or coplanarity of the other(s) can be controlled directly with a positional tolerance applied at RFS, MMC, or LMC. Likewise, the datum reference can apply at RFS, MMC, or LMC. For each controlled feature, the tolerance establishes either a Level 4 virtual condition boundary or a central tolerance zone (see section 5.11.1) located at true position. In this case, no basic dimensions are expressed, because true position is coincident with the referenced datum axis or datum center plane.

All the above principles can be extended to a pattern of coaxial feature groups. For a pattern of counterbored holes, the pattern of holes is located as usual. A single "datum feature" symbol is attached according to section 5.9.2.4. Coaxiality for the counterbores is specified with a separate feature control frame. In addition, a note such as 4X INDIVIDUALLY is placed under the "datum feature" symbol and under the feature control frame for the counterbores, indicating the number of places each applies on an individual basis.

Where the coaxiality or coplanarity of two features is controlled with a positional tolerance of zero at MMC and the datum is also referenced at MMC, it makes no difference which of these features is the datum. For each feature, its TGC, its virtual condition, and its MMC size limit are identical. The same is true in an all-LMC context.

FAQ: *Where a piston's ring grooves interrupt the outside diameter (OD), do I need to control coaxiality among the three separate segments of the OD?*

A: If it weren't for those pesky grooves, Rule #1 would impose a boundary of perfect form at MMC for the entire length of the piston's OD. Instead of using 3X to specify multiple same-size ODs, place the note **THREE SURFACES AS A SINGLE FEATURE** adjacent to the diameter dimension. That forces Rule #1 to ignore the interruptions. A similar note can simplify orientation and/or location control of a pattern of coaxial or coplanar same-size features.

5.12 Runout Tolerance

Runout is one of the oldest and simplest concepts used in GD&T. Maybe as a child you stood your bicycle upside down on the ground and spun a wheel. If you fixed your stare on the shiny rim where it passed a certain part of the frame, you could see the rim wobble from side to side and undulate inward and outward. Instead of the rim running in a perfect circle, it, well—ran out. *Runout*, then, is the variation in the surface elements of a round feature relative to an axis.

5.12.1 Why Do We Use It?

In precision assemblies, runout causes misalignment and/or balance problems. In Fig. 5-128, runout of the ring groove diameters relative to the piston's diameter might cause the ring to squeeze unevenly around the piston or force the piston off center in its bore. A motor shaft that runs out relative to its bearing journals will cause the motor to run out-of-balance, shortening its working life. A designer can prevent such wobble and lopsidedness by specifying a runout tolerance. There are two levels of control, *circular runout* and *total runout*. Total runout adds further refinement to the requirements of circular runout.

5.12.2 How Does It Work?

For as long as piston ring grooves and motor shafts have been made, manufacturers have been finding ways to spin a part about its functional axis while probing its surface with a dial indicator. As the indicator's tip surfs up and down over the undulating surface, its dial swings gently back and forth, visually display-

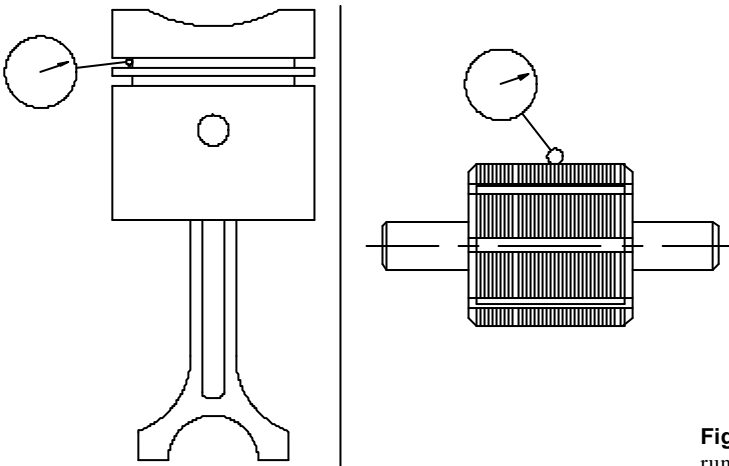


Figure 5-128 Design applications for runout control

ing the magnitude of runout. Thus, measuring runout can be very simple as long as we agree on three things:

- What surface(s) establish the functional axis for spinning—datums
- Where the indicator is to probe
- How much swing of the indicator’s dial is acceptable

The whole concept of “indicator swing” is somewhat dated. Draftsmen used to annotate it on drawings as TIR for “Total Indicator Reading.” Y14.5 briefly called it FIR for “Full Indicator Reading.” Then, in 1973, Y14.5 adopted the international term, FIM for “Full Indicator Movement.” *Full Indicator Movement (FIM)* is the difference (in millimeters or inches) between an indicator’s most positive and most negative excursions. Thus, if the lowest reading is $-.001$ " and the highest is $+.002$ ", the FIM (or TIR or FIR) is $.003$ ".

Just because runout tolerance is defined and discussed in terms of FIM doesn’t mean runout tolerance can only be applied to parts that spin in assembly. Neither does it require the part to be rotated, nor use of an antique twentieth century, jewel-movement, dial indicator to verify conformance. The “indicator swing” standard is an ideal meant to describe the requirements for the surface. Conformance can be verified using a CMM, optical comparator, laser scanning with computer modeling, process qualification by SPC, or any other method that approximates the ideal.

5.12.3 How to Apply It

A runout tolerance is specified using a feature control frame displaying the characteristic symbol for either “circular runout” (a single arrow) or “total runout” (two side-by-side arrows). As illustrated in Fig. 5-129, the arrowheads may be drawn filled or unfilled. The feature control frame includes the runout tolerance value followed by one or two (but never three) datum references.

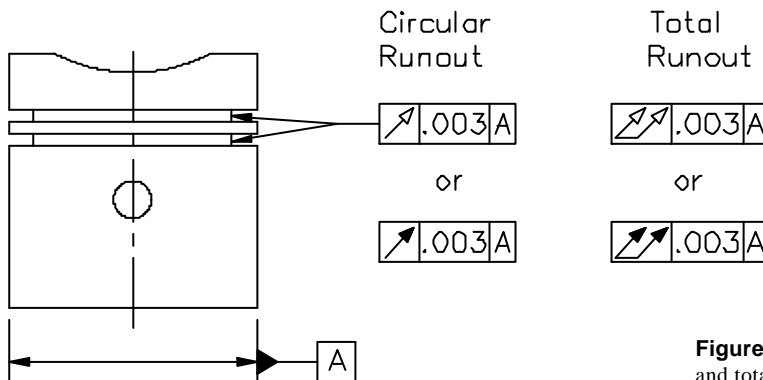


Figure 5-129 Symbols for circular runout and total runout

Considering the purpose for runout tolerance and the way it works, there’s no interaction between a feature’s size and its runout tolerance that makes any sense. In our piston ring groove diameter example, an MMC modifier would be counterproductive, allowing the groove diameter’s eccentricity to increase as it gets smaller. That would only aggravate the squeeze and centering problems we’re trying to correct. Thus, material condition modifier symbols, MMC and LMC, are prohibited for both circular and total runout tolerances and their datum references. If you find yourself wishing you could apply a runout tolerance at MMC, you’re not looking at a genuine runout tolerance application; you probably want positional tolerance instead.

5.12.4 Datums for Runout Control

A runout tolerance controls surface elements of a round feature relative to a datum axis. GD&T modernized runout tolerancing by applying the rigors and flexibility of the DRF. Every runout tolerance shall reference a datum axis. Fig. 5-130 shows three different methods for doing this.

Since a designer wishes to control the runout of a surface as directly as possible, it's important to select a functional feature(s) to establish the datum axis. During inspection of a part such as that shown in Fig. 5-130(a), the datum feature might be placed in a V-block or fixtured in a precision spindle so that the part can be spun about the axis of the datum feature's TGC. This requires that the datum feature be long enough and that its form be well controlled (perhaps by its own size limits or form tolerance). In addition, the datum feature must be easily accessible for such fixturing or probing.

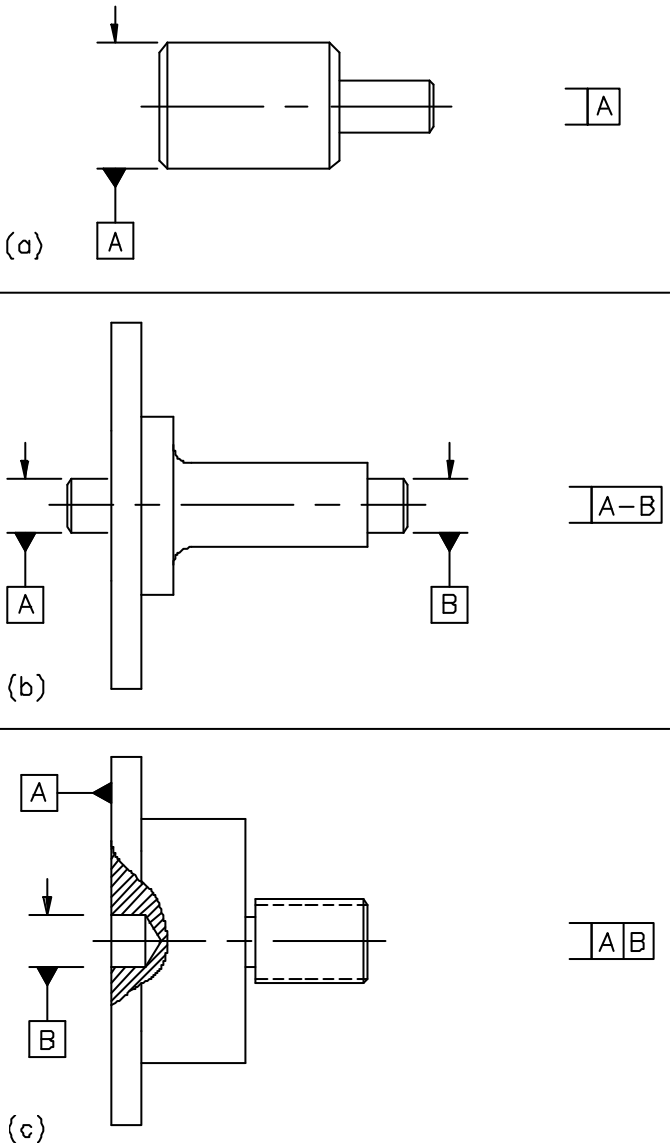


Figure 5-130 Datums for runout control

There are many cases where the part itself is a spindle or rotating shaft that, when assembled, will be restrained in two separate places by two bearings or two bushings. See Fig. 5-131. If the two bearing journals have ample axial separation, it's unrealistic to try to fixture on just one while ignoring the other. We could better stabilize the part by identifying each journal as a datum feature and referencing both as equal co-datum features. In the feature control frame, the datum reference letters are placed in a single box, separated by a hyphen. As we explained in section 5.9.14.2, hyphenated co-datum features work as a team. Neither co-datum feature has precedence over the other. We can't assume the two journals will be made perfectly coaxial. To get a decent datum axis from them, we should add a runout tolerance for each journal, referencing the common datum axis they establish. See Fig. 5-132. This is one of the few circumstances where referencing a feature as a datum in its own feature control frame is acceptable.

Where a single datum feature or co-datum feature pair establishes the axis, further datum references are meaningless and confusing. However, there are applications where a shoulder or end face exerts more leadership over the part's orientation in assembly while the diametral datum feature merely establishes the center of revolution. In Fig. 5-130(c), for example, the face is identified as primary datum feature A and the bore is labeled secondary datum feature B. In inspection, the part will be spun about datum axis B which, remember, is restrained perpendicular to datum plane A.

5.12.5 Circular Runout Tolerance

Circular runout is the lesser level of runout control. Its tolerance applies to the FIM while the indicator probes over a single circle on the part surface. That means the indicator's body is to remain stationary both axially and radially relative to the datum axis as the part is spun at least 360° about its datum axis. The tolerance applies at every possible circle on the feature's surface, but each circle may be evaluated separately from the others.

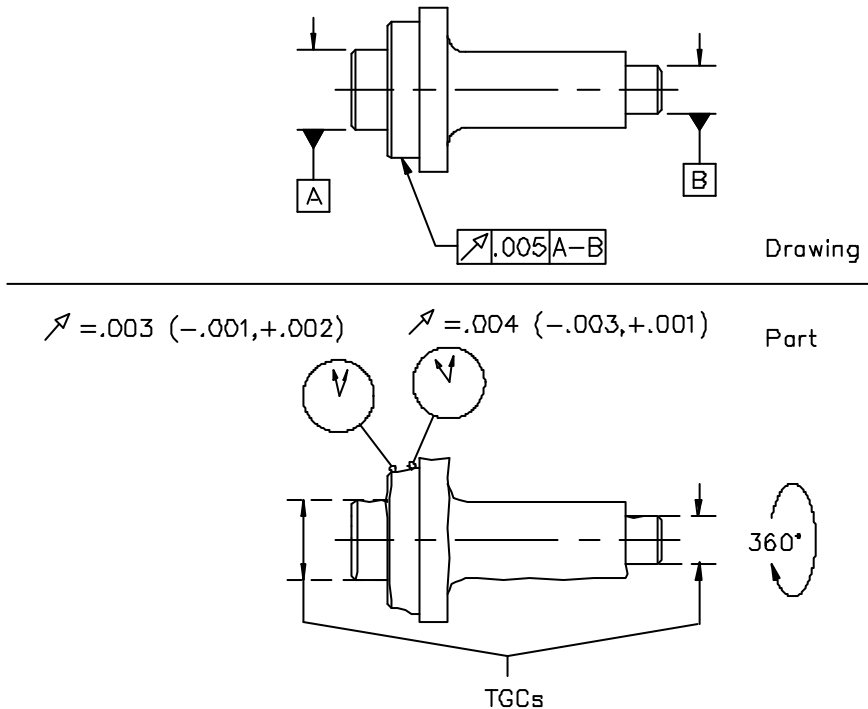


Figure 5-131 Two coaxial features establishing a datum axis for runout control

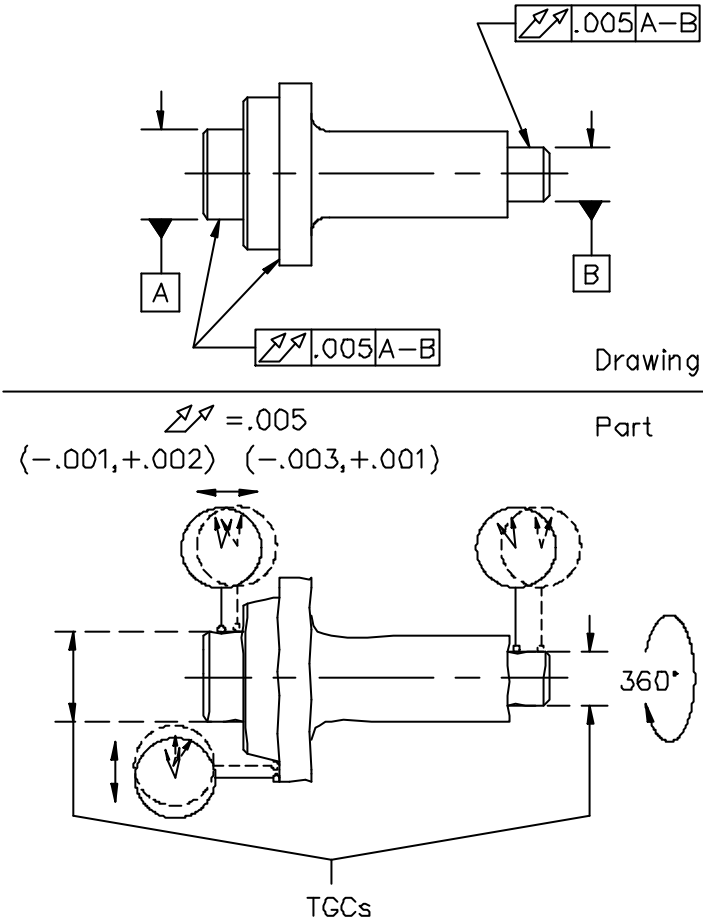


Figure 5-132 Runout control of hyphenated co-datum features

Let's evaluate the .005 circular runout tolerance of Fig. 5-131. We place an indicator near the left end of the controlled diameter and spin the part 360°. We see that the farthest counterclockwise excursion of the indicator dial reaches $-.001$ " and the farthest clockwise excursion reaches $+.002$ ". The circular runout deviation at that circle is $.003$ ". We move the indicator to the right and probe another circle. Here, the indicator swings between $-.003$ " and $+.001$ ". The difference, $.004$ ", is calculated without regard for the readings we got from the first circle. The FIM for each circle is compared with the $.005$ " tolerance separately.

Obviously, we can't spend all day trying to measure infinitely many circles, but after probing at both ends of the feature and various places between, we become confident that no circle along the feature would yield an FIM greater than, perhaps, $.004$ ". Then, we can conclude the feature conforms to the $.005$ " circular runout tolerance.

Circular runout can be applied to any feature that is nominally cylindrical, spherical, toroidal, conical, or any revolute having round cross sections (perpendicular to the datum axis). When evaluating noncylindrical features, the indicator shall be continually realigned so that its travel is always normal to the surface at the subject circle. See Fig. 5-133. Circular runout can also be applied to a face or face groove that is perpendicular to the datum axis. Here, the surface elements are circles of various diameters, each concentric to the datum axis and each evaluated separately from the others.

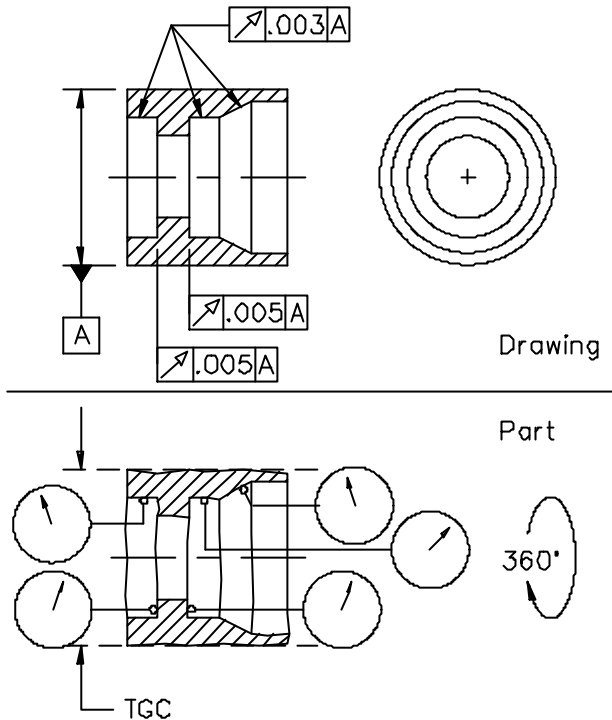


Figure 5-133 Application of circular runout

5.12.6 Total Runout Tolerance

Total runout is the greater level of runout control. Its tolerance applies to the FIM while the indicator sweeps over the entire controlled surface. Rather than each circular element being evaluated separately, the total runout FIM encompasses the highest and lowest of all readings obtained at all circles.

For a nominally cylindrical feature, the indicator's body shall be swept parallel to the datum axis, covering the entire length of the controlled feature, as the part is spun 360° about the datum axis. See Fig. 5-132. Any taper or hourglass shape in the controlled feature will increase the FIM.

For a nominally flat face perpendicular to the datum axis, the indicator's body shall be swept in a line perpendicular to the datum axis, covering the entire breadth of the controlled feature. Any conicity, wobble, or deviations from flatness in the controlled feature increase the FIM. The control imposed by this type of total runout tolerance is identical to that of an equal perpendicularity tolerance with an RFS datum reference.

FAQ: *Can total runout tolerance be applied to a cone?*

A: For any features other than cylinders or flat perpendicular faces, the indicator would have to be swept along a path neither parallel nor perpendicular to the datum axis. Since the standards have not adequately defined these paths, avoid such applications.

5.12.7 Application Over a Limited Length

Since a runout tolerance applies to surface elements, it sometimes makes sense to limit the control to a limited portion of a surface. A designer can do this easily by applying a chain line as described in section 5.8.8.

5.12.8 When Do We Use a Runout Tolerance?

Runout tolerances are especially suited to parts that revolve about a datum axis in assembly, and where alignments and dynamic balance are critical. Circular runout tolerance is often ideal for O-ring groove diameters, but watch out for surfaces inaccessible to an indicator tip. This might be an internal O-ring groove where the cylinder bore is the datum. How can an inspector spin the part about that bore and get his indicator tip into the groove at the same time? As we said, there are other inspection methods, but a designer should always keep one eye on practicality.

The following equations pertain to the controls imposed by circularity, cylindricity, concentricity, circular runout, and total runout when applied to a revolute or cylindrical feature.

$$\text{CIRCULARITY} + \text{CONCENTRICITY} = \text{CIRCULAR RUNOUT}$$

$$\text{CYLINDRICITY} + \text{CONCENTRICITY} = \text{TOTAL RUNOUT}$$

Remember that FIM is relatively simple to measure and reflects the combination of out-of-roundness and eccentricity. It's quite complex to differentiate between these two constituent variations. That means checking circularity or concentricity apart from the other requires more sophisticated and elaborate techniques. Of course, there are cases where the design requires tight control of one (say, circularity); to impose the same tolerance for the other (concentricity) would significantly complicate manufacturing. However, if this won't be a problem, use a runout tolerance.

A runout tolerance applies directly to surface elements. That distinguishes it from a positional tolerance RFS that controls only the coaxiality of the feature's actual mating envelope. Positional tolerancing provides no form control for the surface. While the positional tolerance coaxiality control is similar to that for runout tolerance, the positional tolerance is modifiable to MMC or LMC. Thus, where tolerance interaction is desirable and size limits will adequately control form, consider a positional tolerance instead of a runout tolerance.

FAQ: *Can I apply a runout tolerance to a gear or a screw thread?*

A: Avoid doing that. Remember that a runout tolerance applies to the FIM generated by surface elements. Some experts suggest modifying the runout tolerance by adding the note **PITCH CYLINDER**. We feel that subverts the purpose for runout tolerance and requires unique and complicated inspection methods. Consider a positional tolerance instead.

FAQ: *A feature's runout tolerance has to be less than its size tolerance, right?*

A: Wrong. A feature's size limits don't control its runout; neither does a runout tolerance control the feature's size. Depending on design considerations, a runout tolerance may be less than, equal to, or greater than the size tolerance. One can imagine scenarios justifying just about any ratio. That's why it's important to consider each runout tolerance independently and carefully.

FAQ: *Can I apply a runout tolerance "unless otherwise specified" in the tolerance block or by a general note?*

A: Yes, but identify a datum feature and reference it with the runout tolerance. A runout tolerance with no datum reference is meaningless and illegal. Many novice inspectors encountering a general runout tolerance with no datum reference start checking every possible pairing of features—for five diameters, that's 20 checks! Also, consider each feature to which the runout tolerance will apply and be careful not to rob any feature of usable and needed tolerance.

5.12.9 Worst Case Boundaries

Instead of troweling on feature control frames for form and location, a clever designer can often simplify requirements by using a few well-thought-out runout tolerances to control combinations of relationships.

A circular runout or total runout tolerance applied to an internal or external diameter feature yields a worst case inner boundary equal in size to the feature's small-limit size minus the value of its runout tolerance and a worst case outer boundary equal in size to the feature's large-limit size plus the value of its runout tolerance. The inner or outer boundary can be exploited to protect a secondary requirement for clearance without using a separate positional tolerance.

5.13 Profile Tolerance

In the previous sections, we've covered nearly all the principles needed to control planar features and simple features of size. In the old MIL-STD-8 drawing standards, that was as far as GD&T went. However, automobiles, airplanes, and ships are replete with parts having nonplanar, noncylindrical, nonspherical features. Such irregularly shaped *profiled features* couldn't be geometrically controlled until 1966 when the first edition of Y14.5 introduced "profile of a line" and "profile of a surface" characteristic symbols and feature control frames for controlling profiled features. The 1973 revision of Y14.5 introduced datum references in profile feature control frames. Finally, designers could apply all the power and precision of GD&T to nearly every imaginable type of part feature.

The 1982 and 1994 revisions of Y14.5 enhanced the flexibility of profile tolerancing to the extent that now just about every characteristic of just about every type of feature (including planes and simple features of size) can be controlled with a profile tolerance. Thus, some gurus prescribe profile tolerancing for everything, as if it's "the perfect food." (We address that notion in Section 17.)

The fundamental principles of profile tolerancing are so simple that the Math Standard covers them fully with just one column of text. However, the Math Standard only addresses the meaning of the tolerance. Profile tolerancing's multitude of application options and variations comprise quite a lot of material to learn.

5.13.1 How Does It Work?

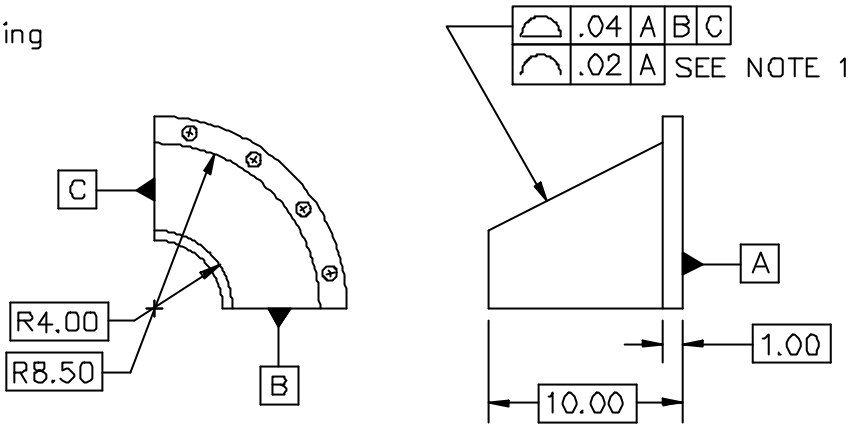
Every profile tolerance relies on a *basic profile*. See Fig. 5-134. This is the profiled feature's nominal shape usually defined in a drawing view with basic dimensions. A profile tolerance zone is generated by offsetting each point on the basic profile in a direction normal to the basic profile at that point. This offsetting creates a "band" that follows the basic profile. The part feature (or 2-D element thereof) shall be contained within the profile tolerance zone. In addition, the surface (or 2-D element) shall "blend" everywhere. We interpret this to mean it shall be tangent-continuous.

There are two levels of profile tolerance control. The difference between the two levels is analogous to the difference between flatness and straightness tolerances. *Profile of a surface* provides complete 3-D control of a feature's total surface. *Profile of a line* provides 2-D control of a feature's individual cross-sectional elements. Either type of control may be related to a DRF.

5.13.2 How to Apply It

Application of a profile tolerance is a three-step process: 1) define the basic profile, 2) define the tolerance zone disposition relative to the basic profile, and 3) attach a profile feature control frame.

Drawing



NOTE 1: THIS TOLERANCE APPLIES WHEN DATUM FEATURE A IS MOUNTED AGAINST A FLAT SURFACE USING FOUR .250-28 BOLTS TORQUED TO 10 FOOT POUNDS.

Part

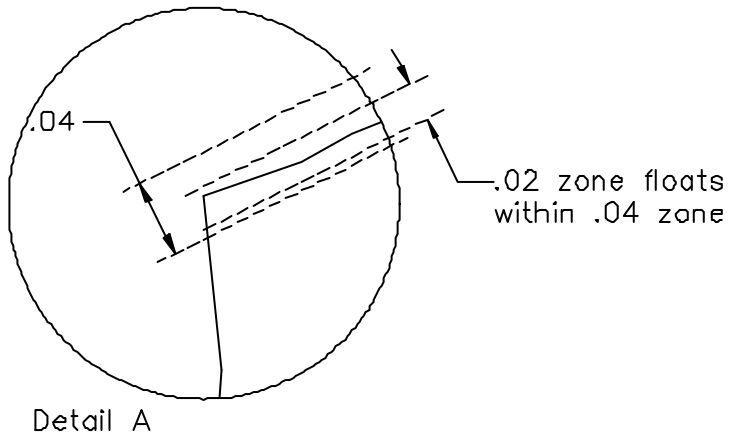
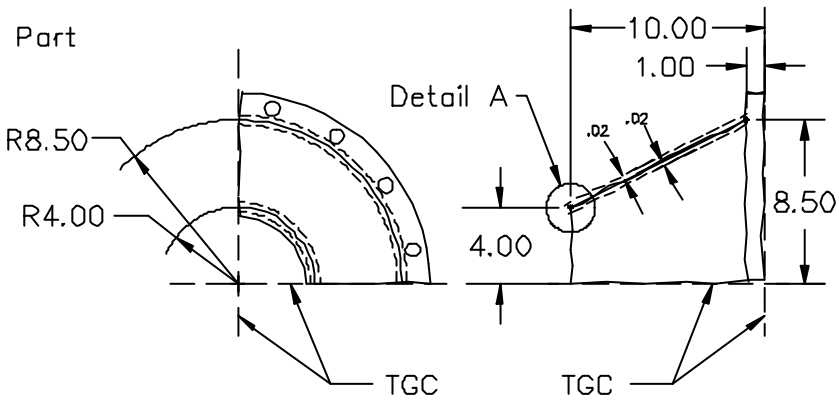


Figure 5-134 Application of profile tolerances

5.13.3 The Basic Profile

You can specify the basic profile by any method that defines a unique and unambiguous shape for the controlled feature. The most common methods are projecting a 3-D figure onto a plane or taking cross sections through the figure. The resulting 2-D profile is shown in a drawing view. We call this 2-D graphical representation the *profile outline*. Basic dimensions are specified for the basic profile to define each of its elements. Such basic dimensions may include lengths, diameters, radii, and angles. Alternatively, a coordinate grid system might be established, with points or nodes on the basic profile listed in a table. Yet another method is to provide one or more mathematical formulas that define the elements of the basic profile, perhaps accompanied by one or more basically dimensioned nodes or end points.

A CAD/CAM model's digital representation of a basic profile also qualifies. It's not necessary to attach basic dimensions to the model since the computer already "understands" the ones and zeros that define it. In a paperless manufacturing environment, the "undimensioned" model along with a profile tolerance specification are all that's needed by automated equipment to make and inspect the profiled feature. This method accommodates truly 3-D-profiled features having varying cross sections, such as a turbine blade or an automobile windshield.

While any of these or other methods could be used, the designer must take into account the expected manufacturing methods and ensure that the basic profile specifications are accessible and usable. This consideration may prescribe multiple 2-D drawing views to show, for example, an airplane wing at several different cross sections.

5.13.4 The Profile Tolerance Zone

As depicted in Fig. 5-135, the profile tolerance zone is generated by offsetting each point on the basic profile in a direction normal to the basic profile at that point. This tolerance zone may be *unilateral* or *bilateral* relative to the basic profile. For a unilateral profile tolerance, the basic profile is offset totally in one direction or the other by an amount equal to the profile tolerance. See Figs. 5-135(b) and (c). For a bilateral profile tolerance, the basic profile is offset in both directions by a combined amount equal to the profile tolerance. Equal offsets of half the tolerance in each direction—*equal-bilateral tolerance*—is the default. See Fig. 5-135(a). Though the offsets need not be equal, they shall be uniform everywhere along the basic profile.

Regardless of the tolerance zone's disposition relative to the basic profile, it always represents the range of allowable variation for the feature. You could also think of this disposition as the basic profile running along one boundary of the tolerance band, or somewhere between the two boundaries. In any case, since the variations in most manufacturing processes tend to be equal/bidirectional, programmers typically program tool paths to target the mean of the tolerance zone. With an equal-bilateral tolerance, the basic profile runs right up the middle of the tolerance zone. That simplifies programming because the drawing's basic dimensions directly define the mean tool path without any additional calculations. Programmers love equal-bilateral tolerances, the default.

Of course, a unilateral tolerance is also acceptable. The drawing shall indicate the offset direction relative to the basic profile. Do this as shown in Fig. 5-135(b) and (c) by drawing a phantom line parallel to the basic profile on the tolerance zone side. Draw the phantom line (or curve) only long enough to show clearly. The distance between the profile outline and the phantom line is up to the draftsman, but should be no more than necessary for visibility after copying (don't forget photoreduction), and need not be related to the profile tolerance value.

A pair of short phantom lines can likewise be drawn to indicate a bilateral tolerance zone with unequal distribution. See Fig. 5-135(d). Draw one phantom line on each side of the profile outline with one visibly farther away to indicate the side having more offset. Then, show one basic dimension for the distance between the basic profile and one of the boundaries represented by a phantom line.

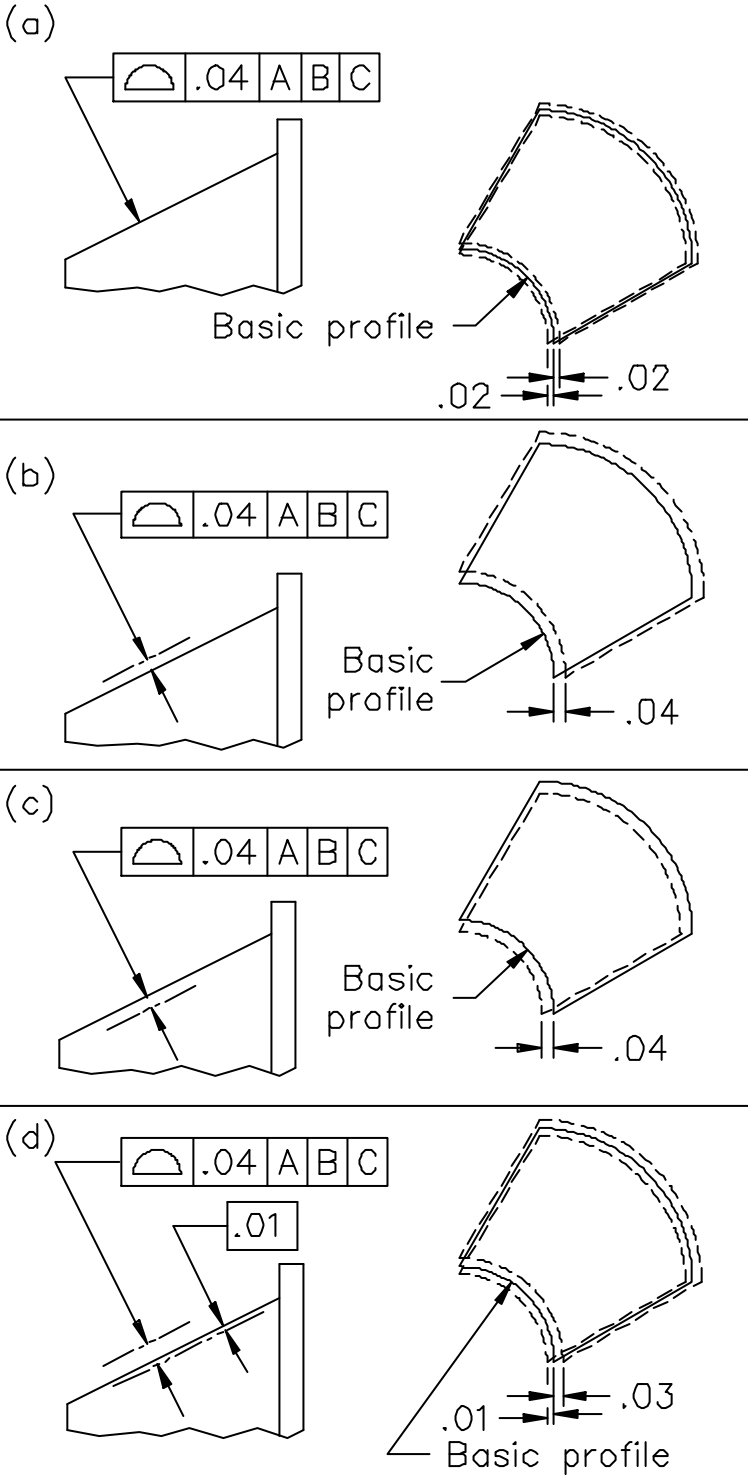


Figure 5-135 Profile tolerance zones

On complex and dense drawings, readers often fail to notice and comprehend such phantom lines, usually with disastrous consequences. Unequal-bilateral tolerancing is particularly confusing. If practicable, designers should spend a few extra minutes to convert the design for equal-bilateral tolerances. The designer will only have to make the computations once, precluding countless error-prone calculations down the road.

5.13.5 The Profile Feature Control Frame

A profile tolerance is specified using a feature control frame displaying the characteristic symbol for either “profile of a line” (an arc with no base line) or “profile of a surface” (same arc, with base line). The feature control frame includes the profile tolerance value followed by up to three datum references, if needed. Where the profile tolerance is equal-bilateral, the feature control frame is simply leader-directed to the profile outline, as in Fig. 5-135(a). Where the tolerance is unilateral or unequal-bilateral, dimension lines are drawn for the width of the tolerance zone, normal to the profile as in Fig. 5-135(b) through (d). One end of a dimension line is extended to the feature control frame.

5.13.6 Datums for Profile Control

Where a profile tolerance need only control a feature’s shape, it’s unnecessary to relate the profile tolerance zone to any DRF. Thus, there are many applications where the profile feature control frame should have no datum references. Where the tolerance must also control the orientation, or orientation and location of the profiled feature, the tolerance zone shall be related to a DRF. Depending on design requirements, the DRF may require one, two, or three datum references in the profile feature control frame.

5.13.7 Profile of a Surface Tolerance

A feature control frame bearing the “profile of a surface” symbol specifies a 3-D tolerance zone having a total width equal to the tolerance value. The entire feature surface shall everywhere be contained within the tolerance zone. If a DRF is referenced, it restrains the orientation, or orientation and location of the tolerance zone.

5.13.8 Profile of a Line Tolerance

A feature control frame bearing the “profile of a line” symbol specifies a tolerance zone plane containing a 2-D profile tolerance zone having a total width equal to the tolerance value. As the entire feature surface is swept by the tolerance zone plane, its intersection with the plane shall everywhere be contained within the tolerance zone.

Where no DRF is referenced, the tolerance plane’s orientation and sweep shall be normal to the basic profile at each point along the profile. For a revolute, such as shown in Fig. 5-136, the plane shall sweep radially about an axis. Within the plane, the orientation and location of the tolerance zone may adjust continuously to the part surface while sweeping. Alternatively, one or two datums may be referenced as necessary to restrain the orientation of the tolerance plane as it sweeps. Depending on the datums chosen, the DRF might also restrain the orientation of the tolerance zone within the sweeping plane. Any basic dimensions that locate the zone relative to the referenced DRF will restrain the zone’s location as well. Addition of a secondary or tertiary datum reference could arrest for the zone all three degrees of translation. For a nominally straight surface, the sweeping plane would then generate a 3-D zone identical to that specified by the “profile of a surface” symbol. To limit the control to 2-D, then, a designer must be careful not to overrestrain the tolerance plane and zone.

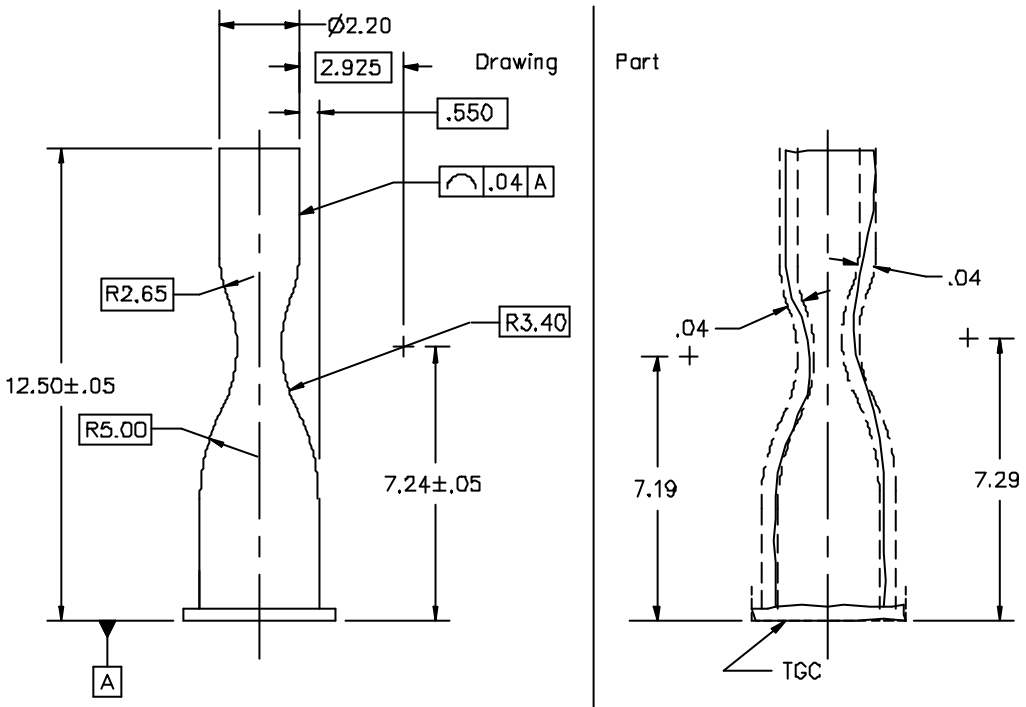


Figure 5-136 Profile of a line tolerance

FAQ: How can I get the orientation restraint I need from a DRF without getting location restraint I don't want?

A: Currently, there's no symbolic way to "switch off" a DRF's origins. In the rare case where basic dimensions define the basic profile, but you don't want the location restraint, you'll have to add a note to the drawing.

5.13.9 Controlling the Extent of a Profile Tolerance

By default, a single profile tolerance applies to a single tangent-continuous profiled feature. There are cases where a feature's tangency or continuity is interrupted, inconveniently dividing it into two or more features. We'd hate to plaster identical profile feature control frames all around a drawing view like playbills at a construction site. In other cases, different portions of a single feature should have different profile tolerances. An example is where only a portion of a feature is adjacent to a thin wall.

Y14.5 provides three tools for expanding or limiting the extent of a profile tolerance: the "all around" symbol, the ALL OVER note, and the "between" symbol. These allow the designer very precise control of profiled features. In our explanations for them, we'll be referring to the *subject view*—a single drawing view that shows a profile outline with a profile feature control frame.

The “all around” symbol (a circle) modifies a profile tolerance to apply all around the entire outline shown in the subject view regardless of breaks in tangency. As in Fig. 5-137, the symbol is drawn at the “elbow” in the leader line from the feature control frame. “All around” control does not extend to surfaces or edges parallel to the viewing plane or to any feature not shown in the subject view.

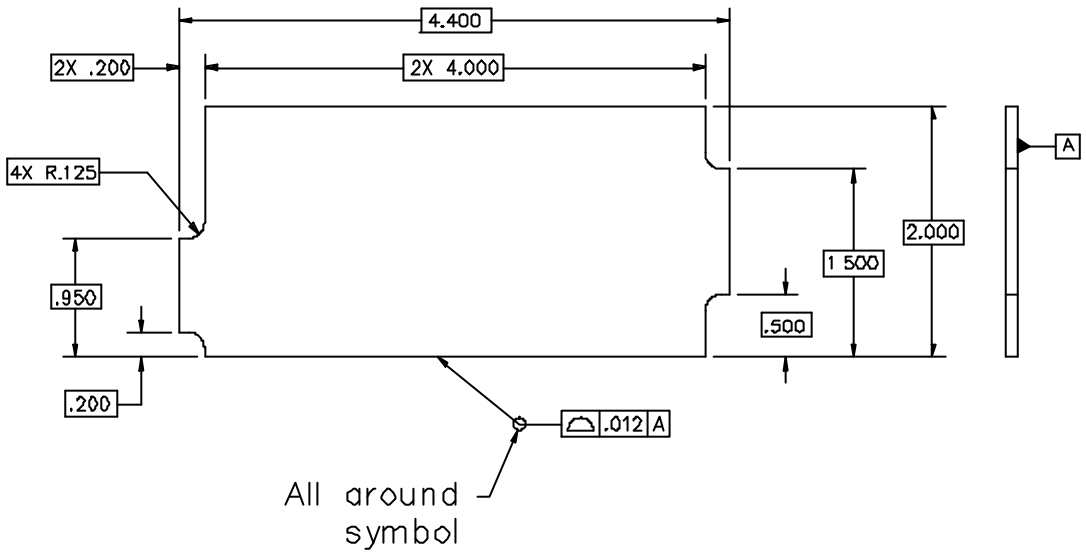


Figure 5-137 Profile “all around”

The note ALL OVER has not yet been replaced with a symbol. When the note appears below a profile feature control frame, as in Fig. 5-138, it modifies the profile tolerance to extend all over every surface of the part, including features or sections not shown in the subject view. (Any feature having its own specifications is exempt.) The few applications where this is appropriate include simple parts, castings, forgings, and truly 3-D profiled features. For example, we might specify an automobile door handle or the mold for a shampoo bottle with profile of a surface ALL OVER.

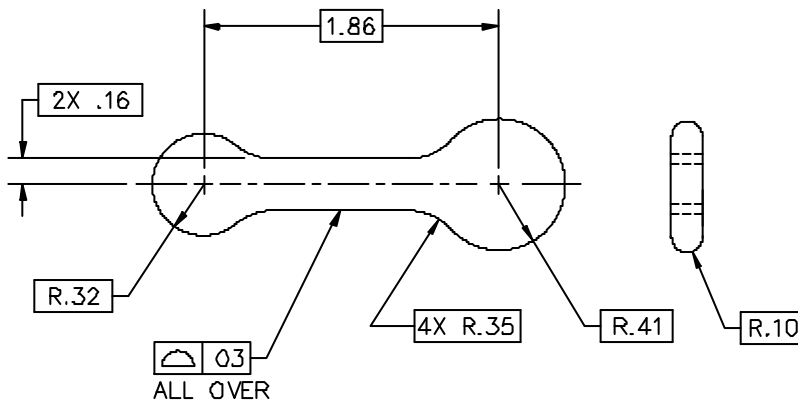


Figure 5-138 Profile “all over”

The third method is to indicate (in the subject view) two points along the basic profile as terminations for the subject tolerance zone. Each point is designated by directing a reference letter to the point with a leader. See Fig. 5-139. If a terminating point is not located at an obvious break in the continuity or tangency of the basic profile, it shall be located with basic dimensions. In addition, the same two reference letters are repeated adjacent to the profile feature control frame, separated by the “between” symbol (a two-headed arrow). The tolerance applies along the basic profile only between the designated terminating points. Neither the choice of reference letters, their relative placement in the subject view, nor their sequence before or after the “between” symbol have any bearing on which portion of the feature is concerned. Where the profile outline closes upon itself, as in Fig. 5-139, the terminating points divide the outline into two portions, both of which can be interpreted as “between” the pair of points. The tolerance applies only to the portion having a leader from the feature control frame. A more complex profile outline having multiple feature control frames with more than two terminating points might require more care in clarifying the extents of the zones.

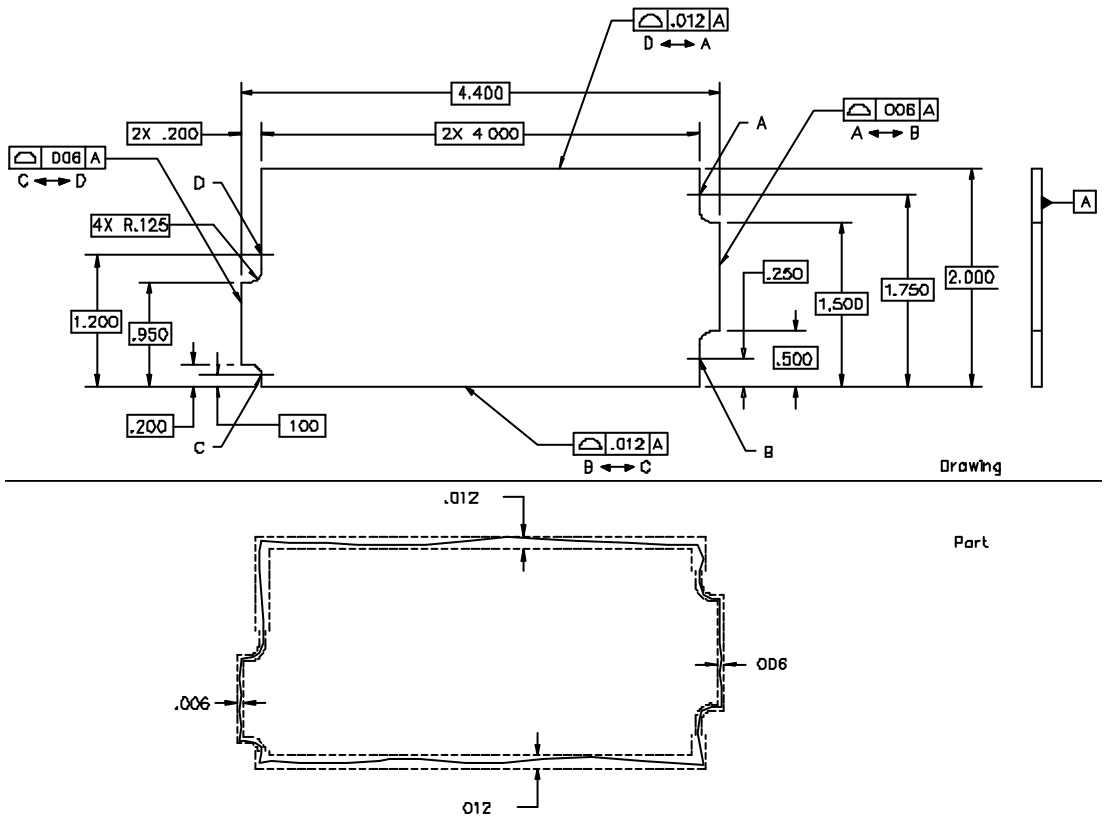


Figure 5-139 Profile “between” points

If, by using any of the above techniques, a profile tolerance is extended to include a sharp corner, the boundary lines for each adjacent surface are extended to intersect. In some designs, the intersection of the zones may not provide adequate control of the corner radius. A separate radius tolerance (as described in section 5.8.10) may be applied as a refinement of the profile control.

5.13.10 Abutting Zones

Abutting profile tolerance zones having boundaries with dissimilar offsets can impose weird or even impossible constraints on the surface. For example, if a zone unilaterally offset in one direction abuts a zone unilaterally offset in the other direction, the transition between zones has zero width. Where zones intersect at a corner, the surface radius could have concave, convex, and straight portions. A designer must carefully consider what the surface contour will be through the transition.

Remember that manufacturing variation tends to be equal/bidirectional, and that tool path programmers target the mean of the tolerance zone. Thus, where the designer makes a narrow unilateral zone about a much wider unilateral zone, the tool path within the wider zone is “programmer’s choice.” The programmer might choose to do one of the following.

- Keep the tool path consistently close to the basic profile, discarding tolerance in the wider zone.
- Make an abrupt step in the surface to always follow the median.
- Make a tapered transition to the median.

Since none of the choices are completely satisfactory, we have one more reason to try to use equal-bilateral tolerance zones.

5.13.11 Profile Tolerance for Combinations of Characteristics

By skillfully manipulating tolerance values and datum references, an expert designer can use profile tolerancing to control a surface’s form, orientation, and/or location. That’s desirable where other types of tolerances, such as size limits, flatness, and angularity tolerances are inapplicable or awkward. For example, in Fig. 5-140, the profile tolerance controls the form of a conical taper. The reference to datum A additionally controls the cone’s orientation, and the reference to datum B controls the axial location of the cone relative to the end face. In this case, size limits are useless, but a single profile tolerance provides simple and elegant control. In other cases where more specialized controls will work just fine, it’s usually less confusing if the designer applies one or more of them instead.

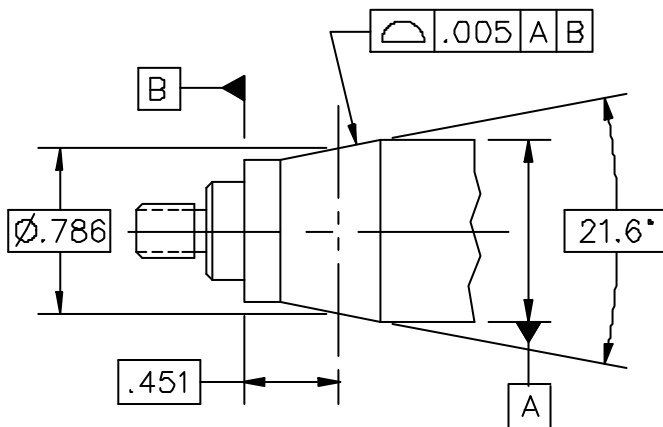


Figure 5-140 Profile tolerancing to control a combination of characteristics

5.13.11.1 With Positional Tolerancing for Bounded Features

Profile tolerancing can be teamed with positional tolerancing to control the orientation and location of bounded features having opposing elements that partly or completely enclose a space. See section 5.11.6.3.

5.13.12 Patterns of Profiled Features

The principles explained in sections 5.11.7 through 5.11.7.5 for controlling patterns of features of size can be extended to patterns of profiled features. Rather than a framework of Level 4 virtual condition boundaries, a profile tolerance applied to a feature pattern establishes a framework of multiple profile tolerance zones. Within this framework, the orientation and location of all the zones are fixed relative to one another according to the basic dimensions expressed on the drawing.

5.13.12.1 Single-Segment Feature Control Frame

Where feature “size,” form, orientation, location, and feature-to-feature spacing can all share a single tolerance value, a single-segment profile feature control frame is recommended. Fig. 5-141 shows a pattern of three mounting feet controlled for coplanarity. All points on all three feet shall be contained between a pair of parallel plane boundaries. This effectively controls the flatness of each foot as well as the coplanarity of all three together to prevent rocking. (A flatness tolerance would apply to each foot only on an individual basis.)

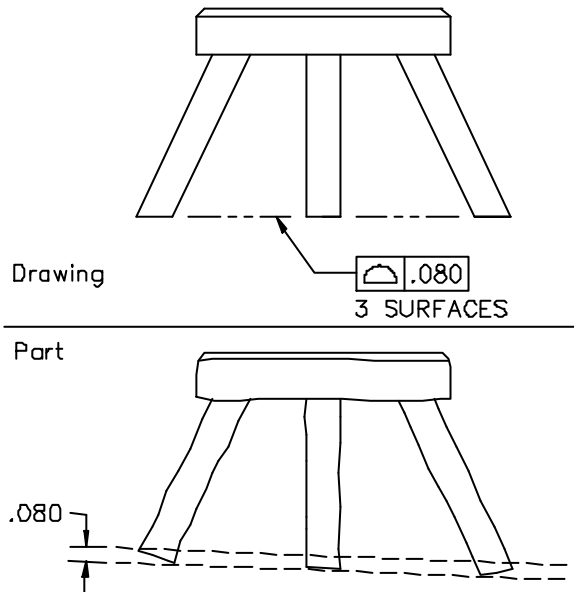


Figure 5-141 Profile tolerance to control coplanarity of three feet

5.13.12.2 Composite Feature Control Frame

A composite feature control frame can specify separate tolerances for overall pattern location and spacing. The few differences in symbology between composite positional and composite profile controls are obvious when comparing Fig. 5-119 with Fig. 5-142. The composite profile feature control frame contains a single entry of the “profile of a surface” symbol. The upper segment establishes a framework (PLTZF) of wider profile tolerance zones that are basically located and oriented relative to the referenced datums. The lower segment provides a specialized refinement within the constraints of the upper segment. It establishes a framework (FRTZF) of comparatively narrower zones that are basically oriented, but not located, relative to the referenced datums. All the rules given in section 5.11.7.3 governing datum references, tolerance values, and simultaneous requirements apply for composite profile tolerances as well.

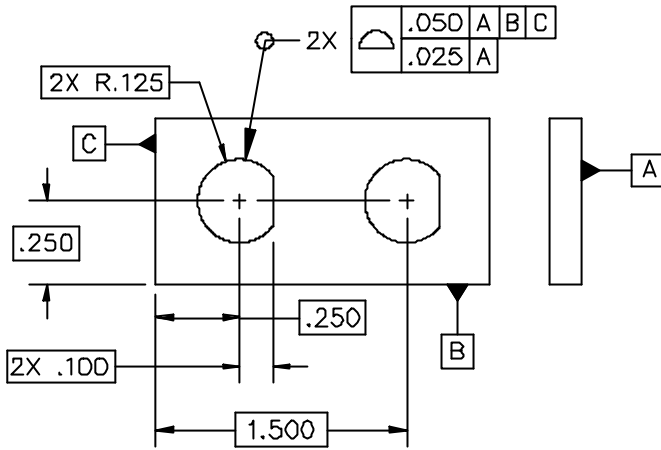


Figure 5-142 Composite profile for a pattern

5.13.12.3 Stacked Single-Segment Feature Control Frames

Where it's necessary to specify different location requirements for a pattern of profiled features relative to different planes of the DRF, stacked single-segment profile feature control frames may be applied as described in section 5.11.7.4. Each of the stacked feature control frames establishes a framework of profile tolerance zones that are basically located and oriented relative to the referenced datums. There is no explicit congruence requirement between the two frameworks. But, if features are to conform to both tolerances, tolerance zones must overlap to some extent. All the rules given in section 5.11.7.5 governing datum references, tolerance values, and simultaneous requirements apply for stacked single-segment profile tolerances as well.

5.13.12.4 Optional Level 2 Control

For features of size such as holes, size limits or tolerances and Rule #1 specify Level 2 form control. For profiled features, each profile tolerance zone provides a degree of Level 2 control (for feature "size" and form). However, where no pattern-controlling tolerance provides adequate Level 2 control, a separate profile tolerance may be added above and separated from the pattern-controlling frame(s). In Fig. 5-143,

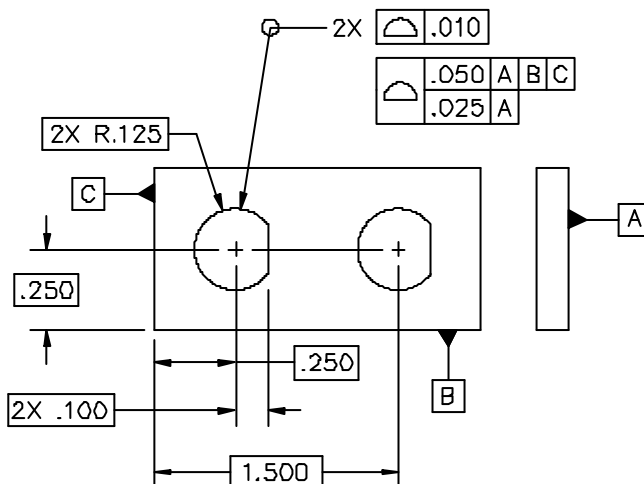


Figure 5-143 Composite profile tolerancing with separate Level 2 control

the profile tolerance of .010 establishes a discrete profile tolerance zone for each individual feature. As with the Level 2 size limit boundaries for holes in a pattern, there is no basic relationship between these Level 2 profile zones. They are all free to float relative to each other and relative to any datums. (Note: If the Level 2 feature control frame were added as a third segment of the composite control, the Level 2 profile zones would be basically related to each other.) Of course, the Level 2 tolerance must be less than any pattern-controlling tolerances to have any effect.

5.13.13 Composite Profile Tolerance for a Single Feature

For features of size, different characteristic symbols denote the four different levels of control. But, for irregularly shaped nonsize features, the same “profile of a surface” symbol is used for each level. In Fig. 5-144, for example, we want to refine a bounded feature’s orientation within the constraints of its locating tolerance. Simply stacking two single-segment profile feature control frames would be confusing. Many people would question whether the .020 tolerance controls location relative to datum B. Instead, we’ve borrowed from pattern control the composite feature control frame containing a single entry of the “profile of a surface” symbol. Though our “pattern” has only one feature, the tolerances mean the same.

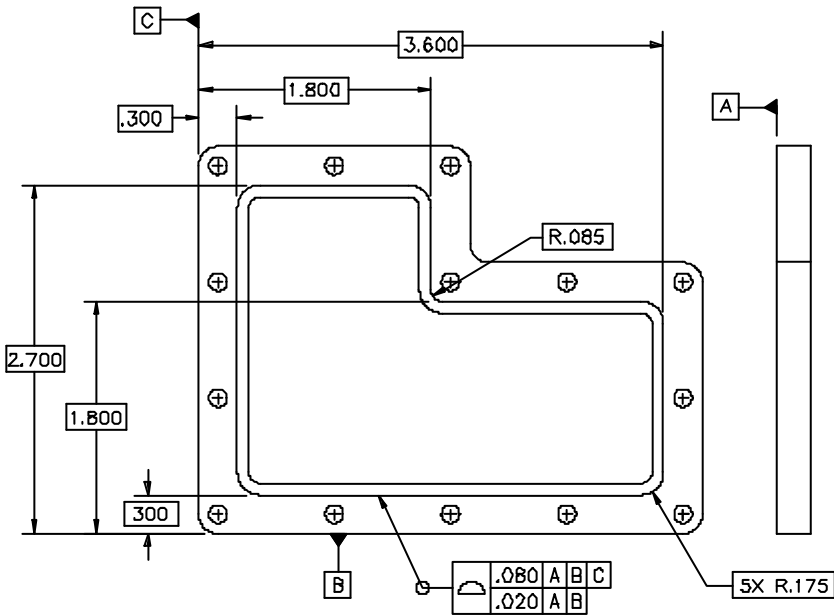


Figure 5-144 Composite profile tolerance for a single feature

In Fig. 5-144, the upper segment establishes a .080 wide profile tolerance zone basically located and oriented relative to the DRF A|B|C. The lower segment provides a specialized refinement within the constraints of the upper segment. It establishes a .020 wide zone basically oriented, but not located, relative to the DRF A|B. All the rules given in section 5.11.7.3 governing datum references, tolerance values, and simultaneous requirements apply for a composite profile “pattern of one.”

5.14 Symmetry Tolerance

Symmetry is the correspondence in size, contour, and arrangement of part surface elements on opposite sides of a plane, line, or point. We usually think of symmetry as the twofold mirror-image sort of balance

about a center plane shown in Fig. 5-145(a) and (b). There are other types as well. A three-lobe cam can have symmetry, both the obvious twofold kind about a plane as shown in Fig. 5-145(c), and a threefold kind about an axis as shown in Fig. 5-145(d). The pentagon shown in Fig. 5-145(e) has fivefold symmetry about an axis. GD&T's symmetry tolerances apply at the *lowest order of symmetry*—the lowest prime divisor of the number of sides, facets, blades, lobes, etc., that the feature is supposed to have. Thus, a 27-blade turbine would be controlled by threefold symmetry. For a hexagonal flange (six sides), twofold symmetry applies. By agreement, a nominally round shaft or sphere is subject to twofold symmetry as well.

5.14.1 How Does It Work?

The Math Standard describes in detail how symmetry tolerancing works. Generically, a symmetry tolerance prescribes that a datum plane or axis is extended all the way through the controlled feature. See Fig. 5-146. From any single point on that datum within the feature, vectors or *rays* perpendicular to the datum

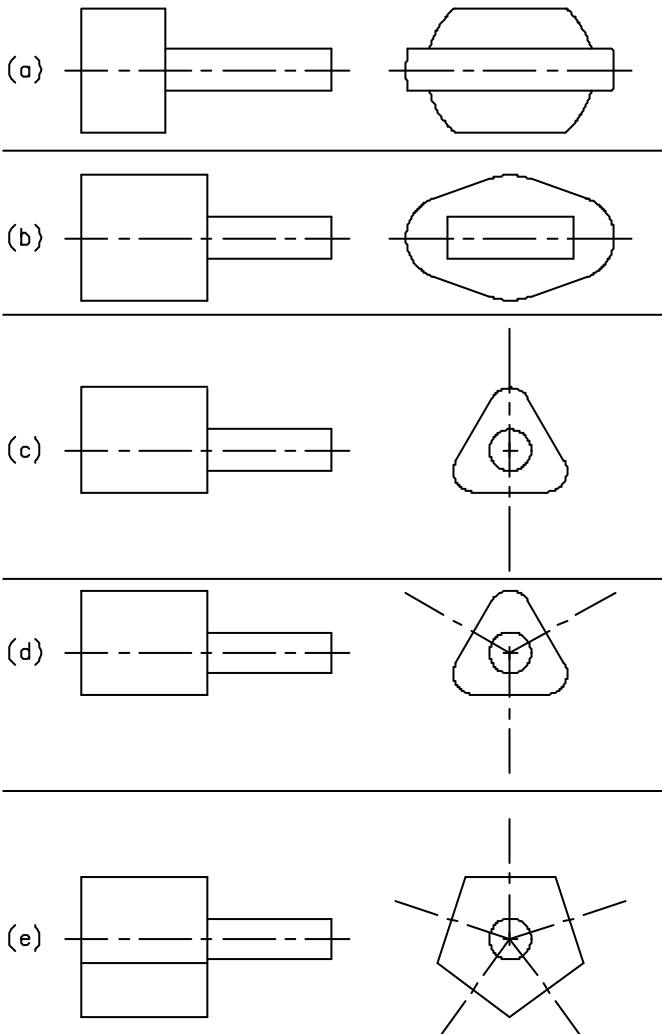


Figure 5-145 Types of symmetry

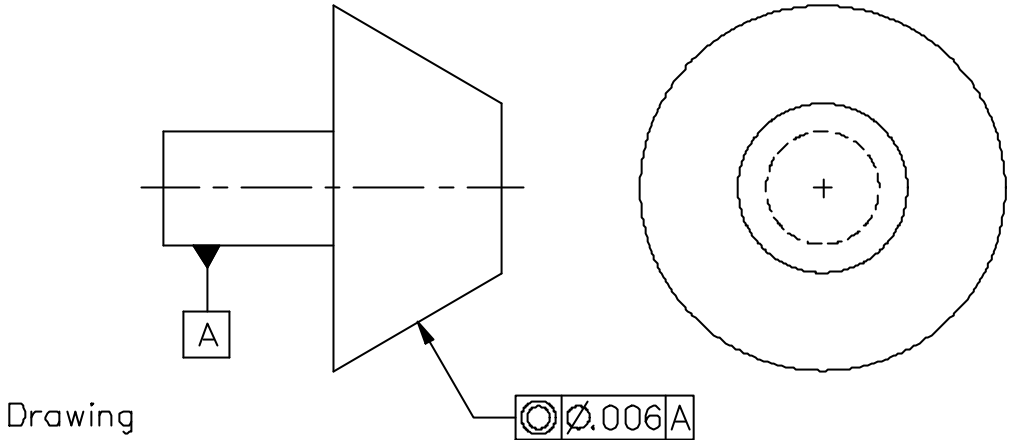


Figure 5-146 Symmetry construction rays

are projected to intersect the feature surface(s). For common twofold symmetry, two rays are projected, 180° apart. From those intersection points, a median point (centroid) is constructed. This median point shall lie within a tolerance zone that is uniformly distributed about the datum.

If one of the construction rays hits a small dent in the surface, but an opposite ray intersects a uniform portion of the surface, the median point might lie outside the tolerance zone. Thus, symmetry tolerancing demands that any local “low spot” in the feature surface be countered by another “low spot” opposite. Similarly, any “high spot” must have a corresponding “high spot” opposite it. Symmetry tolerancing primarily prevents “lopsidedness.”

As you can imagine, inspecting a symmetry tolerance is no simple matter. Generally, a CMM with advanced software or a dedicated machine with a precision spindle should be used. For an entire feature to conform to its symmetry tolerance, all median points shall conform, for every possible ray pattern, for every possible origin point on the datum plane or axis within the feature. Although it’s impossible to verify infinitely many median points, a sufficient sample (perhaps dozens or hundreds) should be constructed and evaluated.

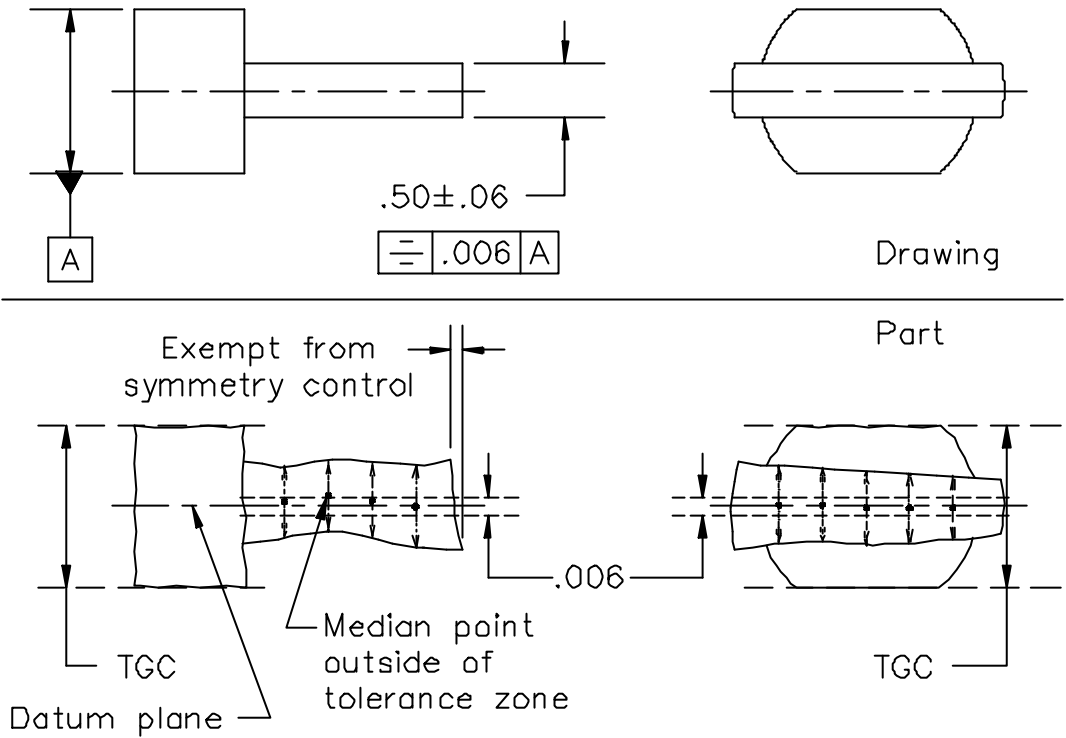


Figure 5-147 Symmetry tolerance about a datum plane

At the ends of every actual bore or shaft, and at the edges of every slot or tab, for example, the terminating faces will not be perfectly perpendicular to the symmetry datum. Though one ray might intersect a part surface at the extreme edge, the other ray(s) could just miss and shoot off into the air. This also happens at any cross-hole, flat, keyseat, or other interruption along the controlled feature(s). Obviously then, unopposed points on the surface(s), as depicted in Fig. 5-147, are exempt from symmetry control. Otherwise, it would be impossible for any feature to conform.

5.14.2 How to Apply It

A symmetry tolerance is specified using a feature control frame displaying the characteristic symbol for either “concentricity” (two concentric circles) or “symmetry about a plane” (three stacked horizontal bars). See Figs. 5-146 through 5-148. The feature control frame includes the symmetry tolerance value followed by one, two, or three datum references.

There’s no practical interaction between a feature’s size and the acceptable magnitude of lopsidedness. Thus, material condition modifier symbols, MMC and LMC, are prohibited for all symmetry tolerances and their datum references.

5.14.3 Datums for Symmetry Control

Symmetry control requires a DRF. A primary datum plane or axis usually arrests the three or four degrees of freedom needed for symmetry control. All datum references shall be RFS.

5.14.4 Concentricity Tolerance

Concentricity tolerancing of a revolute, as illustrated in Fig. 5-146, is one of the most common applications of symmetry tolerancing. It's specified by a feature control frame containing the "concentricity" symbol. In this special symmetry case, the datum is an axis. There are two rays 180° apart (colinear) perpendicular to the datum axis. The rays intersect the feature surface at two diametrically opposed points. The midpoint between those two surface points shall lie within a cylindrical tolerance zone coaxial to the datum and having a diameter equal to the concentricity tolerance value.

At each cross-sectional slice, the revolving rays generate a locus of distinct midpoints. As the rays sweep the length of the controlled feature, these 2-D loci of midpoints stack together, forming a 3-D "wormlike" locus of midpoints. The entire locus shall be contained within the concentricity tolerance cylinder. Don't confuse this 3-D locus with the 1D derived median line defined in section 5.6.4.2.

5.14.4.1 Concentricity Tolerance for Multifold Symmetry about a Datum Axis

The explanation of concentricity in Y14.5 is somewhat abstruse because it's also meant to support multifold symmetry about an axis. Any prime number of rays can be projected perpendicular from the datum axis, provided they are coplanar with equal angular spacing. For the 3-lobe cam in Fig. 5-148, there are three rays, 120° apart. A 25-blade impeller would require five rays spaced 72° apart, etc.

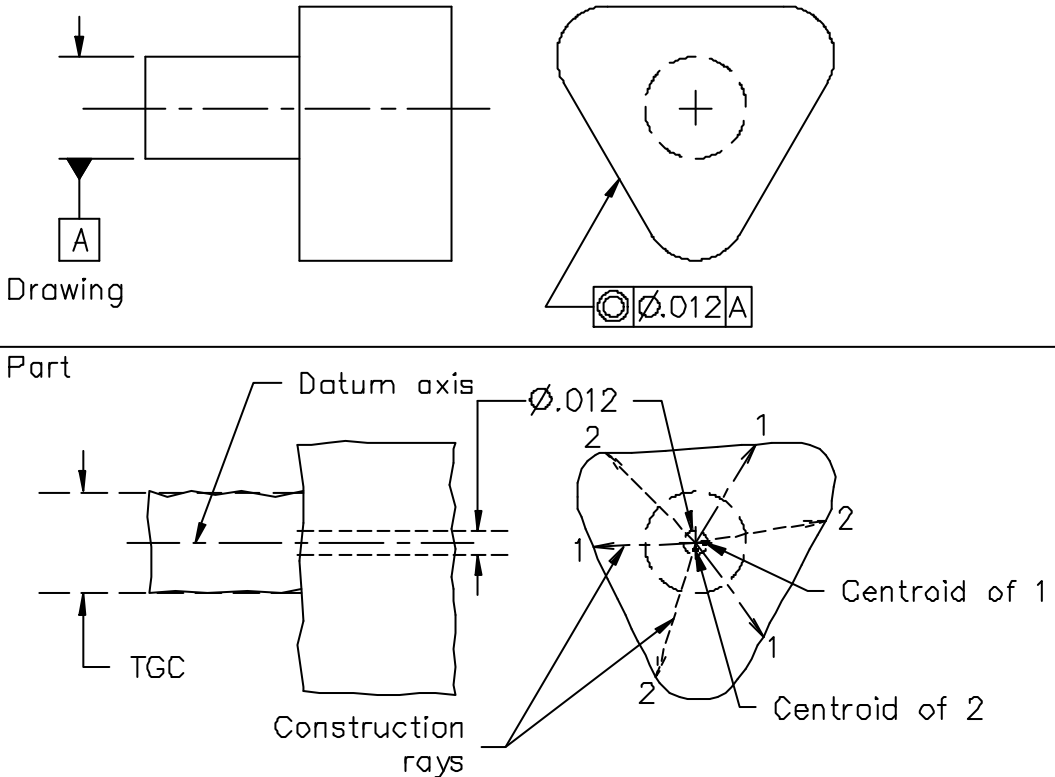


Figure 5-148 Multifold concentricity tolerance on a cam

From the multiple intersection points, a centroid is then constructed and checked for containment within the tolerance zone. The standards don't specify how to derive the centroid, but we recommend the Minimum Radial Separation (MRS) method described in ANSI B89.3.1-1972. Obviously, verification is well beyond the capability of an inspector using multiple indicators and a calculator. Notice that as the rays are revolved about the datum axis, they intersect the surface(s) at vastly different distances from center. Nevertheless, if the part is truly symmetrical, the centroid still remains within the tolerance cylinder.

5.14.4.2 Concentricity Tolerance about a Datum Point

The “concentricity” symbol can also be used to specify twofold or multifold symmetry about a datum point. This could apply to a sphere, tetrahedron, dodecahedron, etc. In all cases, the basic geometry defines the symmetry rays, and centroids are constructed and evaluated. The tolerance value is preceded by the symbol $S\emptyset$, specifying a spherical tolerance zone.

5.14.5 Symmetry Tolerance about a Datum Plane

The other symmetry symbol, having three horizontal bars, designates symmetry about a plane. Y14.5 calls this application **Symmetry Tolerancing to Control the Median Points of Opposed or Correspondingly-Located Elements of Features**. Despite this ungainly and nondescriptive label, symmetry tolerancing about a plane works just like concentricity except for two differences: the symmetry datum is a plane instead of an axis; and the symmetry can only be twofold. See Fig. 5-147. From any point on the datum plane between the controlled surfaces, two rays are projected perpendicular to the datum, 180° apart (colinear). The rays intersect the surfaces on either side of the datum. The midpoint between those two surface points shall be contained between two parallel planes, separated by a distance equal to the symmetry tolerance value. The two tolerance zone planes are equally disposed about (thus, parallel to) the datum plane. All midpoints shall conform for every possible origin point on the datum plane between the controlled surfaces.

As the rays sweep, they generate a locus of midpoints subtly different from the derived median plane defined in section 5.6.4.2. The symmetry rays are perpendicular to the datum plane, while the derived median plane's construction lines are perpendicular to the feature's own center plane. It's not clear why the methods differ or whether the difference is ever significant.

Symmetry tolerancing about a plane does not limit feature size, surface flatness, parallelism, or straightness of surface line elements. Again, the objective is that the part's mass be equally distributed about the datum. Although a symmetry or concentricity tolerance provides little or no form control, it always accompanies a size dimension that provides some restriction on form deviation according to Rule #1.

5.14.6 Symmetry Tolerancing of Yore (Past Practice)

Until the 1994 edition, Y14.5 described concentricity tolerancing as an “axis” control, restraining a separate “axis” at each cross-section of the controlled feature. A definition was not provided for *axis*, nor was there any explanation of how a two-dimensional imperfect shape (a circular cross-section) could even have such a thing. As soon as the Y14.5 Subcommittee defined the term *feature axis*, it realized two things about the feature axis: it's what ordinary positional tolerance RFS controls, and it has nothing to do with lopsidedness (balance). From there, symmetry rays, median points, and worms evolved.

The “Symmetry Tolerance” of the 1973 edition was exactly the same as positional tolerance applied to a noncylindrical feature RFS. (See the note at the bottom of Fig. 140 in that edition.) The three-horizontal bars symbol was simply shorthand, saving draftsmen from having to draw circle-S symbols. Partly because of its redundancy, the “symmetry tolerance” symbol was cut from the 1982 edition.

5.14.7 When Do We Use a Symmetry Tolerance?

Under any symmetry tolerance, a surface element on one “side” of the datum can “do anything it wants” just as long as the opposing element(s) mirrors it. This would appear to be useful for a rotating part that must be dynamically balanced. However, there are few such assemblies where GD&T alone can adequately control balance. More often, the assembly includes setscrews, keyseats, welds, or other attachments that entail a balancing operation after assembly. And ironically, a centerless ground shaft might have near-perfect dynamic balance, yet fail the concentricity tolerance because its out-of-roundness is 3-lobed.

FAQ: *Could a note be added to modify the concentricity tolerance for a cylinder to 3-fold symmetry?*

A: Sure.

FAQ: *Can I use a symmetry tolerance if the feature to be controlled is offset (not coaxial or coplanar) from the datum feature?*

A: Nothing in the standard prohibits that, either. Be sure to add a basic dimension to specify the offset. You may also need two or even three datum references.

FAQ: *Since a runout tolerance includes concentricity control and is easier to check, wouldn't it save money to replace every concentricity tolerance with an equal runout tolerance? We wouldn't need concentricity at all.*

A: Though that is the policy at many companies, there's another way to look at it. Let's consider a design where significant out-of-roundness can be tolerated as long as it's symmetrical. A concentricity tolerance is carefully chosen. We can still use runout's FIM method to inspect a batch of parts. Of those conforming to the concentricity tolerance, all or most parts will pass the FIM test and be accepted quickly and cheaply. Those few parts that fail the FIM inspection may be re-inspected using the formal concentricity method. The concentricity check is more elaborate and expensive than the simple FIM method, but also more forgiving, and would likely accept many of the suspect parts. Alternatively, management may decide it's cheaper to reject the suspect parts without further inspection and to replace them. The waste is calculated and certainly no worse than if the well-conceived concentricity tolerance had been arbitrarily converted to a runout tolerance. The difference is this: If the suspect parts are truly usable, the more forgiving concentricity tolerance offers a chance to save them.

5.15 Combining Feature Control Frames

In section 5.6, we defined four different levels of GD&T control for features of size. In fact, the four levels apply for every feature.

Level 1: 2-D form at individual cross sections

Level 2: Adds third dimension for overall form control

Level 3: Adds orientation control

Level 4: Adds location control

For every feature of every part, a designer must consider all the design requirements, including function, strength, assemblability, life expectancy, manufacturability, verification, safety, and appearance. The designer must then adequately control each part feature, regardless of its type, at each applicable level of control, to assure satisfaction of all design requirements. For a nonsize feature, a single “profile”

or “radius” tolerance will often suffice. Likewise, a feature of size might require nothing more than size limits and a single-segment positional tolerance.

In addition to the design requirements listed, many companies include cost considerations. In cost-sensitive designs, this often means maximizing a feature’s tolerance at each level of control. The designer must understand the controls imposed at each level by a given tolerance. For example, where a Level 4 (location) tolerance has been maximized, it might not adequately restrict orientation. Thus, a separate lesser Level 3 (orientation) tolerance must be added. Even that tolerance, if properly maximized, might not adequately control 3-D form, etc. That’s why it’s not uncommon to see two, or even three feature control frames stacked for one feature, each maximizing the tolerance at a different level.

5.16 “Instant” GD&T

Y14.5 supports several general quasi-GD&T practices as alternatives to the more rigorous methods we’ve covered. To be fair, they’re older practices that evolved as enhancements to classical tolerancing methods. However, despite the refinement and proliferation of more formal methods, the quasi-GD&T practices are slow to die and you’ll still see them used on drawings. Designers might be tempted to use one or two of them to save time, energy, and plotter ink. We’ll explain why, for each such practice, we feel that’s false economy.

5.16.1 The “Dimension Origin” Symbol

The “dimension origin” symbol, shown in Fig. 5-149, is not associated with any datum feature or any feature control frame. It’s meant to **indicate that a dimension between two features shall originate from one of these features and not the other**. The specified treatment for the originating surface is exactly the same as if it were a primary datum feature. But for some unfathomable reason, Y14.5 adds, **This concept does not establish a datum reference frame...** The treatment for the other surface is exactly the same as if it were controlled with a profile of a surface tolerance. We explained in section 5.10.8 why this practice is meaningless for many angle dimensions. Prevent confusion; instead of the “dimension origin” symbol, use a proper profile or positional tolerance.

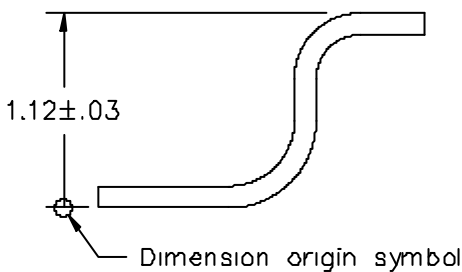


Figure 5-149 Dimension origin symbol

5.16.2 General Note to Establish Basic Dimensions

Instead of drawing the “basic dimension” frame around each basic dimension, a designer may designate dimensions as basic by **specifying on the drawing (or in a document referenced on the drawing) the general note: UNTOLERANCED DIMENSIONS LOCATING TRUE POSITION ARE BASIC**. This could be extremely confusing where other untoleranced dimensions are not basic, but instead default to tolerances expressed in a tolerance block. Basic dimensions for angularity and profile tolerances, datum targets, and more would still have to be framed unless the note were modified. Either way, the savings in ink are negligible compared to the confusion created. Just draw the frames.

5.16.3 General Note in Lieu of Feature Control Frames

Y14.5 states that linear and angular dimensions may be related to a DRF without drawing a feature control frame for each feature. [T]he desired order of precedence may be indicated by a note such as: **UNLESS OTHERWISE SPECIFIED, DIMENSIONS ARE RELATED TO DATUM A (PRIMARY), DATUM B (SECONDARY), AND DATUM C (TERTIARY)**. However, applicable datum references shall be included in any feature control frames used. It's not clear whether or not this practice establishes virtual condition boundaries or central tolerance zones for the affected features, and if so, of what sizes and shapes. As we explained in section 5.10.8, for some angle dimensions a wedge-shaped zone is absurd.

The hat trick of “instant” GD&T is to combine the above two “instant basic dimensions” and “instant datum references” notes with an “instant feature control” note, such as **PERFECT ORIENTATION (or COAXIALITY or LOCATION OF SYMMETRICAL FEATURES) AT MMC REQUIRED FOR RELATED FEATURES**. This should somehow provide cylindrical or parallel-plane tolerance zones equivalent to zero positional or zero orientation tolerances at MMC for all “related features” of size.

Throughout this chapter, we've emphasized how important it is for designers to consider carefully and individually each feature to maximize manufacturing tolerances. Certainly, troweling on GD&T with general notes does not require such consideration, although, neither does the practice preclude it. And while there may be drawings that would benefit from consolidation and unification of feature controls, we prefer to see individual, complete, and well-thought-out feature control frames.

5.17 The Future of GD&T

GD&T's destiny is clearly hitched to that of manufacturing technology. You wouldn't expect to go below deck on *Star Trek's USS Enterprise* and find a machine room with a small engine lathe and a Bridgeport mill. You might find instead some mind-bogglingly precise process that somehow causes a replacement “Support, Dilithium Crystal” to just “materialize” out of a dust cloud or a slurry. Would Scotty need to measure such a part?

Right now, the rapid-prototyping industry is making money with technology that's only a couple of generations away from being able to “materialize” high-strength parts in just that way. If such a process were capable of producing parts having precision at least an order of magnitude more than what's needed, the practice of measuring parts would indeed become obsolete, as would the language for specifying dimensional tolerances. Parts might instead be specified with only the basic geometry (CAD model) and a process capability requirement.

History teaches us that new technology comes faster than we ever expected. Regardless of our apprehension about that, history also reveals that old technology lingers on longer than we expected. In fact, the better the technology, the slower it dies. An excellent example is the audio Compact Cassette, introduced to the world by Philips in 1963. Even though Compact Discs have been available in every music store since 1983, about one-fourth of all recorded music is still sold on cassette tapes. We can likewise expect material removal processes and some form of GD&T to enjoy widespread use for at least another two decades, regardless of new technology.

In its current form, GD&T reflects its heritage as much as its aspirations. It evolved in relatively small increments from widespread, time-tested, and work-hardened practices. As great as it is, GD&T still has much room for improvement. There have been countless proposals to revamp it, ranging from moderate streamlining to total replacement. Don't suppose for one second that all such schemes have been hare-brained. One plan, for example, would define part geometry just as a coordinate measuring machine sees it—vectorially. Such a system could expedite automated inspection, and be simpler to learn. But does it preclude measurements with simple tools and disenfranchise manufacturers not having access to a CMM? What about training? Will everyone have to be fluent in two totally different dimensioning and tolerancing languages?

As of this writing, the international community is much more receptive to radical change than the US. Europe is a hotbed of revolutionary thought; any daring new schemes will likely surface there first. Americans can no longer play isolationism as they could decades ago. Many US companies are engaged in multinational deals where a common international drawing standard is mandatory. Those companies are scarcely able to insist that standard be Y14.5. There are always comments about “the tail wagging the dog,” but the US delegation remains very influential in ISO TC 213 activity pertaining to GD&T. Thus, in the international standards community, it’s never quite clear where the tail ends and the dog begins.

Meanwhile, Americans are always looking for ways to simplify GD&T, to make their own Y14.5 Standard thinner (or at least to slow its weight gain). You needn’t study GD&T long to realize that a few characteristic symbols are capable of controlling many more attributes than some others control. For example, a surface profile tolerance can replace an equal flatness tolerance. Why do we need the “flatness” symbol? And if the only difference between parallelism, perpendicularity, and angularity is the basic angle invoked, why do we need three different orientation symbols? In fact, couldn’t the profile of a surface characteristic be modified slightly to control orientation?

These are all valid arguments, and taken to the next logical step, GD&T could be consolidated down to perhaps four characteristic symbols. And following in the same logic, down to three or two symbols, then down to one symbol. For that matter, not even one symbol would be needed if it were understood that each feature has default tolerance boundaries according to its type. The document that defines such tolerance zones might have only thirty pages. This would be GD&T at its leanest and meanest! OK, so why don’t we do it?

That argument assumes that the complexity of a dimensioning and tolerancing system is proportional to the number of symbols used. Imagine if English had only 100 words, but the meanings of those words change depending on the context and the facial expression of the speaker. Would that be simpler? Easier to learn? No, because instead of learning words, a novice would have to learn all the rules and meanings for each word just to say “Hello.” There’s a lot to be gained from simplification, but there’s also a huge cost.

In fact, GD&T’s evolution could be described as a gradual shift from simplicity toward flexibility. As users become more numerous and more sophisticated, they request that standards add coverage for increasingly complex and esoteric applications. Consequently, most issues faced by the Y14.5 committee boil down to a struggle to balance simplicity with flexibility.

It’s impossible to predict accurately where GD&T is headed, but it seems reasonable to expect the Y14.5 committee will continue to fine-tune a system that is rather highly developed, mature, and in widespread international use. Radical changes cannot be ruled out, but they would likely follow ISO activity. Be assured, GD&T’s custodial committees deeply contemplate the future of dimensioning and tolerancing.

Standards committee work is an eye-opening experience. Each volunteer meets dozens of colleagues representing every sector of the industry, from the mainstream Fortune 500 giants to the tiniest outpost ma-and-pa machine shops. GD&T belongs equally to all these constituents. Often, what seemed a brilliant inspiration to one volunteer withers under the hot light of committee scrutiny. That doesn’t mean that nothing can get through committee; it means there are very few clearly superior and fresh ideas under the sun. Perhaps, though, you’ve got one. If so, we encourage you to pass it along to this address.

The American Society of Mechanical Engineers
Attention: Secretary, Y14 Main Committee
345 East 47th Street
New York, NY 10017

5.18 References

1. The American Society of Mechanical Engineers. 1972. *ANSI B89.3.1-1972. Measurement of Out-Of-Roundness*. New York, New York: The American Society of Mechanical Engineers.
2. The American Society of Mechanical Engineers. 1972. *ANSI B4.1-1967. Preferred Limits and Fits for Cylindrical Parts*. New York, New York: The American Society of Mechanical Engineers.
3. The American Society of Mechanical Engineers. 1978. *ANSI B4.2-1978. Preferred Metric Limits and Fits*. New York, New York: The American Society of Mechanical Engineers.
4. The American Society of Mechanical Engineers. 1982. *ANSI Y14.5M-1982, Dimensioning and Tolerancing*. New York, New York: The American Society of Mechanical Engineers.
5. The American Society of Mechanical Engineers. 1995. *ASME Y14.5M-1994, Dimensioning and Tolerancing*. New York, New York: The American Society of Mechanical Engineers.
6. The American Society of Mechanical Engineers. 1994. *ASME Y14.5.1-Mathematical Definition of Dimensioning and Tolerancing Principles*. New York, New York: The American Society of Mechanical Engineers.
7. International Standards Organization. 1985. *ISO8015. Technical Drawings -- Fundamental Tolerancing Principle*. International Standards Organization: Switzerland.