THIRTEENTH ANNUAL
BiTS 2012
Burn-in & Test Strategies Workshop

TechTalk

ARCHIVE 2012

GEOMETRIC DIMENSIONING AND TOLERANCING FOR BURN-IN AND TEST PROFESSIONALS

by

Thomas Allsup
President
Anida Technologies

Building on his 2006 *"Geometric Dimensioning and Tolerancing" primer tutorial, which regularly tops the BiTS charts as most frequently downloaded archived presentation, Thomas Allsup returns to further enlighten BiTS Workshop attendees with an exciting new look at GD&T.*

Mr. Allsup will introduce and explain the symbolic based language of GD&T and provide a handy tool – a periodic table of GD&T symbols. Attendees who are new to GD&T will receive a solid foundation of the nomenclature and their correct usage while experienced designers will have a chance to discuss many practical applications as the topic relates to socket tolerancing.

ABSTRACT

Geometric Dimensioning and Tolerances (GD&T) is the common language used to describe the allowable variances of manufactured feature sizes, shapes, and locations beyond that which can be controlled by regular rectilinear and angular dimensions and tolerances.

Semiconductor component and socket manufacturer drawings both use GD&T to insure their respective components fit and function mechanically together.

This new tutorial is presented in three sections: Section one provides a highly abbreviated "How to Spell GD&T" review of the fundamentals of GD&T, Section two explains the first changes to the ASME Y14.5 standard in fifteen years, particularly where those changes impact semiconductor professionals, and Section three contains a series of public domain semiconductor component drawings that will be carefully dissected to explain how GD&T was used correctly and incorrectly.

Geometric Dimensioning and Tolerancing for Burn-In and Test Professionals

Thomas Allsup

2012 BiTS Workshop
March 4 - 7, 2012

anida technologies

Description

Building on the "How to Spell GD&T" tutorial, we will review the fundamentals of the GD&T, explain the changes in the new ASME Y14.5-2009 standard, and explore some actual semiconductor GD&T drawing examples.

Abstract

Geometric Dimensioning and Tolerances (GD&T) is the common language used to describe the allowable variances of manufactured feature sizes, shapes, and locations beyond that which can be controlled by regular rectilinear and angular dimensions and tolerances. Semiconductor component and socket manufacturer drawings both use GD&T to insure their respective components fit and function mechanically together. The "How to Spell GD&T" tutorial previously presented at BiTS provided a detailed primer of how to read GD&T symbols on drawings and provides an introduction to this tutorial. This new tutorial is presented in three sections: Section one provides a highly abbreviated "How to Spell GD&T" review of the fundamentals of GD&T, Section two explains the first changes to the ASME Y14.5 standard in fifteen years particularly where those changes impact semiconductor professionals, and Section three contains a series of public domain semiconductor component drawings that will be carefully dissected to explain how GD&T was used correctly and incorrectly.

George Carlin

American Comedian 1937-2008

- Hosted first Saturday Night Live episode on October 11, 1975
  - The producers told George if he did a good job they would invite him back.
- In season 10, he hosted Saturday Night Live for the second time on November 10, 1984
  - George commented during this monologue that he was happy they thought he did a good job.

Thomas Allsup

- I am not a GD&T expert but I did stay at a La Quinta last night.
- Thomas received his BSME from Oklahoma State University - Go Pokes!
- In 1990, Thomas got his MSME from UT Arlington.
- Thomas ended his PhD in Mechanical Engineering from UTA as an All-But-Dissertation in 1994.
- Thomas is has also ABD in General Engineering PhD online from Kennedy Western University.
- Thomas has spent many years as a design engineer in consumer products, semiconductor devices, burn-in sockets and many optoelectronic devices.
- I learned GD&T out of necessity not desire…

Agenda

- Hour 1
  - Review GD&T Fundamentals
- Hour 2
  - What's New in the ASME Y14.5-2009 Standard
- Hour 3
  - GD&T Semiconductor Drawing Examples

ASME Y14M-1994
The Standard

ISO 1101:2004

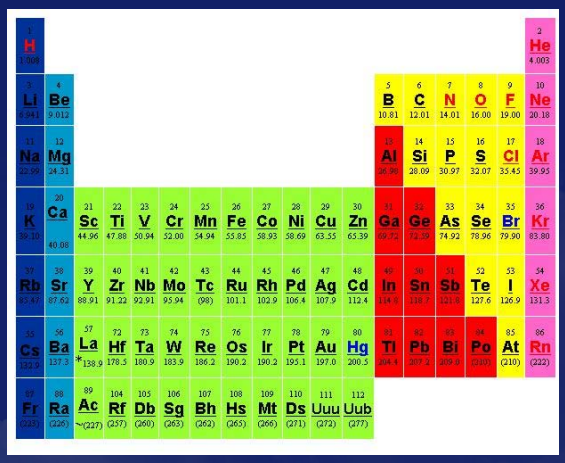
- *Official Title: Geometrical Product Specifications (GPS) -- Geometrical tolerancing -- Tolerances of form, orientation, location and run-out*
- Every revision of the American standard has brought more harmony with the ISO equivalent.
  - And vice versa as well.
- Everyone has heard of ISO9000 but little is said about ISO1101 because it is very close to ASME Y14.5M-1994.

FOPRL?

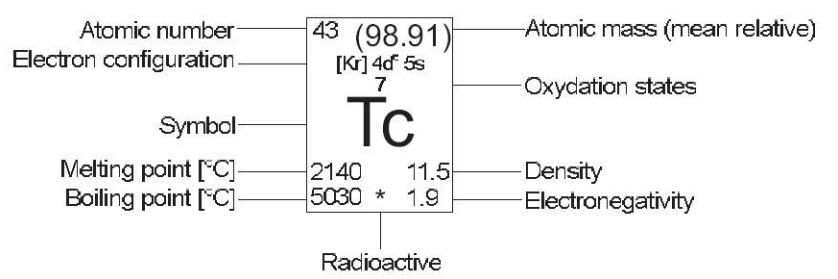
- This section starts looking at the 14 GD&T control symbols.
- We will use a technique that I developed based on a concept similar to the Periodic Table of Elements in Chemistry.

Periodic Table of Elements

- Remember high school chemistry?
- Columns and rows both group elements.
- Each box has information about just that element.



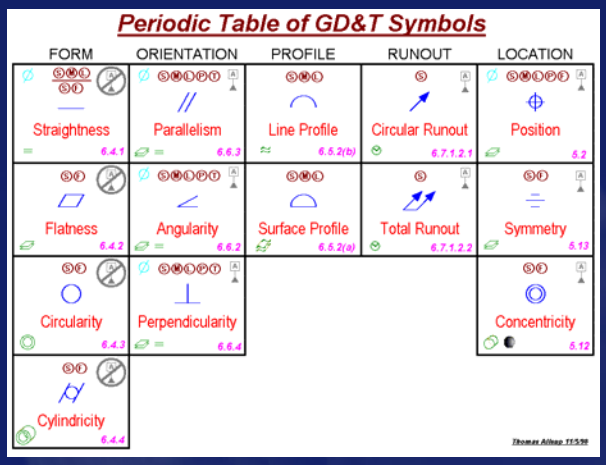
Element Information



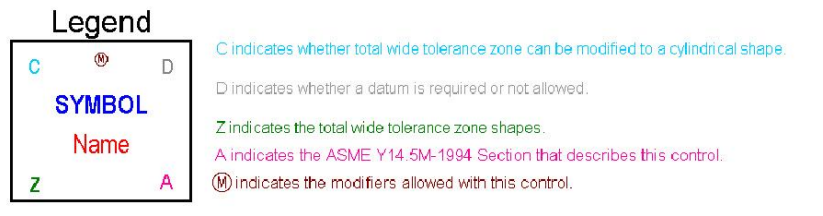
In each element, the chart makers place information so someone "skilled in the arts" will have all the information they need to work with the element.

Periodic Table of GD&T

- Columns and rows both group controls.
- Each box has information about just that control.



FOPRL Legend



In each control, we place information so someone "skilled in the arts" will have all the information they need to work with the control.

Five Kinds of Geometric Control

- All of these controls act just like they sound like:
  - Form
  - Orientation
  - Profile
  - Runout
  - Location
- This is how we get F O P R L.

Form Controls

- First kind of geometric control we look at is the simplest: Form.
- Form control is just like it sounds, they control the acceptable variance in the shape of a feature.
- There are four kinds of form control - I remember them by saying 2D-3D-2D-3D - let's see why.

Form
Straightness

- What geometry is two dimensional and linear?
  - Answer: A straight line
- Let's call the control of a straight line, the STRAIGHTNESS.
- The icon to represent a straight line would be a line —

Form
Flatness

- What geometry is three dimensional and linear?
  - Answer: A flat plane
- Let's call the control of a flat plane, the FLATNESS.
- The icon to represent a flat plane would be the shape of a plane 

Form
Circularity

- What geometry is two dimensional and rotary in nature?
  - Answer: A circle
- Let's call the control of a circle, the CIRCULARITY.
- The icon to represent a circle would be a circle 

Form
Cylindricity

- What geometry is three dimensional and rotary in nature?
  - Answer: A cylinder (Don't guess sphere)
- Let's call the control of a cylinder, the CYLINDRICITY.
- The icon to represent a cylinder would be a made up symbol 
  - More on why this makes sense later.

FOPRL
In Process

FORM	ORIENTATION	PROFILE	RUNOUT	LOCATION
STRAIGHTNESS –				
FLATNESS 				
CIRCULARITY ○				
CYLINDRICITY 				

We can cheat and see that there must be three orientation controls.

Orientation Controls

- Second kind of geometric control we look at is the next simplest: Orientation.
- Orientation control is just like it sounds, they control the acceptable variance in the direction of a feature.
- There are three kinds of orientation control - I remember them thinking about my arm swing from horizontal to vertical.

Aerobics Time!

Orientation Angles

- When your arm is horizontal, what is the angle it forms with the ground?
  - Zero, it is parallel with the ground.
- As you rotate you arm, the angle is some arbitrary angle.
- When your arm in pointing up, what is the angle it forms with the ground?
  - Ninety degrees, it is perpendicular with the ground.

Since the ground is our reference, let's call it a datum. There'll be much more on these later.

Orientation
Parallelism

- What orientation is at zero degrees from the reference?
  - Parallelism
- The icon to represent when things should be parallel would be a two parallel lines //

Cylindricity Revisited

- Two dimensional rotational form control Circularity ○
  plus Parallel orientation of the sides // equals three dimensional rotational form control Cylindricity ⌭
- It kind of all makes sense, doesn't it?

Orientation
Angularity

- What orientation is any angle?
  - Answer: Angularity
- The icon to represent when two things are at an arbitrary angle looks like an angle 

Orientation
Perpendicularity

- What orientation is at ninety degrees from the reference?
  - Answer: Perpendicularity
- The icon to represent when things should be perpendicular would be something that looks like 

FOPRL
In Process

FORM	ORIENTATION	PROFILE	RUNOUT	LOCATION
STRAIGHTNESS –	PARALLELISM //			
FLATNESS ▱	ANGULARITY ∠			
CIRCULARITY ○	PERPENDICULARITY ⊥			
CYLINDRICITY ⌭				

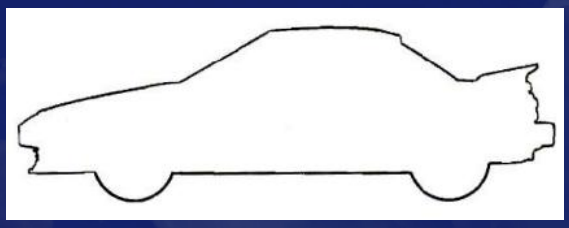
We can cheat and see that there must be two profile controls.

Profile Controls

- Now let's start looking at the complicated geometric controls: Profile Controls.
- Profile control is just like it sounds, they control the acceptable variance in the profile of a feature.
- There are two kinds of profile control - I remember them by thinking 2D-3D.

Complex Profiles?

- Most people think of complex shapes like the contour of a car but the profile can be as simple as a line or circle.
- Inspectors love checking profile controlled features because they can place a clear overlay on the feature and simply say pass or fail.



Profile Control
Line

- We call the two dimensional profile control "*line profile*" or "*profile of a line*".
  - The line in question can be any shape, or any number of lines, arcs, or splines.
- The icon used to represent a line profile is ⌒

Don't turn that frown upside down!

Profile Control
Surface

- We call the three dimensional profile control "*surface profile*" or "*profile of a surface*".
  - The surface in question can be any shape, or any number of planes or surfaces.
- The icon used to represent a surface profile is ⌓

FOPRL
In Process

FORM	ORIENTATION	PROFILE	RUNOUT	LOCATION
STRAIGHTNESS ⏤	PARALLELISM //	LINE PROFILE ⌒		
FLATNESS ⏥	ANGULARITY ∠	SURFACE PROFILE ⌓		
CIRCULARITY ○	PERPENDICULARITY ⊥			
CYLINDRICITY ⌭				

We can cheat and see that there must be two runout controls as well.

Runout Controls

- Let's continue looking at the complicated geometric controls: Runout Controls.
- Runout control is just like it sounds, they control the acceptable variance in a revolved feature.
- There are two kinds of profile control - I remember them by thinking 2D-3D.

You getting tired of 2D-3D?
Last time I use it, promise.

Runout Control
Circular

- We call the two dimensional runout control "*circular runout*".
- The icon used to represent circular runout is ↗
- Note this symbol looks like the needle from a dial indicator and that's exactly how we measure it.

Runout Control
Total

- We call the three dimensional runout control "*total runout*".
- The icon used to represent circular runout is 
- This is a perfect time to talk about what we call the rotation of the dial indicator over a feature.

FOPRL
In Process

FORM	ORIENTATION	PROFILE	RUNOUT	LOCATION
STRAIGHTNESS	PARALLELISM 	LINE PROFILE 	CIRCULAR RUNOUT 	
FLATNESS 	ANGULARITY 	SURFACE PROFILE 	TOTAL RUNOUT 	
CIRCULARITY 	PERPENDICULARITY 			
CYLINDRICITY 				

There's a reason that we put profile and runout next to each other. All these controls could replace all the other controls.

Location Controls

- The last kind of geometric control we look at is the next most complicated: Location.
- Location control is just like it sounds, they control the acceptable variance in the location of a feature.
  - These controls are so important that entire chapter of the standard deals with these controls and entire seminars are based on understanding true position theory and practice.
- There are three kinds of location control - I have a hard time remembering the last two of these and don't have a good way of helping you so it's memorization time.

Location
Position

- The obvious location control for features is position.
- The icon to represent where a feature should be, we use a target symbol 
- *This reminds me of one of my favorite military sayings:*
  - ***Mechanical engineers build weapons.***
  - ***Civil engineers build targets.***

Location
Symmetry

- The first non-obvious location control is called symmetry.
- The icon to represent when two things should be symmetric is ⌯
- Note the icon shows a mirror reference line (the datum) and two little symmetric lines.

I have never placed a symmetric tolerance on a drawing or checked a drawing that had it (correct).

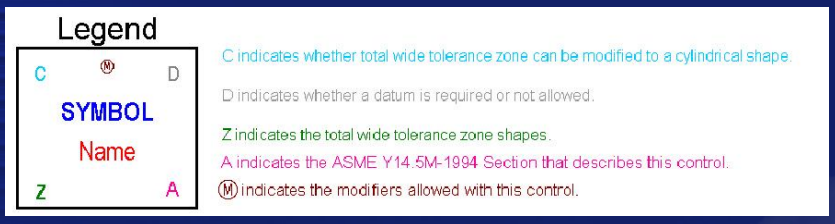
Location
Concentricity

- The second non-obvious location control is called concentricity.
- The icon to represent when two things should have the same center axis is ◎
- Most GD&T newbies love hearing about this symbol and are anxious to use it although they probably should be using runout - this is hard to inspect.

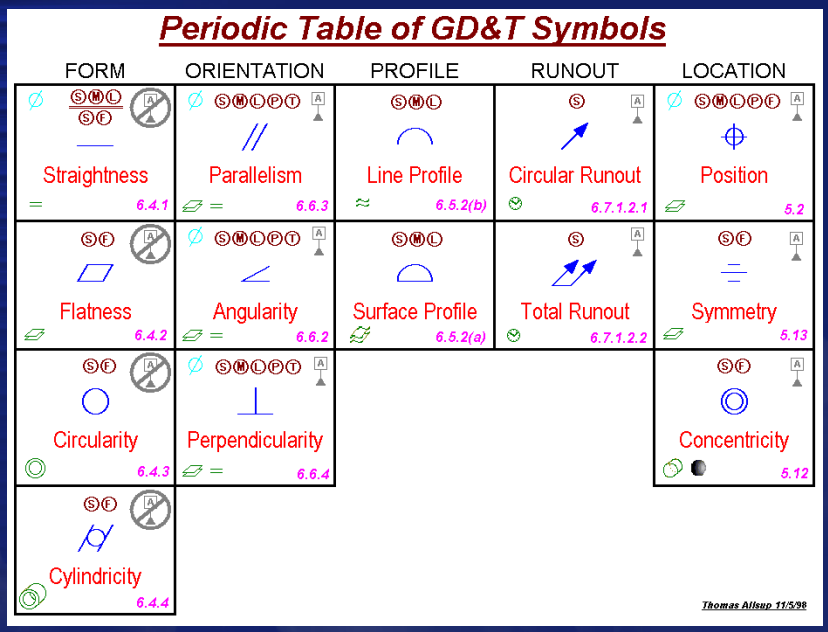
FOPRL
Complete

FORM	ORIENTATION	PROFILE	RUNOUT	LOCATION
STRAIGHTNESS	PARALLELISM	LINE PROFILE	CIRCULAR RUNOUT	POSITION
FLATNESS	ANGULARITY	SURFACE PROFILE	TOTAL RUNOUT	SYMMETRY
CIRCULARITY	PERPENDICULARITY			CONCENTRICITY
CYLINDRICITY				

FOPRL Legend



In each control, we place information so someone "skilled in the arts" will have all the information they need to work with the control.



Features of Size

- We will now go deeper into the concepts of GD&T by starting to look at features of size.
- This conversation will led us to discussing modifiers.
  - Modifiers are the little letters in circles.
- Along the way, we'll learn Rule #2 of GD&T.
- We'll finish by starting to talk about tolerance zones & shapes of all the geometric controls.

What's a Feature?

- ASME Y14.5M-1994 Section 1.3.12 Defines a Feature as the general term applied to a physical portion of a part, such as a surface, pin, tab, hole, or slot.
- In other words, any distinctive portion of a part that might be dimensioned is a "feature".

What is Size?

- 1.3.24 Actual Size : The general term for the size of a produced feature.
  - This is what you measure on a part.
- 1.3.27 Limits Of Size : The specified maximum and minimum sizes.
  - This is the numbers found on the drawing.
- 1.3.28 Nominal Size : The designation used for purposes of general identification.
  - 28 Gauge wire, 1" Schedule 40 pipe, 2x4

Size Isn't Important

Physical features are grouped into two distinct regimes:

- Features that do not depend on size
  - Single surfaces, lines, arcs
  - Sometimes called "Not Related Features"
- "Features of size"
  - Plates, holes, slots, balls
  - Sometimes called "Related Features"

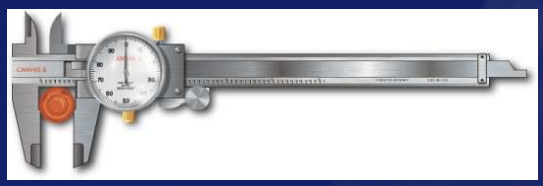
Feature of Size Examples

- One cylindrical surface
- One spherical surface
- Set of two opposed elements
- Set of opposed parallel surfaces

The "Caliper" Check

Things that you are measure with a pair of calipers are features of size:

- Inside Jaws
- Outside Jaws
- Depth Gauge



Why Are Features of Size Important?

- Geometric tolerances for features of size can be ***modified*** according to the "size of the feature".
- Everyone knows that engineers love to modify things.

What Do the Circled Letters Stand For?

- The geometric tolerance for features of size can be modified in several methods but the two most important are:
  - Regardless of Feature Size (RFS)
  - Maximum Material Condition (MMC)
- There is also LMC and Free State
  - For a good time, ask an ISO1101 person for an explanation of the Envelope modifier.

Regardless of Feature Size

- This is the default if no modifier is given.
- The tolerance zone is not affected by the actual size of the feature.
- You don't see the symbol Ⓢ anymore except in GD&T training sessions.
- Just because you don't see the symbol doesn't mean the concept isn't used all the time.

Second Rule of GD&T

- Remember the first rule of GD&T states the limits of size are the first magnitude of control.
- The second rule of GD&T states that if the geometric tolerance is applied to a feature of size then it is assumed to be regardless of feature size.

Maximum Material Condition

- The stated tolerance applies when the most material is there.
  - The tolerance zone increases when there is less material – you get a "***bonus tolerance***" if a hole is large.
- Examples:
  - Thickest plate
  - Smallest hole

"Worst Case Scenario"

MMC is normally valid only when all of these conditions exist:

- Two or more features are interrelated with position or orientation.
- At least one of the features is a feature of size.
- The feature with which MMC is to be applied must be a feature of size with a axis or center plane.

Note: We used to call MMC, the "worst case".

Least Material Condition

- The stated tolerance applies when the least material is there.
  - This is a rarely used modifier.
- Examples:
  - Thinnest plate
  - Largest hole

Why is LMC rare?

- Most tolerance analysis is checking whether part will go together.
- If you are checking if a male part will go into a hole, you need to know the largest male part and the smallest hole - both of which are MMC.
- LMC can be used to see what the maximum clearance is in a system but that analysis is pretty rare.

Free State

- Free State Variation : A term used to describe distortion of a part after removal of forces applied during manufacture.
- You see this on lots of flexible parts, like rubber gaskets, wire forms, and some thin walled plastic components.

Two More Modifiers

- Technically, the following two modifiers do not affect features of size.
- However, there is no good place to put them in this seminar.
- So, here's Tangent Plane and Projected Tolerance Zone.

Tangent Plane

- This modifier tells the inspector to place a tangent plane on a surface and measure the gauge plate, not the part.
- This modifier is commonly used by orientation controls.
  - It will become clear a little later when we look at tolerance shapes.

Ⓣ

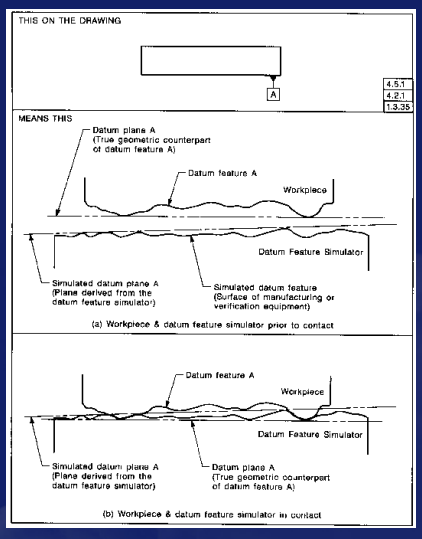
Projected Tolerance Zone

- Used only with position & orientation tolerances.
- Mainly position and perpendicularity.
- Circled P appears after any modifiers and is itself followed by the projected height.
- The words are “with a projected tolerance zone of …”
- For clarification, a chained line can be drawn and dimensioned with a minimum height dimension (not a basic dimension).

Ⓟ

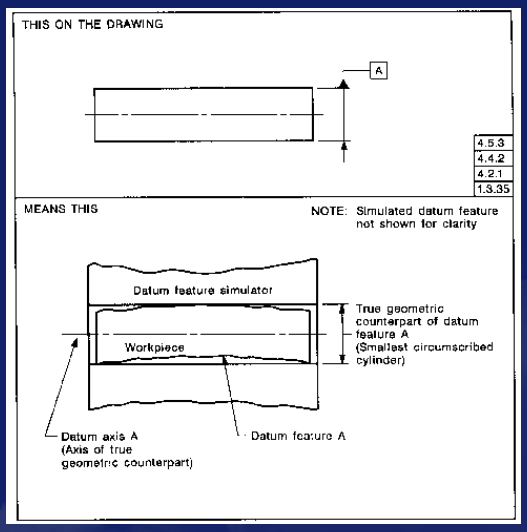
What Does it Mean #1?

- This datum is the bottom surface.
- This datum is not a feature of size.
- The datum simulator can be a gauge plate.
  - Gauge plate needs to be 10X flatter than what you want to check.



What Does It Mean #2?

- This datum is the center axis.
- This datum is a feature of size.
- The datum simulator is a two piece “clamp”.
  - RFS - clamp squeezes
  - MMC - clamp is fixed size



Basic Dimensions

- If you think of datums as a method of immobilizing a part then basic dimensions are just offsets from that reference frame.
- Basic dimensions are boxed dimensions.

.500

- Basic dimensions don’t have tolerances, they are used by other geometric dimensions.

Wrap Up

- 14 GD&T Symbols
  - FOPRL Chart
- Features of Size
- Datums
- Modifiers

After the break, let's look at the new ASME Y14-2009 standard.

Geometric Dimensioning and Tolerancing for Burn-In and Test Professionals

Thomas Allsup

anida technologies

66

Hour 2

- What's New in the new ASME Y14.5-2009 standard.

What a Long Strange Trip It's Been

- The ASME took over the publication of the standard from ANSI in 1989.
  - I still cringe when I hear people say they know ANSI GD&T, it is kind of like saying you know Latin as you try to speak Spanish
- 1994: slightly updated with the biggest change being the addition of metric dimensions hence the "M" in the title.
- 1999: reaffirmed without changes.
  - This is the GD&T standard that an entire generation has used for creating and interpreting drawings.

Remember the Good Times

- In 2009, the standard was changed significantly for the first time since 1994.
- Introducing the new ASME Y14.5-2009!

Ch-ch-ch-changes

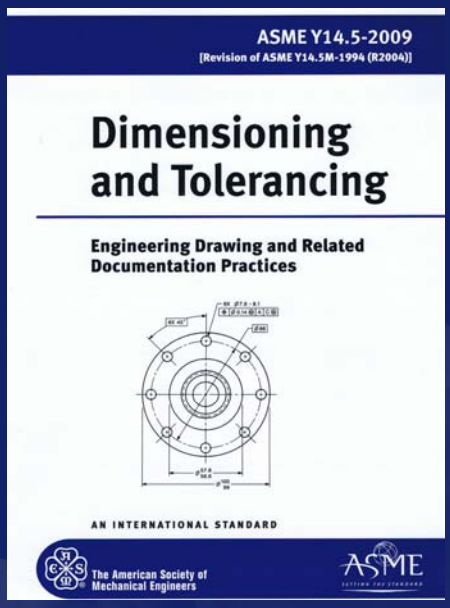
- New standard has new symbols and refines some existing terms but the most obvious change is the order & segregation of the 5 types of controls.
  - FOPRL will be coming up soon…
- Maybe someone on the Y14 committee does listen to us users after all
  - Actually there are quarterly meetings & lots of opportunities to comment on all the drawing standards.

Old Flame

- Before we discuss the changes, an obvious question is do we have to learn the new standard?
  - If you create all your own drawings and never get drawings from customers then you can keep using the old standard.
  - If you are like me and have to interpret whatever is thrown at me then you need to buy and start studying the new standard.
- We'll mention this later but don't throw your old standards away.

Cover Me

- Let's start with the cover.
- Previously the front picture was datums on the bluish green color background
- Now the cover is almost completely white with a blue strip at the top and bottom and a simple relatively small figure of a drilled flange with a single position GD&T tolerance.



Who Needs Pictures?

- When you start using the new standard, your drawing formats should be revised to have words like:
  - "Interpret this drawing using ASME Y14.5-2009"
- Don't throw your old standard away:
  - You might need it to interpret the drawings you created or receive from others that were created from 1994 to 2009.
  - Don't use the drawing date to determine what standard to use, look for the note on the drawing.

I Walk The Line

- In this session, we do not have time to go line by line with the changes but I will try and point out the biggies like the new symbols and the redefined terms.
- ASME offers a full 8 hour seminar discussing every minute change.
  - Hey, who added that comma?
- Appendix A of ASME Y14.5-2009 has a list of every change.

This Song Has No Title

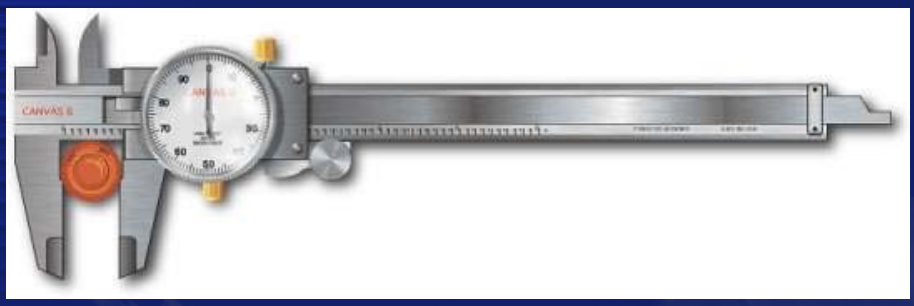
- In Section 1.2.1 Cited Standards now has ASME Y14.41-2003 (reaffirmed in 2008) Digital Product Definition Data Practices.
  - Provides guidance to 3D model with embedded dimensions and tolerances.
- Numerous new citations back to this standard.
  - I wanted to mention that the embedded 3D data has been allowed since 1984 but now it is really well documented.

Material Girl

- Section 1.3.3,1.3.4, and 1.3.49 introduce new datums terms for
  - Least Material Boundary
  - Maximum Material Boundary
  - Regardless of Boundary Size
- The symbols are the same for features.
- Features will continue to use the terms LMC, MMC, and RFS.

Big 'Ol Truck

- Section 1.3.32.2 introduces the new term "Irregular Feature of Size"
- We've always had features of size
  - Remember the "caliper test"?
  - Cylindrical surface
  - Spherical surface
  - Two opposed parallel elements or surfaces
- These are now called "regular" features of size
- Now we get to introduce "Irregular" Features of Size

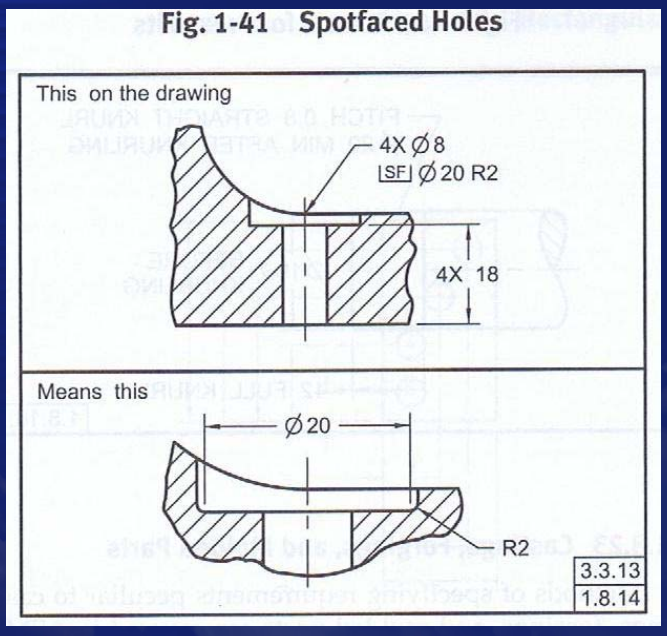


Shapes of Things

- Now an arbitrary profile can be identified as a datum.
- If that profile follows the “caliper test” then material modifiers can be applied.
- Imagine extruded shape profiles, key holes, splines, or other unusual shapes now being able to be considered a datum.

Love is a Bore

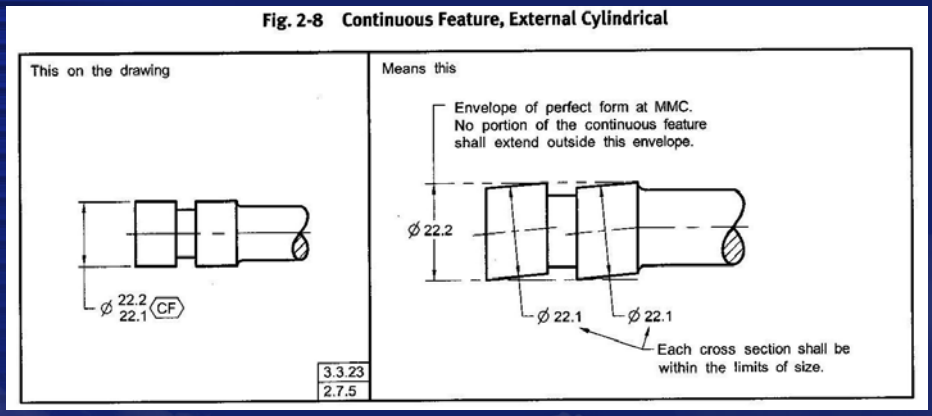
- Section 1.8.14 Spotfaces now have a new symbol that is a counterbore symbol with "SF" inside the symbol.
  - Previously it was the same as the counterbore with no depth specified.
- Spotfaces used to use the same symbol as a counterbore with only the depth missing.
- Now you can also add a radius to the edge of the counterbore as well as the main diameter.



The Continuous Life

- Section 2.7.5 adds a new phrase "CONTINUOUS FEATURE"
- There is a new symbol for this as well, the letters CF in an irregular hexagon.
- A Continuous Feature is two of more features of size that are not contiguous (touching) but wish to be treated as a single surface.
  - Example: A shaft with grooves cut into it. The main shaft could be called a single continuous feature.

Fig. 2-8 Continuous Feature, External Cylindrical

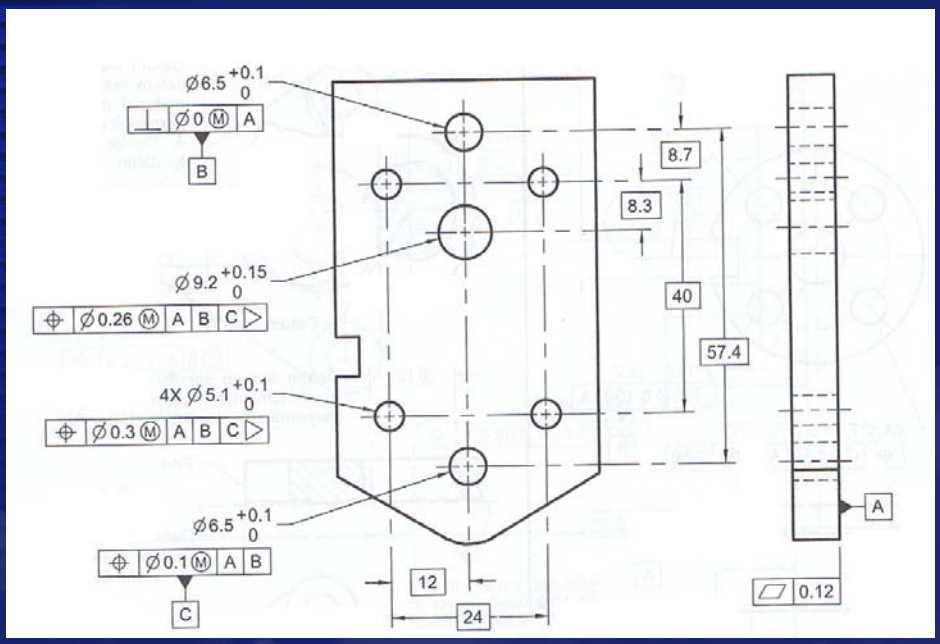


Symbol of Life

- Figure 3-11 Adds the following new symbols:
  - Datum Translation
  - Unequally Disposed Profile
  - Independency

Movin' Out

- New Datum Translation Symbol is a triangle on its side like a pointer.
- This overrides the basic dimension for locating a position of a tolerance zone.
- This only makes sense if you have a couple of geometric tolerances on a single feature and you want one of the datum callouts to move with the limits if the tolerance and one of the datum callouts need to be absolute in space.

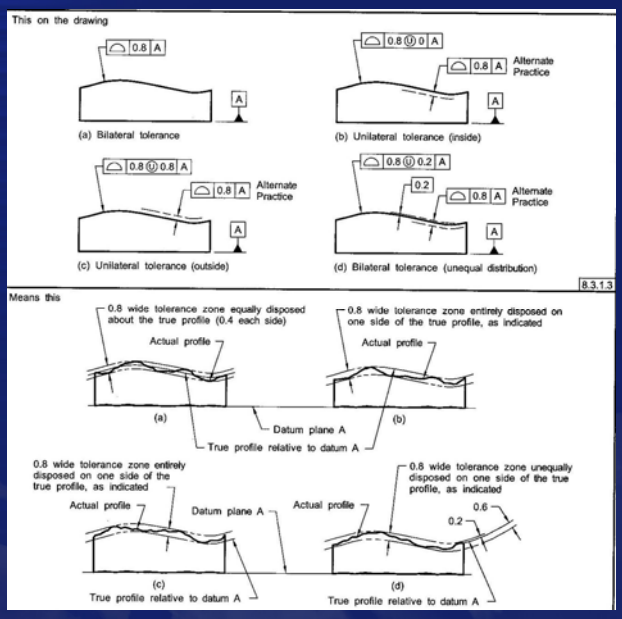


No Equal

- New Unequally Disposed Profile Symbol is a “U” in a circle.
- This concept has always been in the standard but required you use chain lines and basic dimensions to determine the distribution of a profile tolerance zone other than 50%-50% (practice still allowed).
- In the feature control frame you add the symbol and the value of how much material you want to add.
  - 0.5 Ⓤ 0.5 means it is all added
  - 0.5 Ⓤ 0 means it can only remove material
  - 0.5 Ⓤ 0.1 means it can be 0.1 added material and no more than 0.4 removed.

How much is...

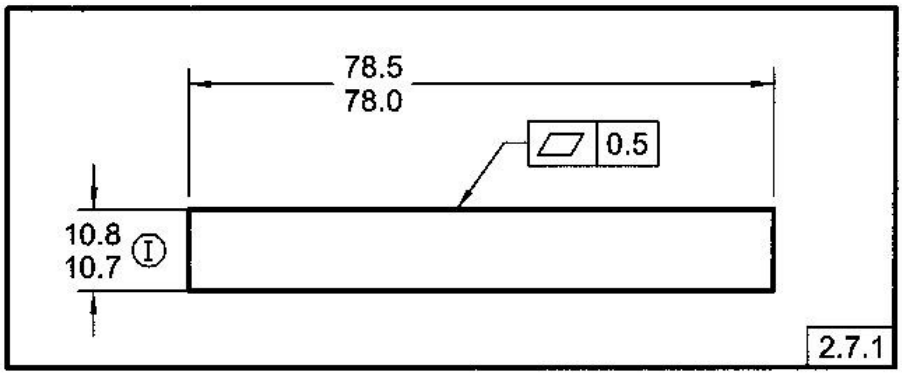
OUT?



Independent

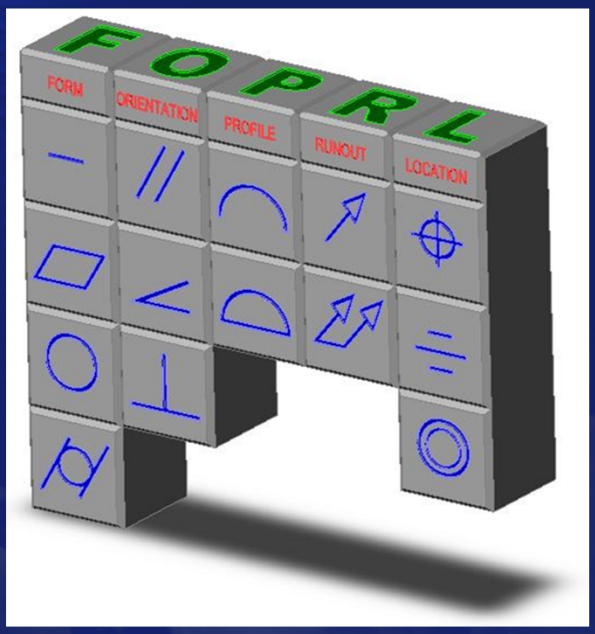
- New Independency Symbol is an ""I" in a circle.
- Previous standard required you write out "Perfect Form at MMC (or LMC) is not required."
- Example: If you say a shaft is toleranced at MMC then it must be straight but size may be all that is important to you so you can
- This choice of symbol and wording baffles me – If would have gone Old School Ghostbusters and made a circular no symbol with a slash through it and "PF" inside.

Fig. 2-7 Independency and Flatness Application



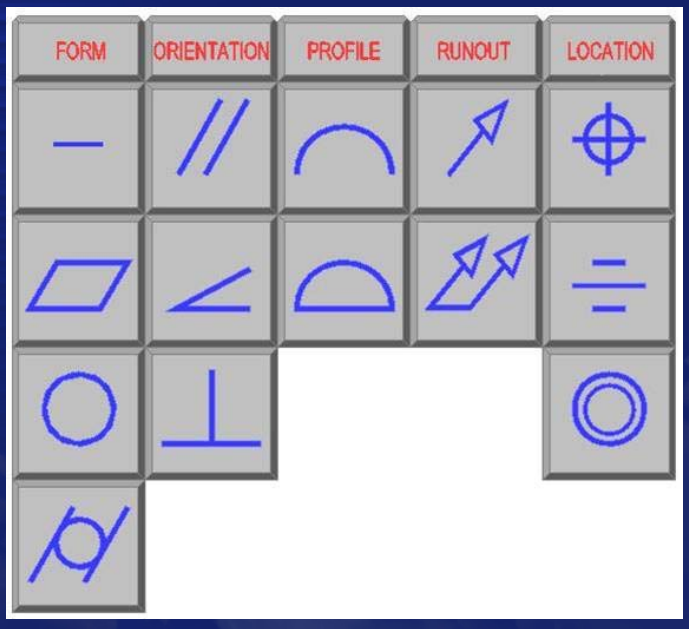
Back To Basics

- Basic dimensions have always had two methods of identification
  - Put a rectangular box around them
  - Put a note that says untoleranced dimensions are basic
- Basic dimensions can now be identified in “digital data file” (explicitly stated for the first time)



Beauty Through Order

- New Order:
  - Form, Orientation, Location, Profile, Runout
- FOLPR is getting closer to FOPRL....
  - It still nice to see the “easier” concepts of form and orientation starting off but with location being such a big oddity I would still make it last but the standard isn’t supposed to be a learning tool.
  - Plus it keeps people like me giving seminars.



Everything In Its Right Place

- What's New with Location
- The chapter is much, much, much longer with lots more examples.
- Whole new section about coaxial features and lists differentiators for coaxial, runout, and concentricity - one of the most confusing tolerances “around”.
  - Sorry for the bad pun.

The Right Profile

- What's New with Profile
- We have always been able to place a circle around the jog of a callout to change it to the “All Around” requirement without the note.
  - This means it only applies to the surfaces in the view called out.
- We can now place a double circle around the jog of a callout to change it to “All Over” requirement.
  - This means it apples to all the surfaces of the part.
  - Can not be placed on an isometric projection – not sure why.
- Profiles can now be datum features and can be modified with material modifiers.

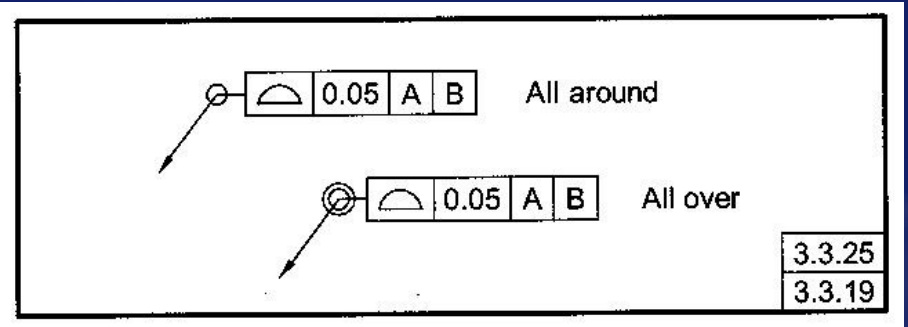
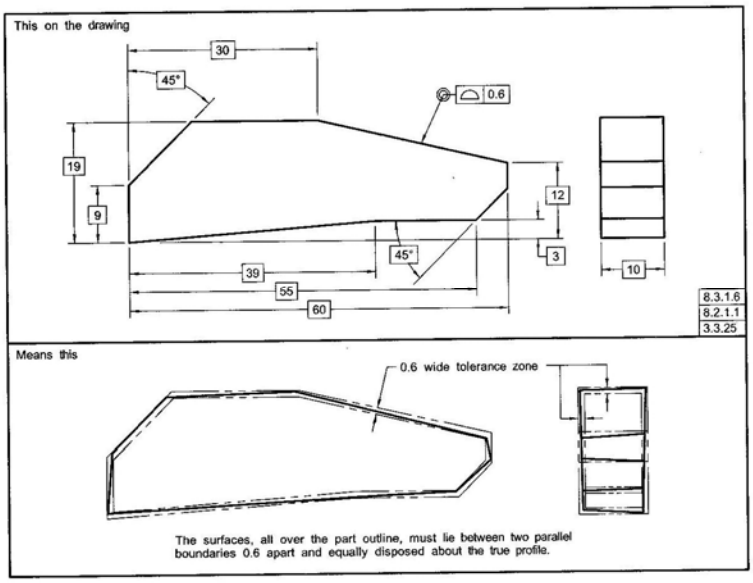
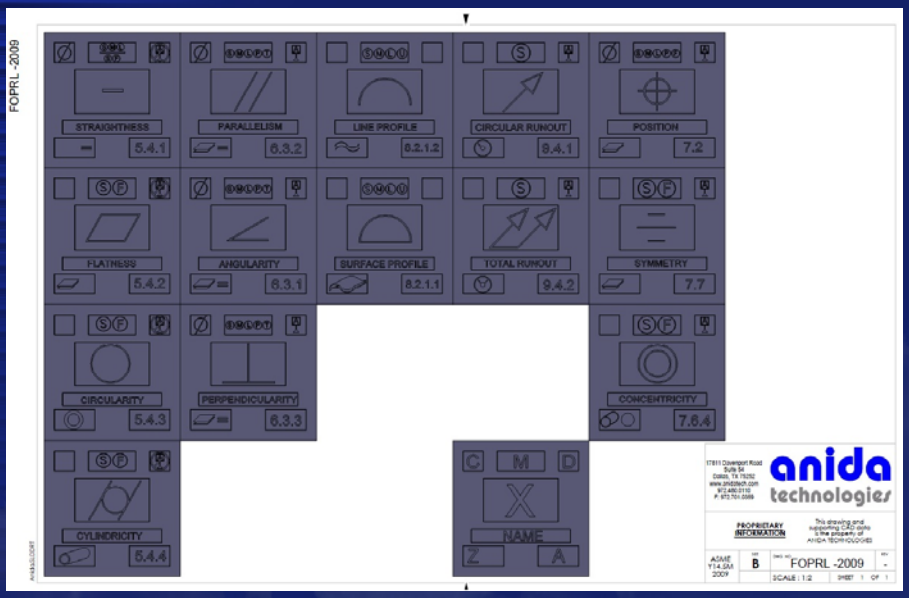


Fig. 8-8 Specifying Profile of a Surface All Over





Back To Basics - Christina Aguilera

Beauty Through Order – Slayer

Big 'Ol Truck - Toby Keith

Changes – David Bowie

Continuous Life - 311

Cover Me – Bruce Springsteen

Everything In Its Right Place – Radiohead

I Walk The Line – Johnny Cash

Independent – Webbie

Love is a Bore- Barbara Streisand

Material Girl – Madonna

Movin' Out – Billy Joel

No Equal – Beatnuts

Old Flame – Alabama

Remember The Good Times – Willie Nelson

Shapes of Things – Yardbirds

Symbol of Life – Paradise Lost

The Right Profile – The Clash

This Song Has No TItle - Elton John

What a Long Strange Trip It's Been – Grateful Dead

Wheel in The Sky – Journey

Who Needs Pictures? – Brad Paisley

Reprise

Geometric Dimensioning and Tolerancing for Burn-In and Test Professionals

Thomas Allsup

anida technologies

103

Hour 3

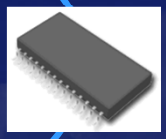
- Examples of Semiconductor Package Drawings
- These drawings come from multiple manufacturers.
  - There are many more good drawings than bad so it was a little difficult to find these bad examples.

Package Outlines

Leaded TO Package

Dual Inline Package

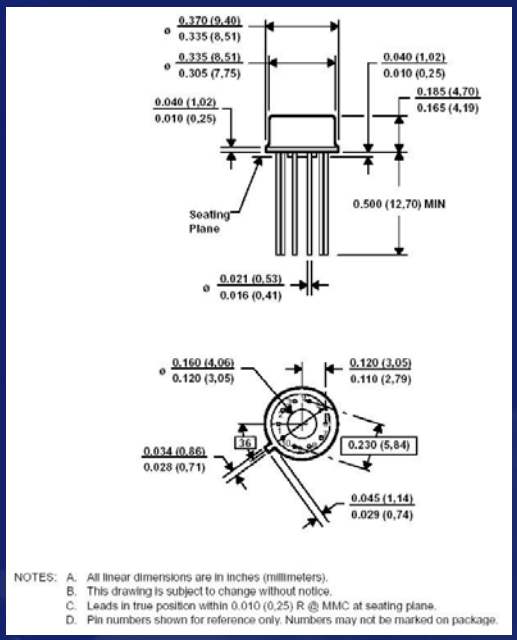
Small Outline Integrated Circuit



Ball Grid Array

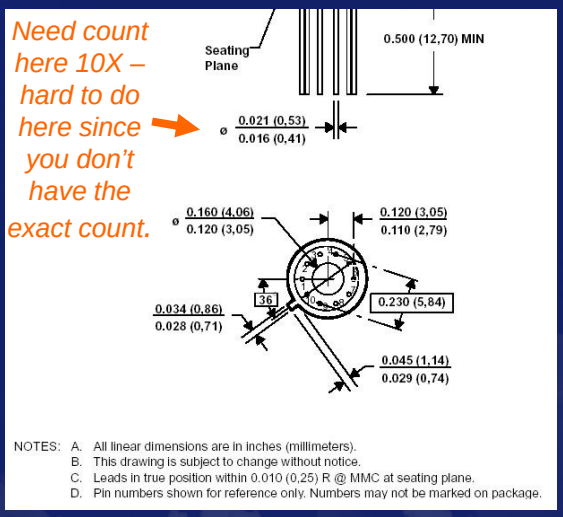
Quad Flat No-Lead

Leaded TO Package



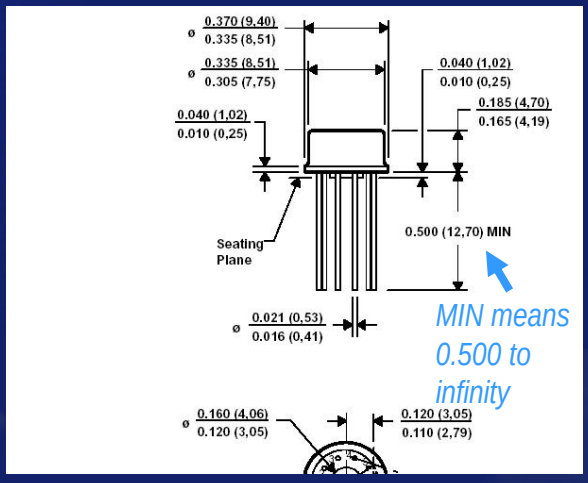
Leaded TO Package

Need count here 10X – hard to do here since you don't have the exact count.



NOTES:
A. All linear dimensions are in inches (millimeters).
B. This drawing is subject to change without notice.
C. Leads in true position within 0.010 (0,25) R @ MMC at seating plane.
D. Pin numbers shown for reference only. Numbers may not be marked on package.

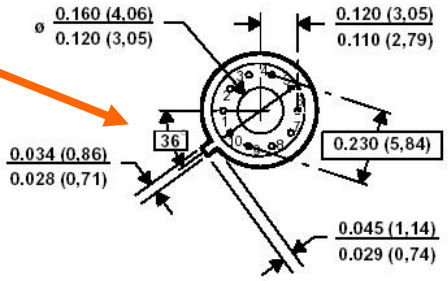
Leaded TO Package



MIN means 0.500 to infinity

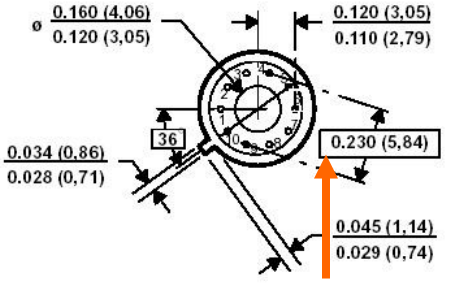
Leaded TO Package

Degree symbol is not legible.



NOTES:
A. All linear dimensions are in inches (millimeters).
B. This drawing is subject to change without notice.
C. Leads in true position within 0.010 (0,25) R @ MMC at seating plane.
D. Pin numbers shown for reference only. Numbers may not be marked on package.

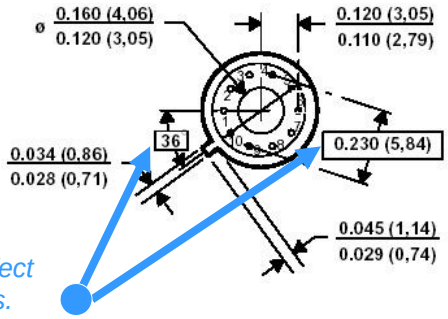
Leaded TO Package



Missing diameter symbol ∅

NOTES:
A. All linear dimensions are in inches (millimeters).
B. This drawing is subject to change without notice.
C. Leads in true position within 0.010 (0,25) R @ MMC at seating plane.
D. Pin numbers shown for reference only. Numbers may not be marked on package.

Leaded TO Package

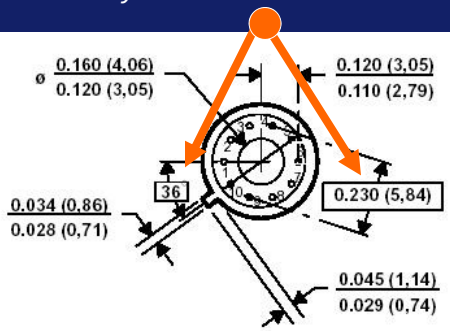


Theoretical perfect position of leads.

NOTES:
A. All linear dimensions are in inches (millimeters).
B. This drawing is subject to change without notice.
C. Leads in true position within 0.010 (0,25) R @ MMC at seating plane.
D. Pin numbers shown for reference only. Numbers may not be marked on package.

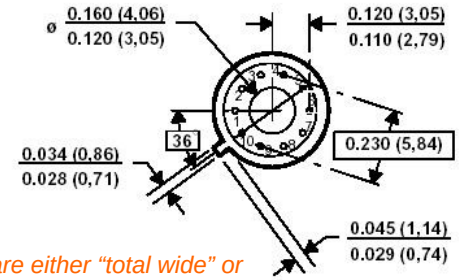
Leaded TO Package

Need count here 9X – hard to do here since you don't have the exact count.



NOTES:
A. All linear dimensions are in inches (millimeters).
B. This drawing is subject to change without notice.
C. Leads in true position within 0.010 (0,25) R @ MMC at seating plane.
D. Pin numbers shown for reference only. Numbers may not be marked on package.

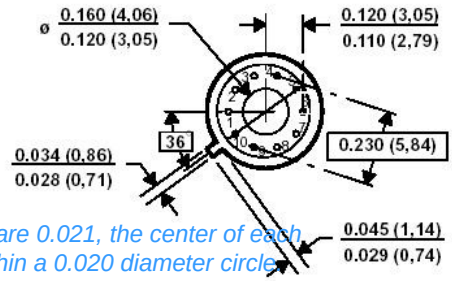
Leaded TO Package



Tolerance zones are either "total wide" or "cylindrical" denoted by a diameter symbol.

NOTES:
A. All linear dimensions are in inches (millimeters).
B. This drawing is subject to change without notice.
C. Leads in true position within 0.010 (0,25) R @ MMC at seating plane.
D. Pin numbers shown for reference only. Numbers may not be marked on package.

Leaded TO Package



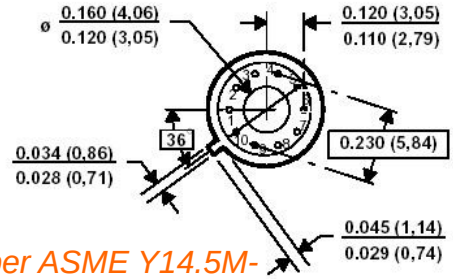
When the leads are 0.021, the center of each lead must be within a 0.020 diameter circle. When the leads are 0.016, the center of each lead must be within a 0.025 diameter circle.

NOTES:
A. All linear dimensions are in inches (millimeters).
B. This drawing is subject to change without notice.
C. Leads in true position within 0.010 (0,25) R @ MMC at seating plane.
D. Pin numbers shown for reference only. Numbers may not be marked on package.

GD&T Aside

- MMC is Maximum Material Condition
- Ⓜ is the symbol for MMC
- MMC means that the feature creates a part that weighs the most.
  - Thickest plate
  - Smallest hole
- Used to be called “Worst Case Scenario”.

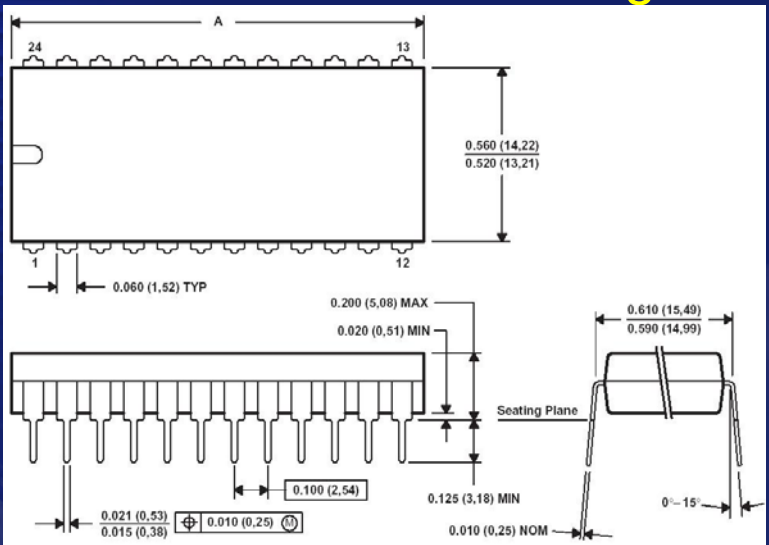
Leaded TO Package



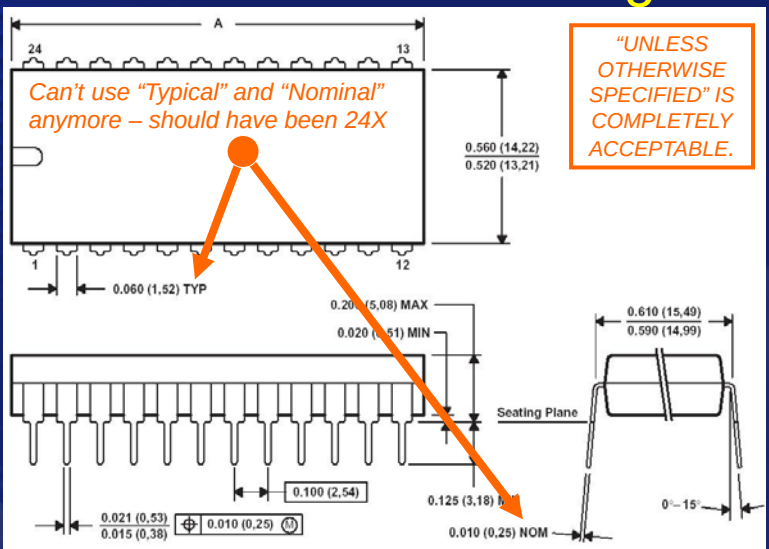
Interpret per ASME Y14.5M-1994 statement missing!

NOTES: A. All linear dimensions are in inches (millimeters).
B. This drawing is subject to change without notice.
C. Leads in true position within 0.010 (0,25) R @ MMC at seating plane.
D. Pin numbers shown for reference only. Numbers may not be marked on package.

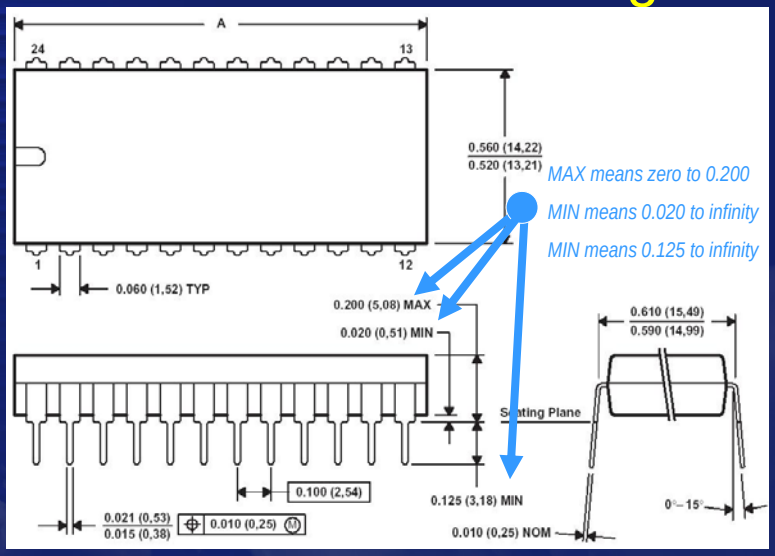
Dual Inline Package



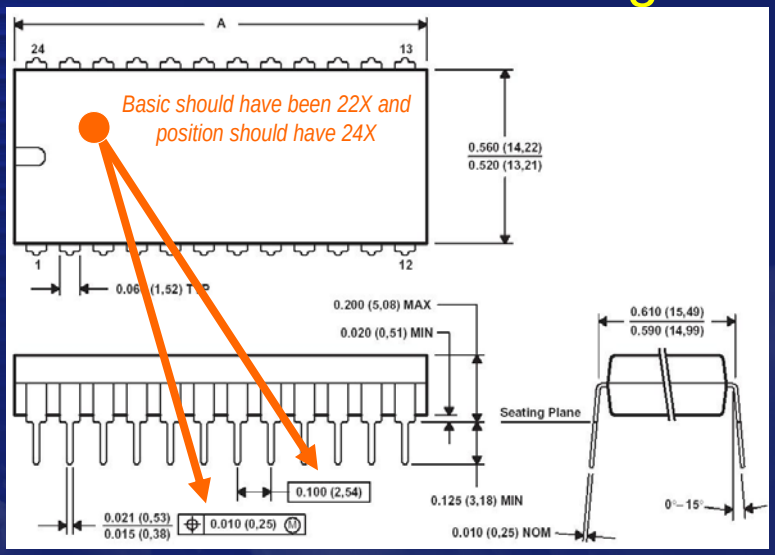
Dual Inline Package



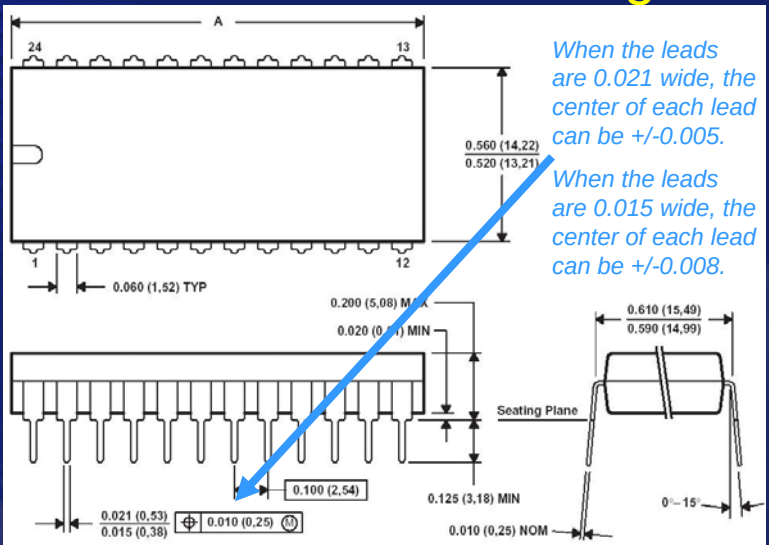
Dual Inline Package



Dual Inline Package



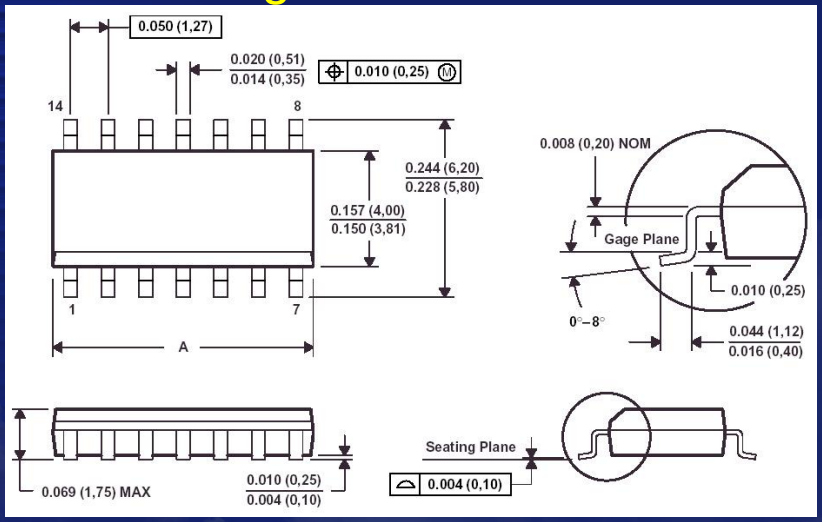
Dual Inline Package



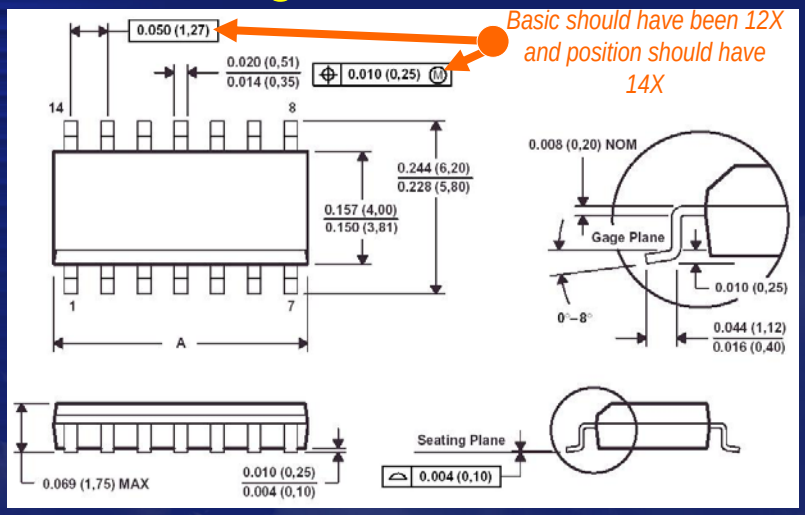
When the leads are 0.021 wide, the center of each lead can be +/-0.005.

When the leads are 0.015 wide, the center of each lead can be +/-0.008.

Small Outline Integrated Circuit

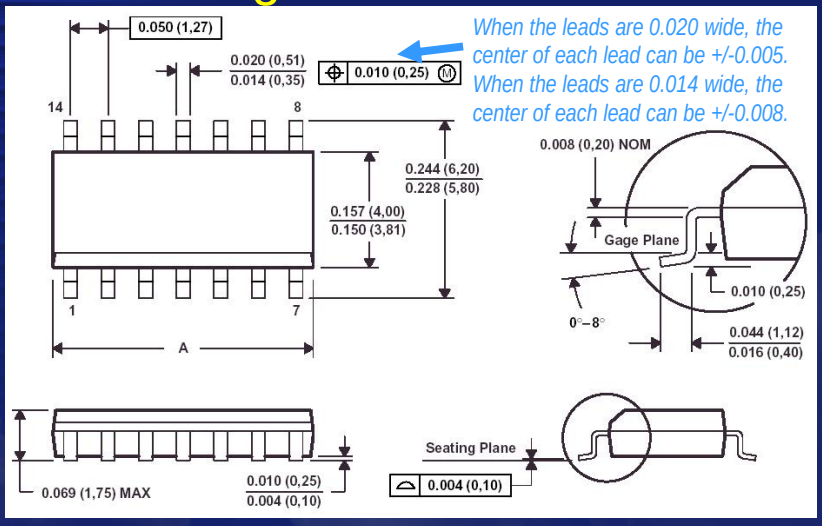


Small Outline Integrated Circuit



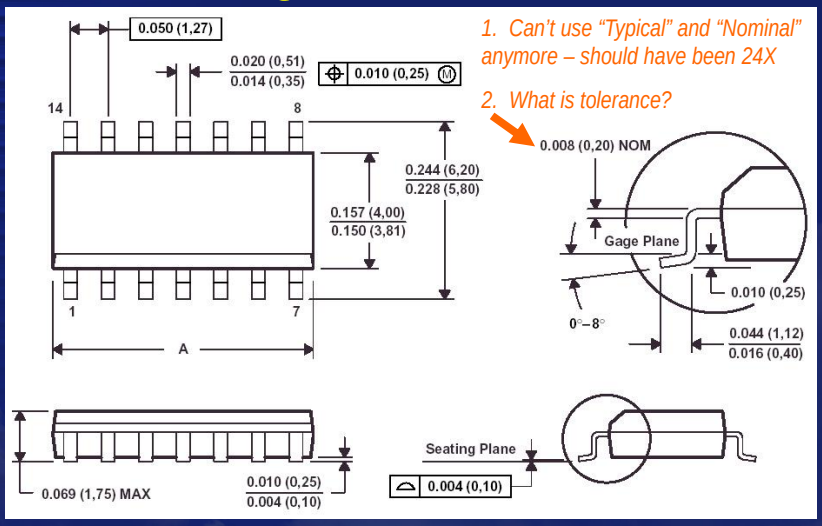
Basic should have been 12X and position should have 14X

Small Outline Integrated Circuit



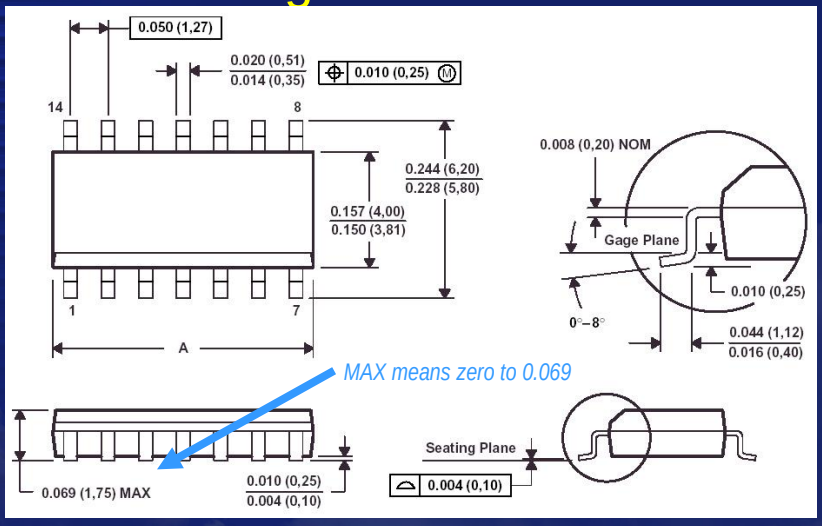
When the leads are 0.020 wide, the center of each lead can be +/-0.005. When the leads are 0.014 wide, the center of each lead can be +/-0.008.

Small Outline Integrated Circuit

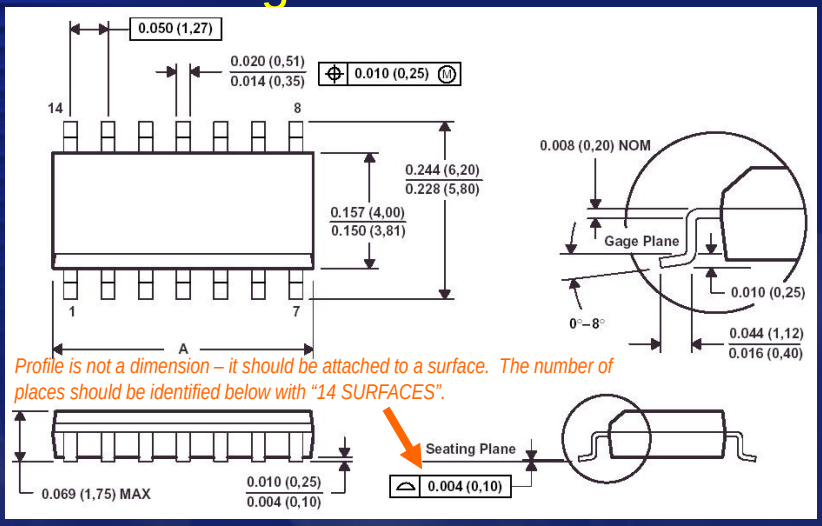


1. *Can't use "Typical" and "Nominal" anymore – should have been 24X*
2. *What is tolerance?*

Small Outline Integrated Circuit



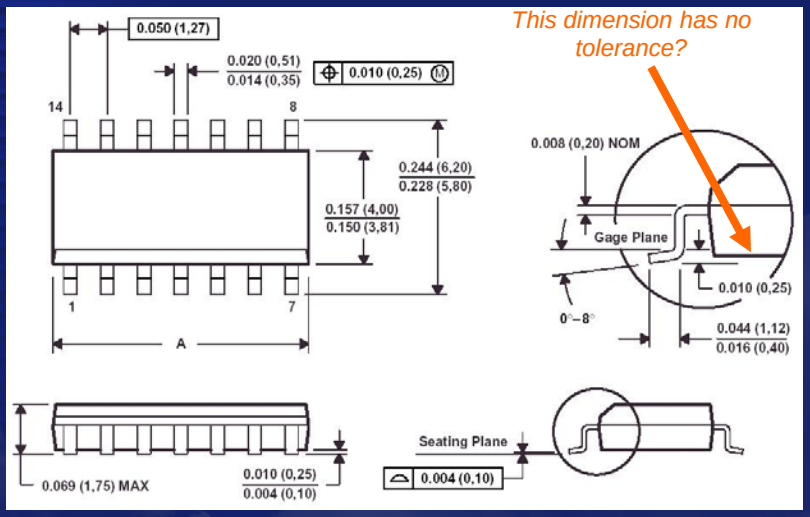
Small Outline Integrated Circuit



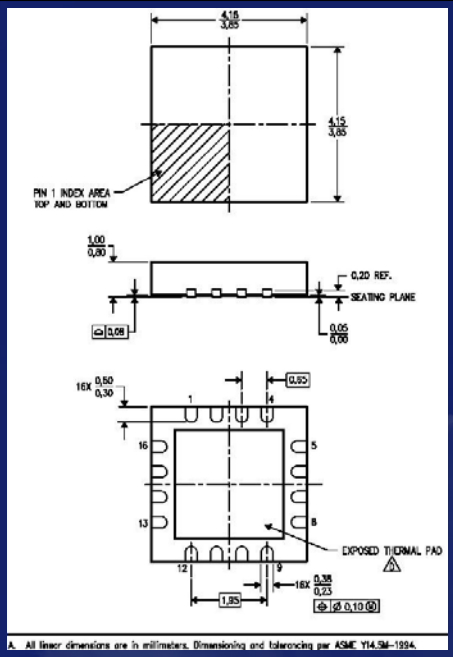
GD&T Aside

- The Profile Tolerance is an excellent tolerance for devices that will be inspected with optical comparators or vision systems.
- The surface must lie between two surfaces that are half the tolerance "above" and "below" perfect form.

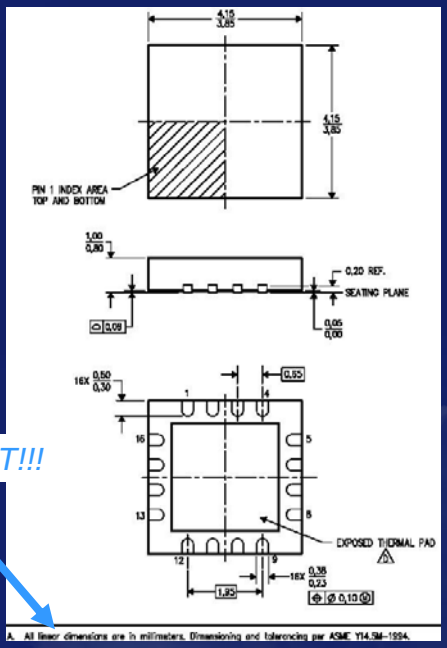
Small Outline Integrated Circuit



Quad Flat No-Lead

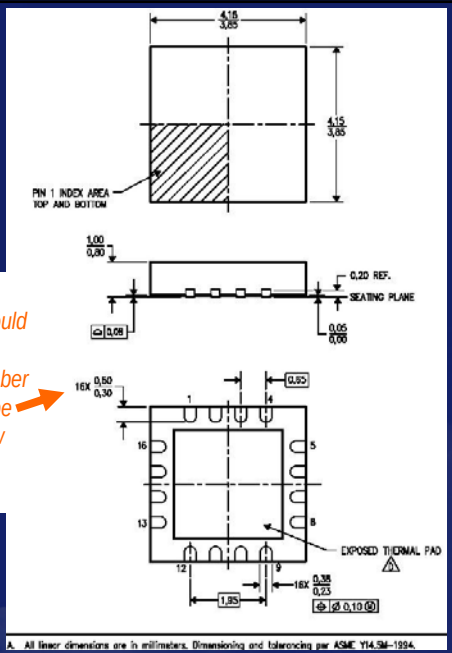


Quad Flat No-Lead



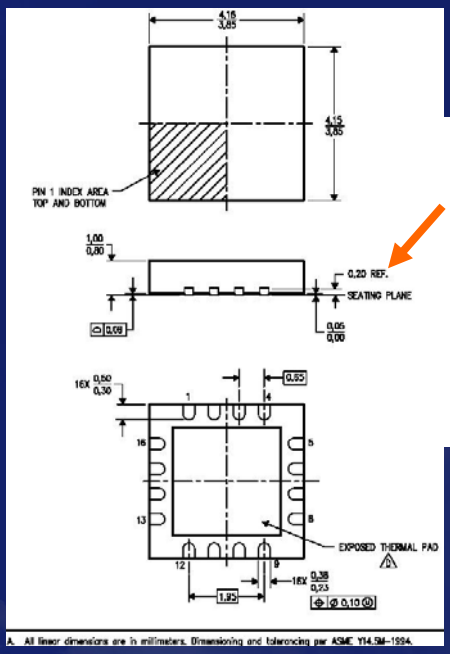
GREAT!!!

Quad Flat No-Lead



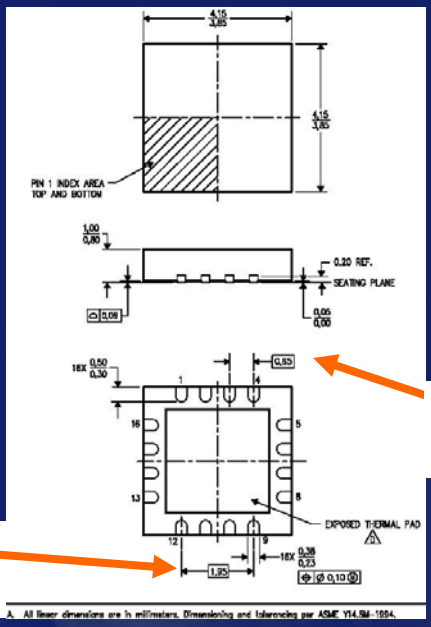
Profile is not a dimension – it should be attached to a surface. The number of places should be identified as below with "16 SURFACES".

Quad Flat No-Lead



1. *Don't use the word "REF" anymore – place dimension in parenthesis.*
2. *No tolerance on this dimension.*
3. *Abbreviations that do not form English words do not use a period.*

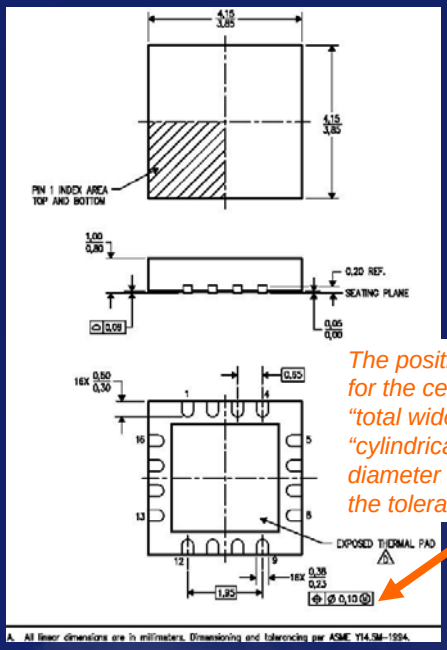
Quad Flat No-Lead



Need count 12X like pad width shown below.

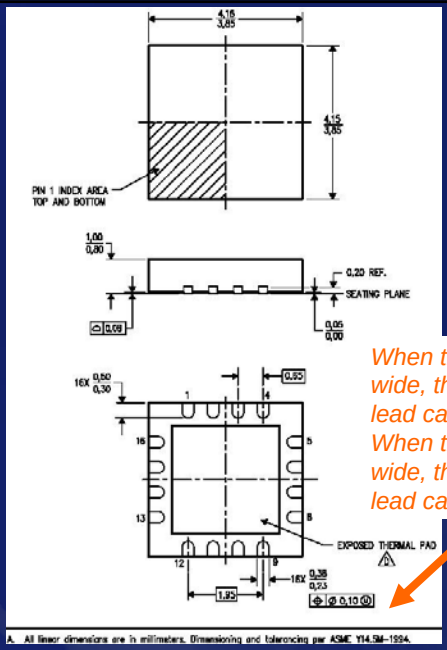
Need count 4X like pad width shown beside.

Quad Flat No-Lead



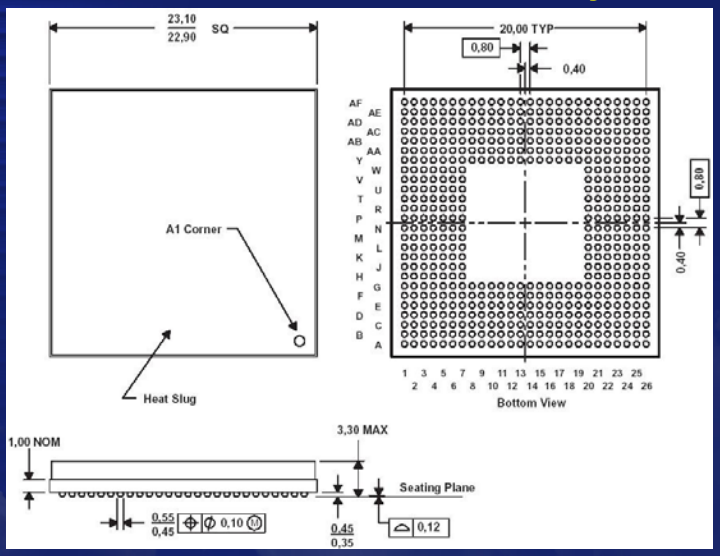
The positional tolerance for the center of the pad is "total wide" , not "cylindrical" so the diameter symbol before the tolerance is incorrect.

Quad Flat No-Lead

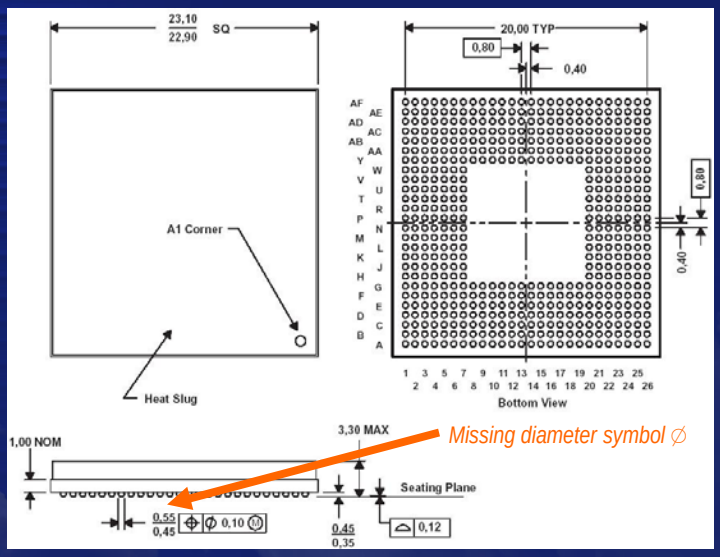


When the leads are 0.38 wide, the center of each lead can be +/-0.05. When the leads are 0.23 wide, the center of each lead can be +/-0.125.

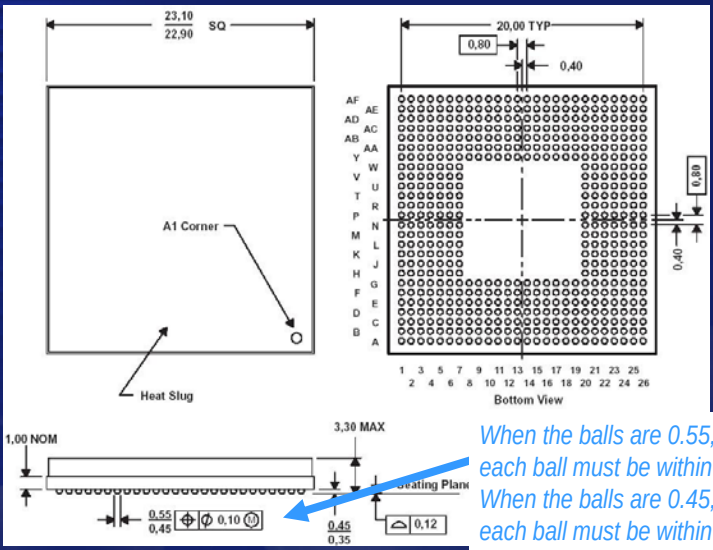
Ball Grid Array



Ball Grid Array



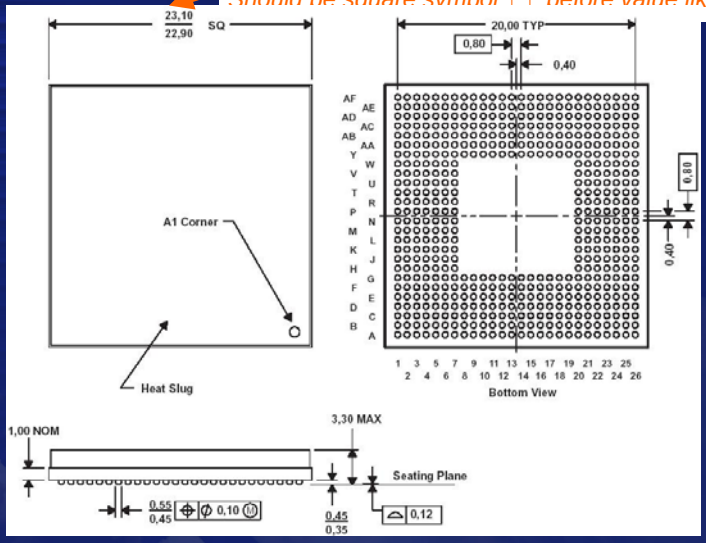
Ball Grid Array



When the balls are 0.55, the center of each ball must be within a 0.1 circle. When the balls are 0.45, the center of each ball must be within a 0.2 circle.

Ball Grid Array

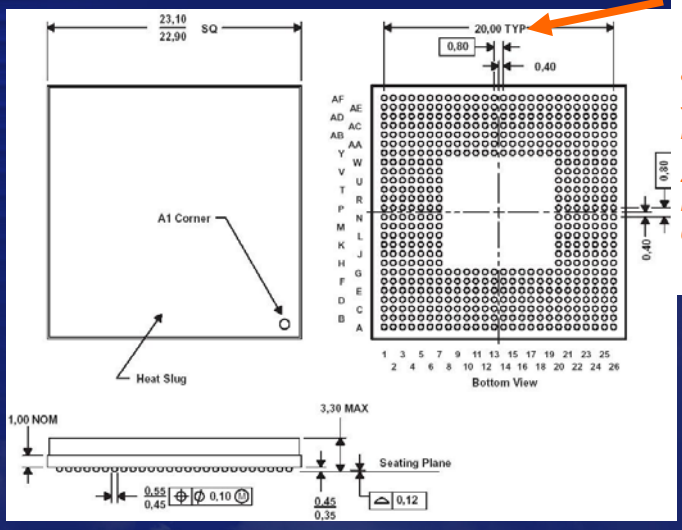
Should be square symbol □ before value like a



GD&T Aside

- Circles
  - Diameter ∅
  - Radius R
- Spheres
  - Diameter S∅
  - Radius SR
- Squares □
- Counterbore ⌴
- Depth ↧
- Spotface ⌴
- Countersink ⌵
- Slope
- Conical Taper

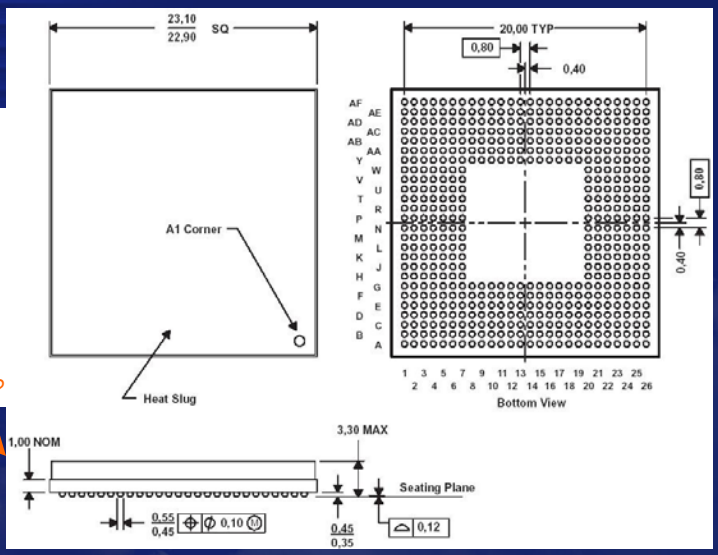
Ball Grid Array



1. *Can't use "Typical" and "Nominal" anymore – should have been 4X*
2. *Should be reference dimension.*

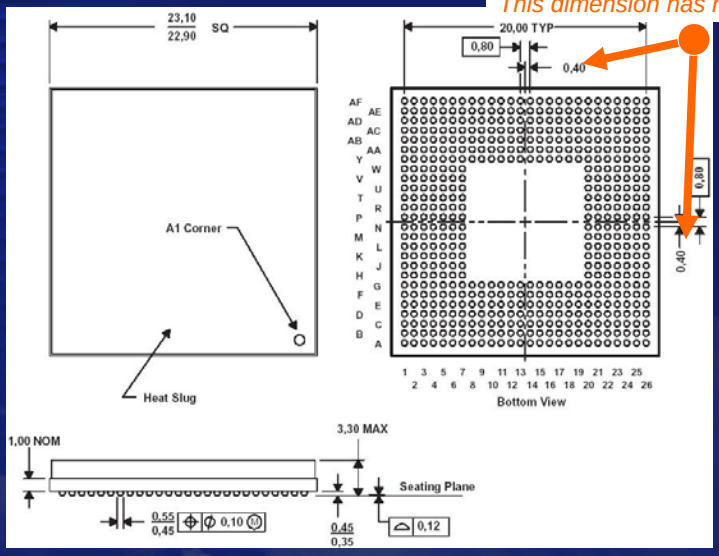
Ball Grid Array

Can't use "Typical" and "Nominal" anymore – Does this mean that this not a toleranced dimension?

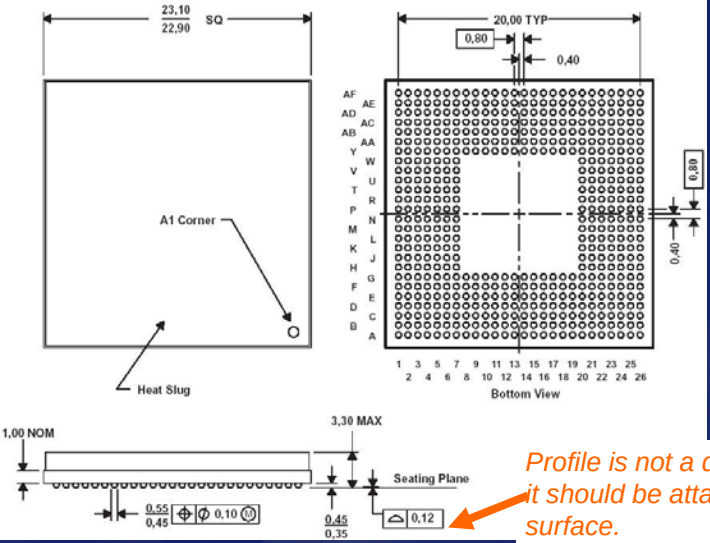


Ball Grid Array

This dimension has no tolerance?

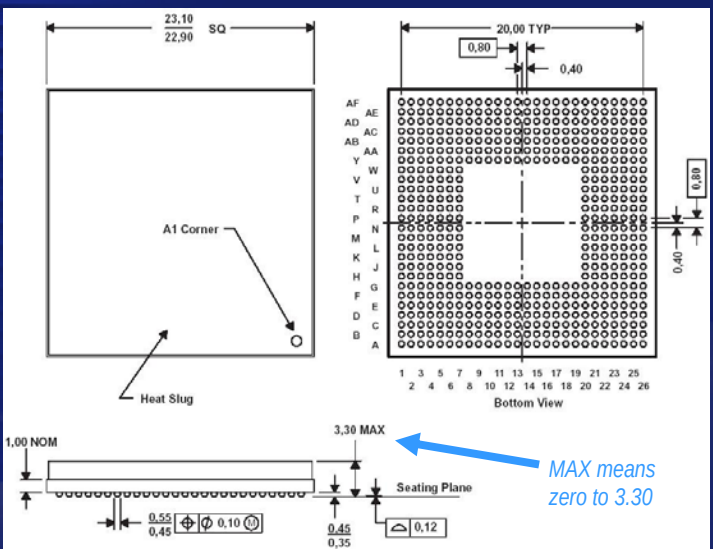


Ball Grid Array



Profile is not a dimension – it should be attached to a surface.

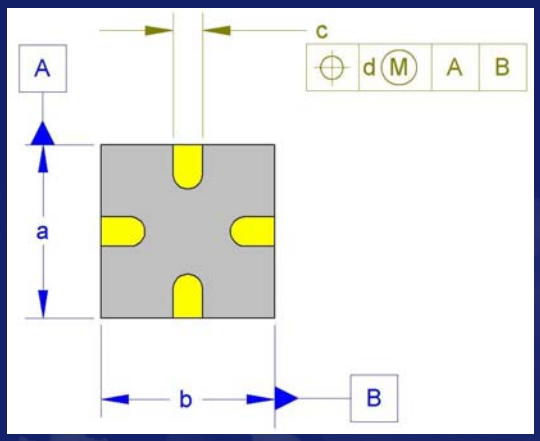
Ball Grid Array



MAX means zero to 3.30

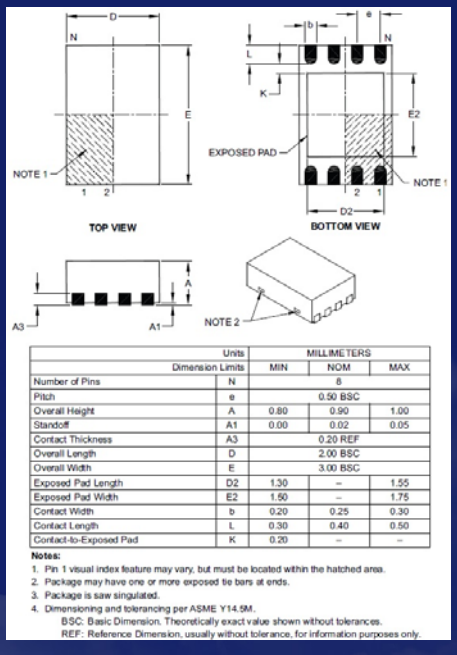
Practical Example

- The pads on this device are “c” wide.
- They are positioned within “d” when “c” is at MMC.



DFN Example

- ASME Y14.5M needs a year.
- BSC is fine but it needs a tolerance somewhere.



Units		MILLIMETERS		
Dimension Limits		MIN	NOM	MAX
Number of Pins	N		8	
Pitch	e		0.50 BSC	
Overall Height	A	0.80	0.90	1.00
Standoff	A1	0.00	0.02	0.05
Contact Thickness	A3		0.20 REF	
Overall Length	D		2.00 BSC	
Overall Width	E		3.00 BSC	
Exposed Pad Length	D2	1.30	–	1.55
Exposed Pad Width	E2	1.50	–	1.75
Contact Width	b	0.20	0.25	0.30
Contact Length	L	0.30	0.40	0.50
Contact-to-Exposed Pad	K	0.20	–	–

Notes:
1. Pin 1 visual index feature may vary, but must be located within the hatched area.
2. Package may have one or more exposed tie bars at ends.
3. Package is saw singulated.
4. Dimensioning and tolerancing per ASME Y14.5M.
   BSC: Basic Dimension. Theoretically exact value shown without tolerances.
   REF: Reference Dimension, usually without tolerance, for information purposes only.

Review

- Hour 1
  - Review GD&T Fundamentals
- Hour 2
  - What's New in the ASME Y14.5-2009 Standard?
- Hour 3
  - GD&T Semiconductor Drawing Examples

Questions?

- Thanks for your attention.
- If you have any questions, please hunt me down during the conference or drop me an email at tallsup@anidatech.com