

Module 1 Objectives



- Explain the elements and purpose of engineering drawings.
- Interpret both 1st Angle and 3rd Angle projections.
- Discuss the types of drawing errors and their costs.
- Define dimensions and tolerances.
- List types of dimensions and tolerances.
- Apply and interpret metric and inch dimensioning.
- Summarize the evolution of GD&T.
- Identify principle ASME and ISO dimensioning standards.
- List the fundamental dimensioning rules.
- Explain the ASME and ISO systems of limits and fits.
- List appropriate applications of coordinate and geometric tolerancing.
- Explain benefits and advantages of GD&T.

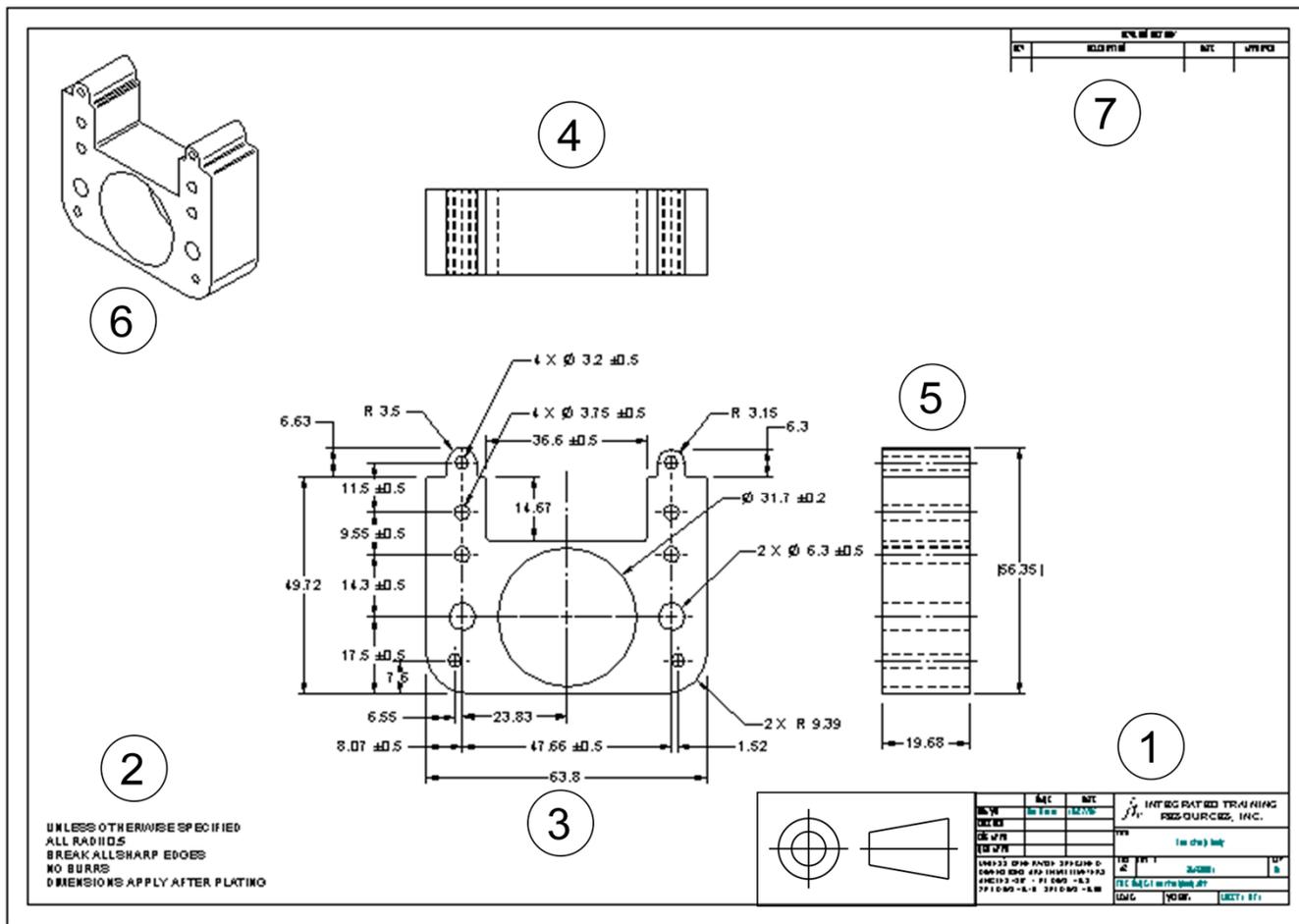
Copyright 1998-2015 ITR

GD&T 1-1

Integrated Training Resources, Inc.

In the space below, write any additional goals, objectives, or expectations you have for this module or course:

Handwritten text: *SAMPLE*



Copyright 1998-2015 ITR

GD&T 1-2

Integrated Training Resources, Inc.

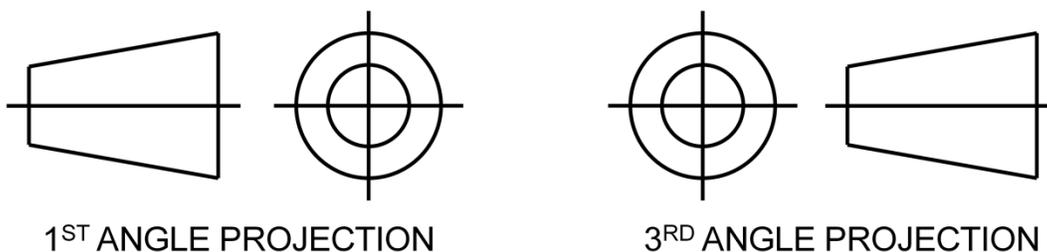
Engineering drawings contain several typical types of information, as illustrated in the drawing above. For the lamination stack body drawing above the circled numbers identify the following elements that are common to many mechanical engineering drawings or blueprints:

1. Title Block, including...
 - Company Name
 - Drawing Title and Date
 - Drawing Number
 - Drawing Revision
 - Drawing Size
 - Designer and Approval Names and Signatures
 - Materials
 - Default Tolerances, Notes, and Specifications
2. General Notes
3. Front View
4. Top View
5. Side View
6. Isometric View
7. Revision Block

Note:
 The principle views, including the front, top, bottom, and side views are typically arranged in grid on the field of the drawing using a technique known as *orthographic projection*.
 In the 3rd angle projection system, views are directly adjacent to the corresponding feature on the front view from which they were established (i.e. the *top* view is *above* the front view); this method is used in North America.
 In the 1st angle projection system, views are placed on the opposite side of the front view feature from which they were established (i.e. the *top* view is place *below* the front view); this method is used in Europe and Asia.

1st and 3rd Angle Projections $\int t_r$

- Multi-view drawings use _____ views based on either the 1st or 3rd angle projection.
- Simplified definition of the two system:
 - 1st Angle means the part is *outside* the bowl.
 - 3rd Angle means the part is *inside* the bowl.
- One of the two symbols below should be added to the title block to designate which system is used.



Copyright 1998-2015 ITR

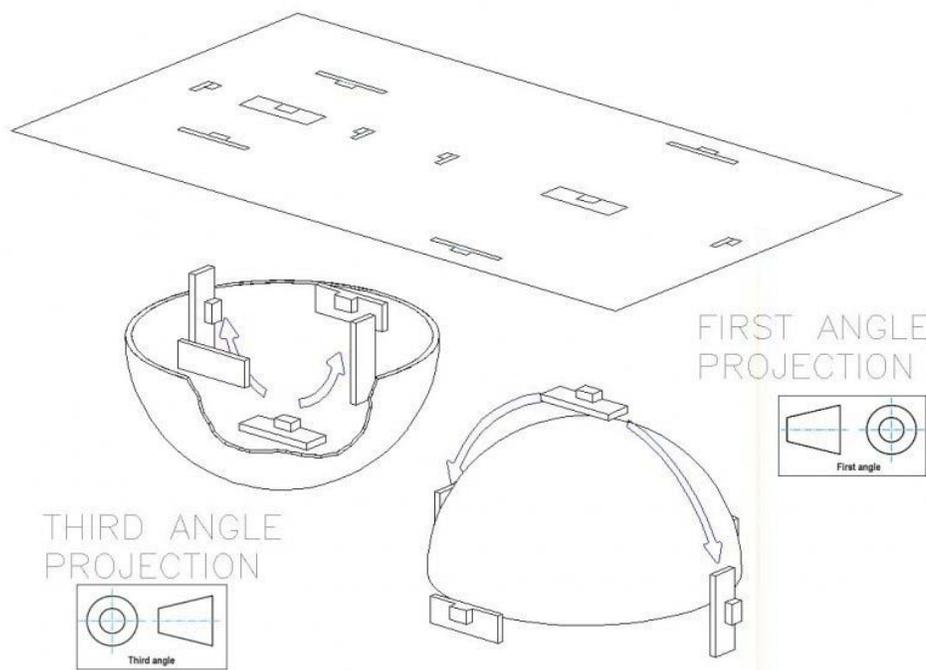
GD&T 1-3

Integrated Training Resources, Inc.

Word list: orthographic

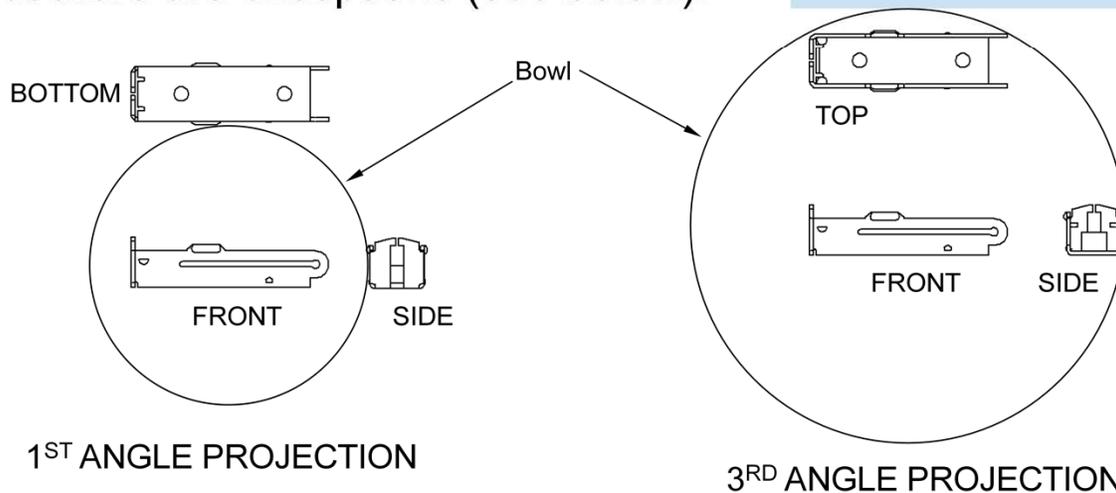
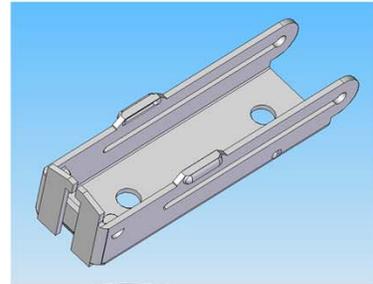
When multiple views are shown on a drawing they are related to each other orthographically, that is, at 90 degrees to each other. How the views are rotated, however, depends on the relational system that has been chosen for the drawing.

Either of two systems are typically used: 1st angle projection or 3rd angle projection. As shown in the illustration below, with 1st angle projection, the views are established based on the part rotating outside the bowl. 3rd angle projection views are established based on the part rotating inside the bowl.

Source: <http://cad-notes.com/2011/10/first-angle-and-third-angle-projection-explained/>

Example Views in Both Systems *for*

- Both systems begin with the front view and rotate the part _____ to establish related views.
- 3rd Angle is usually used on ASME drawings, 1st Angle with ISO drawings, but there are exceptions (see below).



Word list: 90°

First-angle projection is standard throughout Europe (excluding the UK) and Asia. First-angle projection used to be common in the UK, and may still be seen on older design drawings, but has now fallen into disuse in favor of third-angle projection.

Third-angle is standard in the United Kingdom (BS 8888:2006 specifies it as the default projection system), USA ([ASME Y14.3-2003](#) specifies it as the default projection system), Canada, Australia, and Japan.

Beware that the use of 1st or 3rd angle projection is typically based primarily on the country where the design was created or where the drawing was released. However, many multinational companies work with drawings based on both systems, such that both systems may be used on drawings within the same company, division, or even within one product family. And some companies mandate that one system be used for all drawings in the company regardless of the country from where they are released, or regardless of whether they are dimensioned in inches or millimeters.

Several points should be emphasized:

- Neither system is inherently better than the other.
- Both systems will continue to be used for the foreseeable future.
- Only one system should be used on any one drawing; systems should not be mixed on the same drawing.
- All drawing developers and users, including engineers, designers, manufacturing personnel, vendors, suppliers, and machine shops must understand the difference between the two systems and be competent in their proper interpretation.

Engineering Drawing Elements $\int t_r$

Engineering drawings include:

- _____ to define the geometry and shape.
- _____, _____, notes, and symbols to define size, location, orientation, and form of features.
- _____ and symbols to specify materials, treatments, and surface finish requirements.
- Notes and specifications to define _____, reliability, durability, and appearance requirements.
- _____ and revision block to provide traceability and change management information.
- Additional interpretation _____.

Word list: graphics, dimensions, tolerances, notes, performance, title block, aids.

Engineering drawings are either digital or physical documents that describe mechanical engineering designs via graphics, numbers, symbols, and words.

They employ graphics or pictures to define the part geometry and shape. The graphics are further described in terms of dimensions, tolerances, notes, and symbols to define the size, location, orientation, and form of the features. Notes and symbols are added to specify the part material specifications, materials treatment (such as heat treating), and surface finish requirements, including plating or anodization specifications.

Additional notes and specifications are added to define performance, interface, and durability requirements.

The title block and revision block provide traceability and change management information such as the document name, number, reference designs, revision level, release date, sheet size, general tolerances, material specifications, orthographic projection system, as well as the name of the designer, checker, and product engineer.

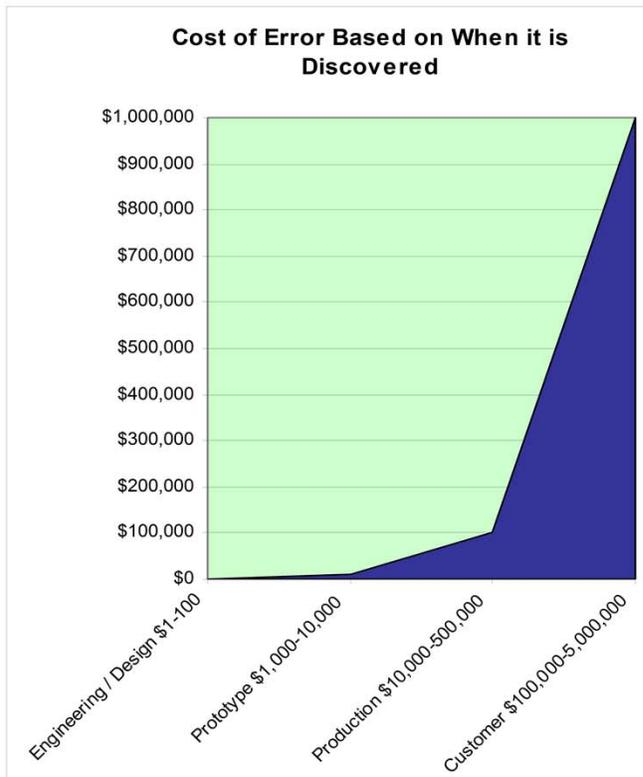
Additional interpretation aids may also be added including the following:

- Datum feature locations and degrees of freedom constrained.
- Navigation notes, including alpha-numeric cues on the border of the drawing.
- References to applicable drawing standards, guidelines, and specifications.
- Keys to interpretation of orthographic projections.
- Keys to interpretation of geometric tolerances or other specifications.
- Identification and designation of special or critical characteristics.

Cost of Drawing Errors



- _____ decisions impact all other program costs.
- The cost of errors and missed opportunities is _____ if they are not identified in the _____ stage.
- Most errors can be _____.
- Many errors are not fixed due to lack of _____ and money.
- _____: drawings errors must be prevented.

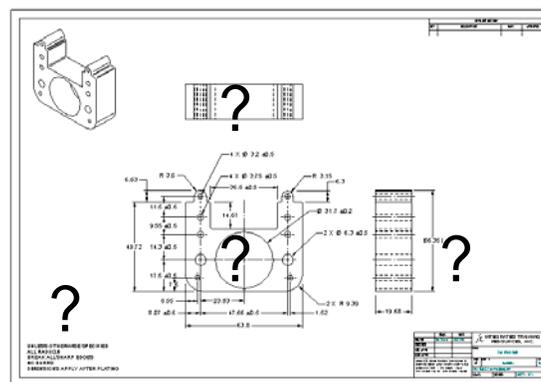


Word list: design, exponential, design, prevented, time, conclusion.

Engineering drawings must be complete; not ambiguous or open to interpretation; technically correct, and correct with respect to the functional requirements, including the final assembly requirements, the internal component interfaces, the manufacturing and assembly interfaces, and inspection and measurement requirements.

Thousands of drawings exist that are “technically correct” with respect to the component in questions, but have serious errors with respect to the design intent within the context of the application, tooling interfaces, and dimensioning and tolerancing logic. Here are some of the most common drawing errors, most of which can be prevented:

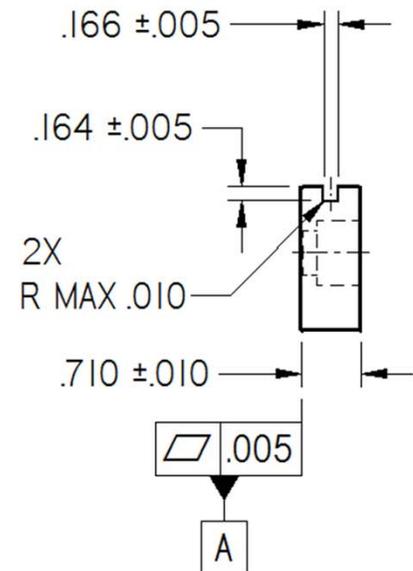
- Missing dimensions and/or tolerances
- Arbitrary dimensioning
- Arbitrary tolerancing
- Incorrect tolerance stacks
- Incorrect or illogical datum features
- Incorrect or illogical datum reference frames
- Tolerances too small or too large
- Illegal dimensioning and tolerancing
- Wrong combinations of coordinate and GD&T
- Wrong use of modifiers
- Missing specifications and notes
- Overuse or misapplication of GD&T



Dimensions and Tolerances

$$\int t_r$$

- A _____ is a numerical value expressed in appropriate units of measure and used to define the size, location, geometric characteristic, or surface texture of a part or part feature.
- A _____ is the total amount a specific dimension is permitted to vary. The tolerance is the difference between the maximum and minimum limits.
- Dimensions and tolerances are used to define _____,
_____,
_____, and
_____.



Word list: dimension, tolerance, size, location, orientation, form

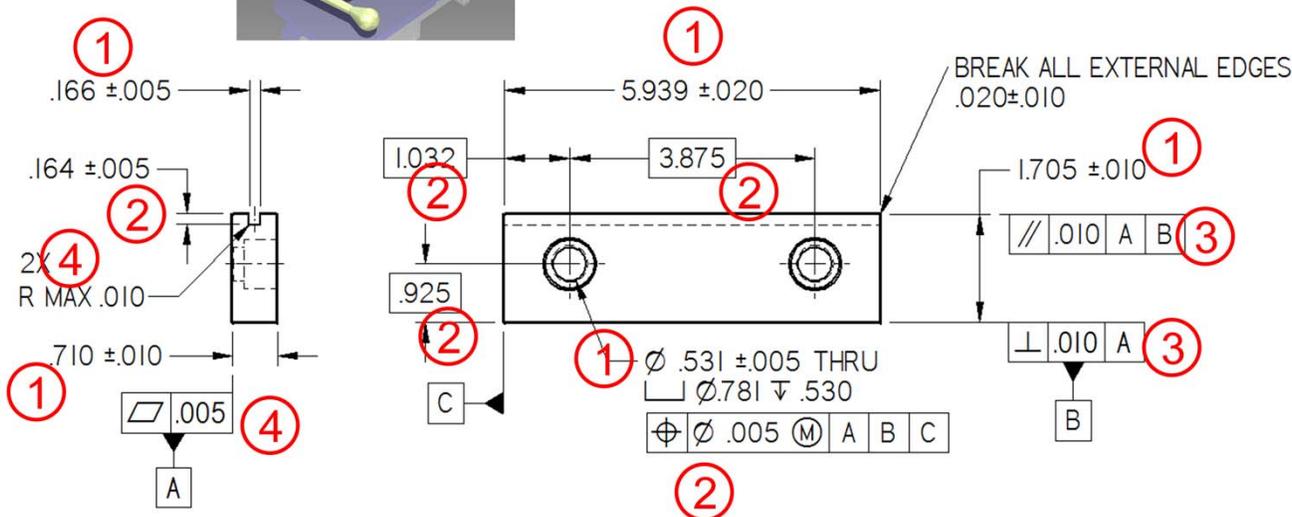
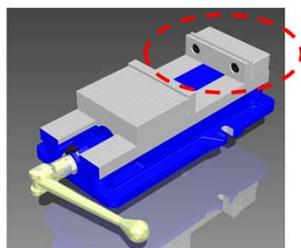
Dimensions typically describe the *desired* value of the size, location, orientation, or form of a part or part feature. Due to variation in products and processes, there is no such thing as perfect; there will always be variation. So there is a need to define what is the acceptable amount of variation, or deviation, from the target specification. The function of the tolerance is to define the maximum and/or minimum limits of variation from the target specification.

Note: reference dimensions, or those dimensions or specifications specified as maximum or minimum do not necessarily have an explicit tolerance associated with them. The tolerance may not be applicable, or may be implied by the limits of the design itself. For example, a radius specification of "Max 0.5mm" does not have an explicit tolerance on the specification. The minimum radius value is 0mm; the implied tolerance, therefore, is 0.5mm, but no explicit target value for the radius has been defined.

The four general categories of features that might be defined using dimensions and tolerances:

- S Size
- L Location
- O Orientation
- F Form

Size, Location, Orientation, Form *It_r*



Dimensions can be used to define four major types of part characteristics; the following types correspond to the circled numbers in the above drawing: 1. Size; 2. Location; 3. Orientation, and 4. Form.

Size

Size dimensions can be used to define cylindrical features such as holes or shafts, and any other features that have two opposed points or surfaces (see Feature of Size). Size dimensions in the drawing above include the .166+/- .005 width of the slot in the left view, the 5.939+/- .020 width in the front view, and the through hole and spot face diameter specifications. Notice that size dimensions define the size of features, not necessarily their location or orientation. The clearance holes, for example requires additional dimensions to define their location with respect to the rest of the part.

Location

Location dimensions can be used to locate or position surfaces, holes, shafts, radii or any other types of part features. Location dimensions are often candidates for use of geometric tolerances, but coordinate tolerances are appropriate in many cases. The basic dimensions define the location of the holes with respect to the bottom and side of the part, which then interpreted based on the position tolerance in the feature control frame below the hole specification.

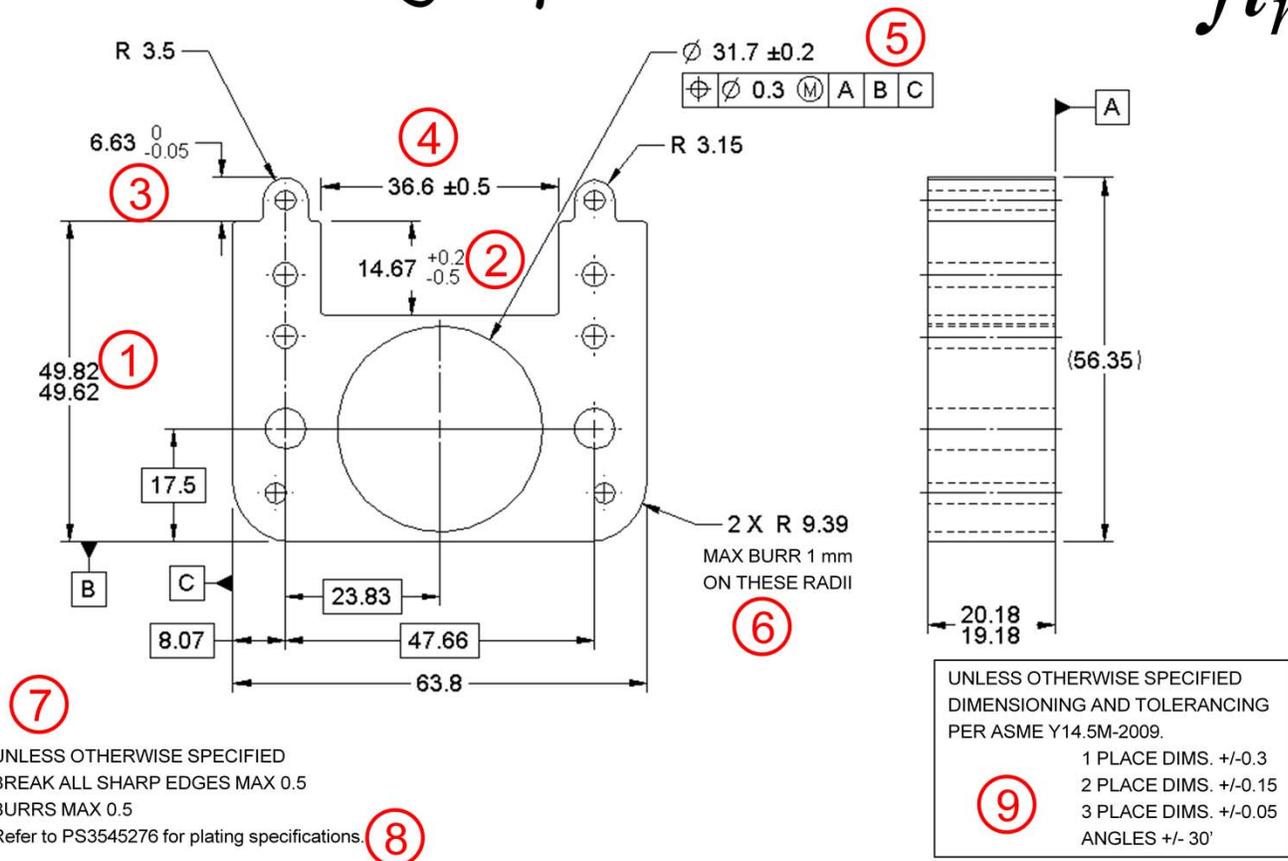
Orientation

Orientation dimensions are often implied, and may be toleranced with a geometric tolerance as with the perpendicularity and parallelism tolerances shown above.

Form

Form variation is often controlled by size tolerances, but can be added to clarify or refine form requirements for important part features. Both the surface finish callout and the radius specifications define form requirements, as does the flatness tolerance applied to the datum A surface.

Tolerancing Options



Copyright 1998-2015 ITR

GD&T 1-10

Integrated Training Resources, Inc.

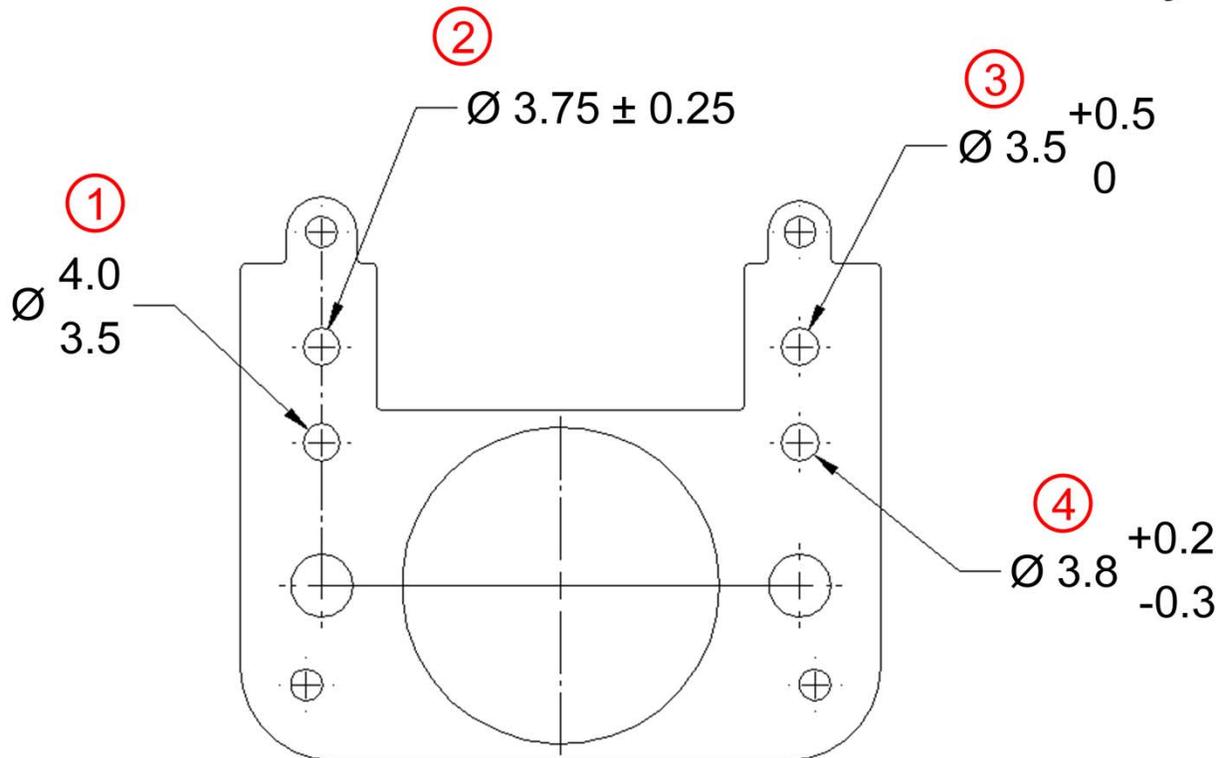
Tolerances may be expressed as follows. The first four are possible coordinate tolerancing expressions.

1. Direct limit dimensioning in which the tolerance is implied.
2. Dimension with unequal bilateral tolerance.
3. Dimension with unilateral tolerance.
4. Target dimension with equal bilateral tolerance. (Note: this method is most common, most widely understood, and typically the best tolerancing option considering DFM and inspection.)
5. Geometric tolerancing via a Feature Control Frame.
6. Within a local note applied to a specific dimension or feature. Note that this note is also an example of a maximum specification which is another type of coordinate tolerancing expression.
7. Within a general note on the field of the drawing.
8. Within another engineering document or tolerancing standard (e.g. ISO 2768)
9. Within a general tolerance block or title block.

The nine methods above are grouped in their typical hierarchy in terms of interpretation. In other words a local dimension and/or tolerance (examples 1-6) takes precedence over a general note (examples 7-8), which in turn takes precedence over the even more general title block tolerances, which are often used as a safeguard when any dimension and/or tolerance may have been left without a tolerance or where a dimension is not shown on the drawing.

Application Note: Most seasoned mechanical engineers and designers are in agreement that those features, characteristics, dimension, and tolerances that are critical to quality should be explicitly shown on the drawing. If it isn't on the drawing, it doesn't exist in any meaningful sense. In practice, only those dimensions that are shown on the drawing will receive any attention during prototyping, tool and equipment qualification, or production approval and inspection.

Coordinate Tolerance Expressions $\int t_r$



Typical coordinate tolerancing expressions fall under two main categories: Limit dimensioning and Plus/minus tolerancing. The circled numbers in the drawing above indicate four typical conventions for applying limit and plus/minus tolerancing.

1. Limit Dimensioning – Limit dimensions are characterized by two values that establish the maximum and minimum limits within which the dimension may fall. The maximum limit is placed above the minimum limit. When expressed on a single line, the minimum limit value is written first, followed by a dash and the maximum limit value (e.g. 3.5-4.0). Also known as max/min or min/max dimensioning.
2. Equal Bilateral Plus/Minus Tolerancing – With this method the dimension is written first followed by a plus/minus expression of tolerance (e.g. 3.75 ± 0.25). As the name implies, an equal tolerance for variation is provided on either side of the target or nominal dimension. This tolerancing convention is by far the most popular and widely accepted.
3. Unilateral Plus/Minus Tolerancing – The dimension is expressed as either the maximum or minimum limit of the tolerance followed by a one-sided tolerance value (e.g. 4 +0.05 –0).
4. Unequal Bilateral Plus/Minus Tolerancing – As with equal bilateral tolerancing the dimension is written first followed by a plus/minus expression of tolerance. The tolerance, however, is not equal on either side of the target or nominal dimension. (e.g. 3.8 +0.2 –0.3).

Note that in all four examples the tolerance and the maximum and minimum limits are the same, but the expressions of these specifications are different. The unilateral and unequal bilateral tolerancing practices may be convenient for designers and product engineers, but do not typically reflect the statistical distributions of production parts, and may be confusing when attempting to define the *target* for the specification. Tolerancing expressions should consider the potential interpretation and impact on all stakeholders, including suppliers, manufacturing, and customers.

ANSI/ASME Limits and Fits $\int t_r$

- ANSI/ASME B4.1 makes recommendations on preferred and _____ sizes, allowances, tolerances, and fits for use on mating parts for both inches and millimeters.
- Note that values must typically be modified to accommodate geometric tolerance of location and orientation.

Nominal Size Range, Inches	Class LC 1			Class LC 2			Class LC 3			Class LC 4			Class LC 5		
	Clearance ^a	Standard Tolerance Limits		Clearance ^a	Standard Tolerance Limits		Clearance ^a	Standard Tolerance Limits		Clearance ^a	Standard Tolerance Limits		Clearance ^a	Standard Tolerance Limits	
		Hole H6	Shaft h5		Hole H7	Shaft h6		Hole H8	Shaft h7		Hole H10	Shaft h9		Hole H7	Shaft g6
Over To	Values shown below are in thousandths of an inch														
0- 0.12	0 0.45	+0.25 0	0 -0.2	0 0.65	+0.4 0	0 -0.25	0 1	+0.6 0	0 -0.4	0 2.6	+1.6 0	0 -1.0	0.1 0.75	+0.4 0	-0.1 -0.35
0.12- 0.24	0 0.5	+0.3 0	0 -0.2	0 0.8	+0.5 0	0 -0.3	0 1.2	+0.7 0	0 -0.5	0 3.0	+1.8 0	0 -1.2	0.15 0.95	+0.5 0	-0.15 -0.45
0.24- 0.40	0 0.65	+0.4 0	0 -0.25	0 1.0	+0.6 0	0 -0.4	0 1.5	+0.9 0	0 -0.6	0 3.6	+2.2 0	0 -1.4	0.2 1.2	+0.6 0	-0.2 -0.6
0.40- 0.71	0 0.7	+0.4 0	0 -0.3	0 1.1	+0.7 0	0 -0.4	0 1.7	+1.0 0	0 -0.7	0 4.4	+2.8 0	0 -1.6	0.25 1.35	+0.7 0	-0.25 -0.65
0.71- 1.19	0 0.9	+0.5 0	0 -0.4	0 1.3	+0.8 0	0 -0.5	0 2	+1.2 0	0 -0.8	0 5.5	+3.5 0	0 -2.0	0.3 1.6	+0.8 0	-0.3 -0.8
1.19- 1.97	0 1.0	+0.6 0	0 -0.4	0 1.6	+1.0 0	0 -0.6	0 2.6	+1.6 0	0 -1	0 6.5	+4.0 0	0 -2.5	0.4 2.0	+1.0 0	-0.4 -1.0
1.97- 3.15	0 1.2	+0.7 0	0 -0.5	0 1.9	+1.2 0	0 -0.7	0 3	+1.8 0	0 -1.2	0 7.5	+4.5 0	0 -3	0.4 2.3	+1.2 0	-0.4 -1.1
3.15- 4.73	0 1.5	+0.9 0	0 -0.6	0 2.3	+1.4 0	0 -0.9	0 3.6	+2.2 0	0 -1.4	0 8.5	+5.0 0	0 -3.5	0.5 2.8	+1.4 0	-0.5 -1.4
4.73- 7.09	0 1.7	+1.0 0	0 -0.7	0 2.6	+1.6 0	0 -1.0	0 4.1	+2.5 0	0 -1.6	0 10.0	+6.0 0	0 -4	0.6 3.2	+1.6 0	-0.6 -1.6

Copyright 1998-2015 ITR

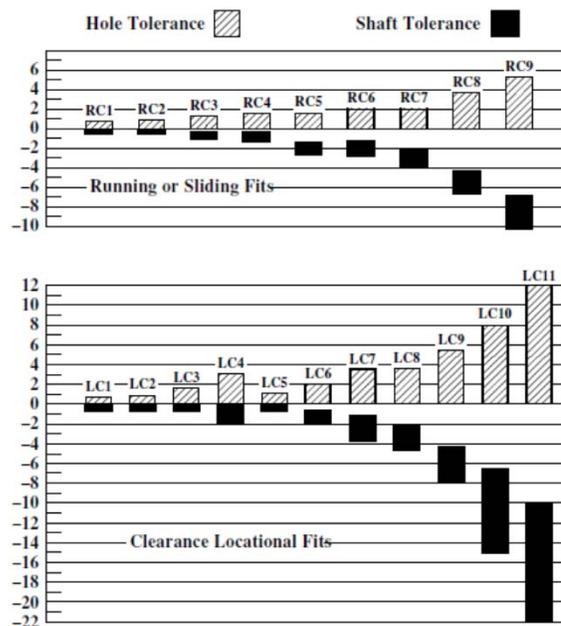
GD&T 1-12

Integrated Training Resources, Inc.

Word list: standard

This American National Standard for Preferred Limits and Fits for Cylindrical Parts, ANSI/ASME B4.1-1967 (R2009), presents definitions of terms applying to fits between plain (non threaded) cylindrical parts and makes recommendations on preferred sizes, allowances, tolerances, and fits for use wherever they are applicable. This standard is in accord with the recommendations of American-British-Canadian (ABC) conferences up to a diameter of 20 inches. The recommendations in the standard are presented for guidance and for use where they might serve to improve and simplify products, practices, and facilities. They should have application for a wide range of products.

For preferred fits, the nominal size of both mating parts is often the same value, regardless of the type of fit. Therefore unilateral and/or unequal bilateral tolerances are typically employed to achieve the proper allowances and fit.

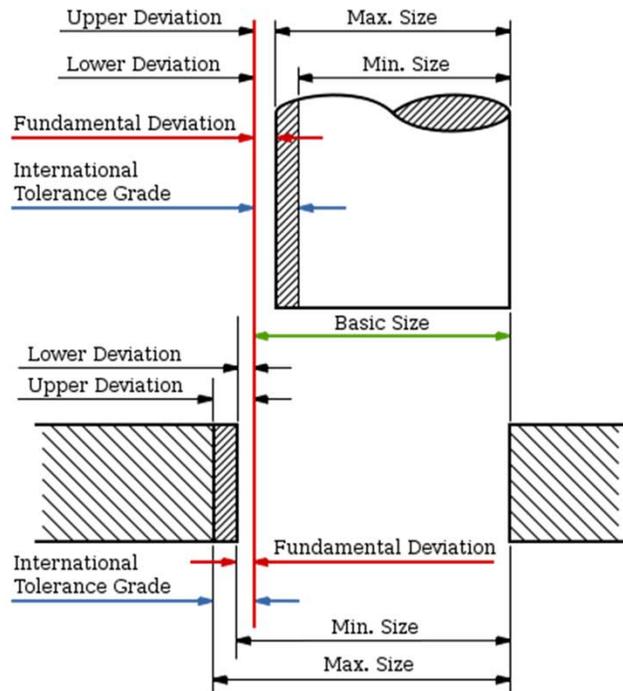
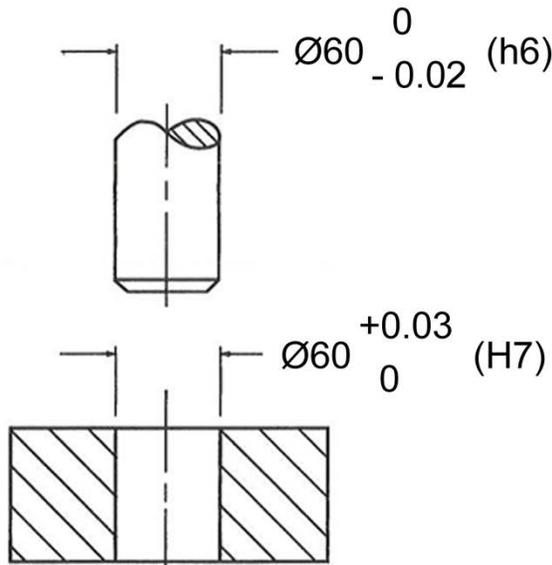


While this practice is convenient for designers and product engineers, it may be preferable to convert these specifications to equal bilateral tolerances, which are typically better understood and more widely accepted globally.

ISO Limits and Fits



- Per ISO 286-1 and 2



Excerpt from ISO 286-2 tables

Nominal hole sizes (mm)													
over	3	6	10	18	30	40	50	65	80	100	120	140	160
inc.	6	10	18	30	40	50	65	80	100	120	140	160	180
micrometres ($10^{-6}m$)													
H6	+3 0	+9 0	+11 0	+13 0	+16 0	+19 0	+22 0	+25 0					
H7	+12 0	+15 0	+18 0	+21 0	+25 0	+30 0	+35 0	+40 0					
H8	+18 0	+22 0	+27 0	+33 0	+39 0	+46 0	+54 0	+63 0					

Description	Hole	Shaft
Loose Running	H11	c11
Free Running	H9	d9
Loose Running	H11	c11
Easy Running - Good quality easy to do-	H8	f8
Sliding	H7	g6
Close Clearance - Spigots and locations	H8	f7
Location/Clearance	H7	h6
Location- slight interference	H7	k6
Location/Transition	H7	n6
Location/Interference- Press fit which can be separated	H7	p6
Medium Drive	H7	s6
Force	H7	u6

Metric Conventions

	Correct	Incorrect <i>It_r</i>
For dimensions or tolerances less than one _____, a zero precedes the decimal point.	0.8±0.1	.8 ±.1
For _____ number dimensions, neither the decimal point nor the zero is shown.	12 ±0.25	12.0 ±0.25
For values less than one millimeter, the last digit to the right of the decimal is not followed by a _____.	10.5±0.1	10.50 ±0.10
For _____ tolerancing, both the plus and minus values have the same number of decimal places.	18.6 ^{+0.25} -0.30	18.6 ^{+0.25} -0.3
For _____ tolerancing, a single zero is shown without a plus or minus sign.	18.85 ⁰ -0.55	18.85 ^{+0.00} -0.55
For _____ dimensions, both the max and min limits carry the same number of decimal places.	∅ 8.800 8.765	∅ 8.8 8.765

Copyright 1998-2015 ITR

GD&T 1-14

Integrated Training Resources, Inc.

Word list: millimeter, whole, zero, bilateral, unilateral, limit.

These conventions yield the following expressions for the specification 15.8-16.0:

Expressed as a target dimension with an equal bilateral tolerance would be 15.9 ± 0.1

Expressed as as a dimension of 16 with a unilateral tolerance would be $16 \begin{matrix} 0 \\ -0.2 \end{matrix}$

Expressed as a dimension of 15.8 with a unilateral tolerance would be $15.8 \begin{matrix} +0.2 \\ 0 \end{matrix}$

Expressed as a dimension of 15.85 with an unequal bilateral tolerance would be $15.85 \begin{matrix} +0.15 \\ -0.05 \end{matrix}$

Use of the millimeter as the unit of measure is supported by both ASME and ISO standards. The use of the inch as the unit of measure is only supported by ASME standards, not ISO. Inch drawings must therefore be based on ASME standards.

Inch Conventions



	Correct	Incorrect
For values less than one inch, a zero is _____ used before the decimal point.	.500±.005	0.5 ±0.005
All dimensions and tolerances shall be to the _____ number of decimal places, zeros being added to the right of the decimal place where necessary.	.750 ±.005 3.600 +.025 -.020 2.853 +.000 -.004	.75 ±.005 3.6 +.025 -.02 2.853 0 -.004
For limit dimensions, both the max and min values shall have the same number of decimal places, zeros being added to the right of the decimal place to maintain _____.	∅ 1.800 1.795	∅ 1.8 1.795

Copyright 1998-2015 ITR

GD&T 1-15

Integrated Training Resources, Inc.

Word list: not, same, uniformity.

It is advisable to use either metric or inch system to communicate dimensioning and tolerancing requirements. At times it may be necessary to use both types of designations on the same engineering drawing. In these cases the suffixes “mm” or “in” should be added after the specification as appropriate to avoid misinterpretation.

Evolution of GD&T



Year	Key Events
1935	ASA publishes American Drawing and Drafting Room Practices.
1940	Chevrolet Draftman's Handbook discusses position tolerancing.
1944	Cylindrical position tolerance system formalized by British.
1945	U.S. Army publishes manual introducing symbols rather than notes.
1946	2 nd Edition of American Drawing and Drafting Room Practices.
1948	British Military publishes Dimensional Analysis of Engineering Design.
1949	U.S. Military publishes MIL-STD-8 on dimensioning and tolerancing.
1953	U.S. Military publishes MIL-STD-8A with 7 GD&T symbols.
1957	ASA publishes first American standard devoted to GD&T.
1966	ANSI Y14.5 standard on Dimensioning and Tolerancing first published.
1969	ISO 1101 standard on Geometric Tolerancing published.
1973	ANSI Y14.5 updated to replace all tolerancing notes with symbols.
1982	ANSI Y14.5M updated based on metric units.
1983	ISO 1101 updated to include additional graphics.
1994	ASME Y14.5M updated creating closer harmonization with ISO standards.
2004	ISO 1101 enhanced with additional technical and graphical clarification.
2009	ASME Y14.5 updated to reorganize the information to facilitate interpretation and application.

Copyright 1998-2015 ITR

GD&T 1-17

Integrated Training Resources, Inc.

Evolution of GD&T

The science of metrology is as old as mankind and goes hand-in-hand with other engineering tools and disciplines such as the use of engineering drawings. The evolution of GD&T in particular has its roots in the industrial revolution that began in the 1700s and ushered in the need for interchangeable parts. This requirement, in turn, led to a need for more precision and accuracy in dimensional requirements. First dimensions, then tolerances, were added to engineering drawings to better communicate functional requirements.

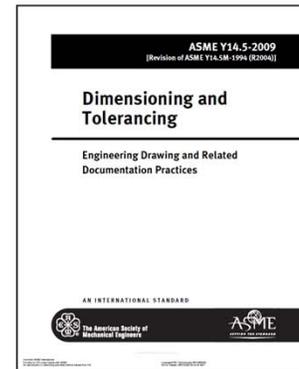
The first serious attempts at codifying and standardizing dimensioning and tolerancing practices for a broad cross sector of industries began after World War I, and received even greater attention and definition during World War II by both the US and British military.

The initial focus of GD&T was to create logical tolerance zones, especially for position tolerances. The other types of tolerances, symbols, and modifiers have been added to the point when, today, there are well developed standards and practices used internationally. The two most popular series of standards are published by the American Society of Mechanical Engineers (ASME) and the International Organisation for Standardization (ISO).

Dimensioning Standards



- Today the two major groups of dimensioning and tolerancing standards are published by _____ and _____.
- The principal ASME standard is _____.
- The principal ISO standard is _____, but must be supplemented with at least _____ additional standards to fully address the most common dimensioning and tolerancing practices.
- For unspecified tolerances, ASME drawings typically show general tolerances in the _____.
- ISO 2768 (parts 1 and 2) are typically the basis for general linear, angular, and geometric tolerances on ISO drawings unless otherwise specified.



Word list: ASME, ISO, Y14.5, 1101, 10, title block.

Dimensioning Standards

Today the two major groups of dimensioning and tolerancing standards are published by the American Society of Mechanical Engineers (ASME) and the International Organization for Standardization (ISO).

The principal ASME standard for dimensioning and tolerancing is Y14.5. This standard is relatively comprehensive in terms of product dimensioning and tolerancing, but must be supplemented with various other standards for a more complete coverage of specific geometries (such as threads, gears, forgings, casting, etc.) and for important practices such as inspection and gaging which are not addressed in Y14.5. See the glossary for additional standards and references.

The principal ISO standard is 1101, but must be supplemented with at least 10 additional standards to fully address the most common dimensioning and tolerancing practices. See the glossary for additional standards and references.

The majority of the fundamental principles of dimensioning and tolerancing are the same under both sets of standards. The standards differ in the interpretation of some of the symbols (e.g. concentricity and symmetry), in the application of some important concepts (e.g. the envelope principle), and in how various dimensioning and tolerancing notations are shown on the drawing. For the most part, however, when someone has a grasp of the concepts, applications, and interpretations of dimensioning and tolerancing based on one of these sets of standards, the understanding is transferable in large part to drawings developed under the other set of standards.

Fundamental Rules from Y14.5 $\int t_r$

NOTES

- (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 ASME Y14.1 and ASME Y14.1M.
- (b) Dimensioning and tolerancing shall be complete so there is full understanding of the characteristics of each feature. Values may be expressed in an engineering drawing or in a CAD product definition data set. See ASME Y14.41. Neither scaling (measuring 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, 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) Nonmandatory processing dimensions shall be identified by an appropriate note, such as "NONMANDATORY (MFG DATA)." Examples of nonmandatory data are processing dimensions that provide for finish allowance, shrink allowance, and other requirements, provided the final dimensions are given on the drawing.
- (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.

Notes

Fundamental Rules, Continued \int_{tr}

NOTES

- (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 2D orthographic drawing at right angles and no angle is specified. See para. 2.1.1.3.
- (j) A 90° basic angle applies where center lines of features in a pattern or surfaces shown at right angles on a 2D orthographic drawing are located or defined by basic dimensions and no angle is specified. See para. 2.1.1.4.
- (k) A zero basic dimension applies where axes, center planes, or surfaces are shown coincident on a drawing, and geometric tolerances establish the relationship among the features. See para. 2.1.1.4.
- (l) Unless otherwise specified, all dimensions and tolerances are applicable at 20°C (68°F) in accordance with ANSI/ASME B89.6.2. Compensation may be made for measurements made at other temperatures.
- (m) Unless otherwise specified, all dimensions and tolerances apply in a free-state condition. For exceptions to this rule see paras. 4.20 and 5.5.
- (n) Unless otherwise specified, all tolerances apply for full depth, length, and width of the feature.
- (o) 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 (e.g., a detail drawing) is not mandatory for that feature at any other level (e.g., an assembly drawing).
- (p) Where a coordinate system is shown on the drawing, it shall be right-handed unless otherwise specified. Each axis shall be labeled and the positive direction shall be shown.

Notes

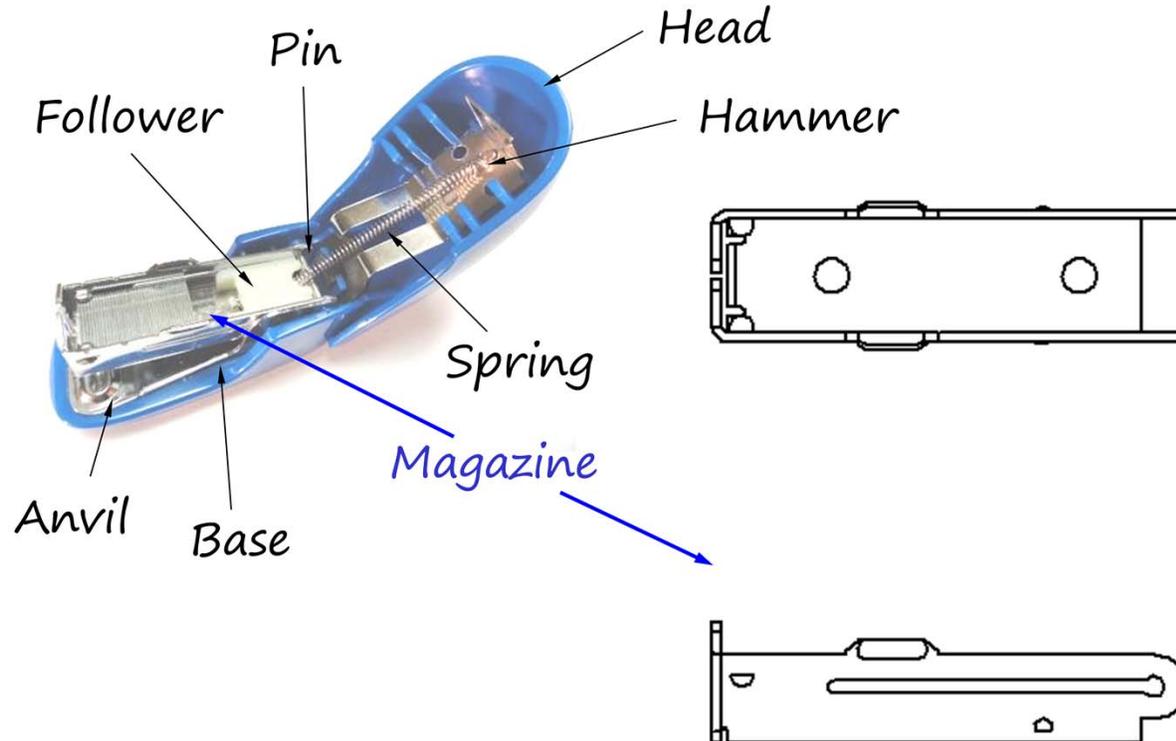
Basic Concepts from ISO 1101 $\int t_r$

From ISO 1101

- 4.1** Geometrical tolerances shall be specified in accordance with functional requirements. Manufacturing and inspection requirements can also influence geometrical tolerancing. NOTE Indicating geometrical tolerances on a drawing does not necessarily imply the use of any particular method of production, measurement or gauging.
- 4.2** A geometrical tolerance applied to a feature defines the tolerance zone within which that feature shall be contained.
- 4.3** A feature is a specific portion of the workpiece, such as a point, a line or a surface; these features can be integral features (e.g. the external surface of a cylinder) or derived (e.g. a median line or median surface). See ISO 14660-1.
- 4.4** According to the characteristic to be toleranced and the manner in which it is dimensioned, the tolerance zone is one of the following:
- the space within a circle;
 - the space between two concentric circles;
 - the space between two equidistant lines or two parallel straight lines;
 - the space within a cylinder;
 - the space between two coaxial cylinders
 - the space between two equidistant surfaces or two parallel planes;
 - the space within a sphere.
- 4.5** Unless a more restrictive indication is required, for example by an explanatory note (see Figure 8), the toleranced feature may be of any form or orientation within this tolerance zone.
- 4.6** The tolerance applies to the whole extent of the considered feature unless otherwise specified as in Clauses 12 and 13.
- 4.7** Geometrical tolerances which are assigned to features related to a datum do not limit the form deviations of the datum feature itself. It may be necessary to specify tolerances of form for the datum feature(s).

Notes

Mini Stapler Case Study

$$\int t_r$$


Basic Steps to Functional Dimensioning and Tolerancing

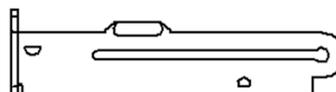
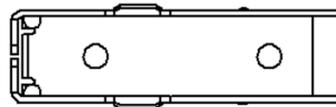
1. Define the **key product outputs** of the overall design in terms of performance, safety, fit, function, customer interface, durability, reliability, appearance, serviceability, DFx, etc. Identify key inputs in terms of components, features, relationships, and specifications that have a direct impact on outputs.
2. Identify **part-to-part interfaces**, especially noting where the parts touch. These are potential critical characteristic candidates as well as datum candidates. At a minimum, these part-to-part interfaces and features should be defined dimensionally as appropriate.
3. Identify **feature-to-feature relationships** within each component. These are also potential critical characteristic candidates as well as datum candidates. At a minimum, these feature-to-feature relationships should be defined dimensionally as appropriate.
4. Identify **part-to-process interfaces**. Study how each component in the assembly is fabricated or manufactured; this will reveal **Design for Manufacturing** considerations and requirements. Then analyze the assembly and test process; this will reveal **Design for Assembly** considerations and opportunities. **Inspection and Gaging** practices should support functional requirements, not conflict with them. These relationships should be also be comprehended dimensionally as appropriate.
5. Apply dimensioning and tolerancing to each component, sub assembly, and assembly as appropriate based on the above requirements using the following basic 3-step method:
 1. Datums
 2. Dimensions
 3. Tolerances

Multiple loops of analysis and application, tolerancing calculations, and other engineering/business factors and constraints must be considered; the above steps provide only a fundamental approach.

The 5 Basic Steps to Functional Dimensioning and Tolerancing



1. Define key product outputs and requirements.
2. Identify _____ interfaces.
3. Identify _____ relationships.
4. Identify _____ interfaces:
 - Fabrication and Manufacturing
 - Assembly
 - Inspection and Test
5. Apply _____ and Tolerancing
 1. Datums
 2. Dimensions
 3. Tolerances



Word list: part-to-part, feature-to-feature, part-to-process, dimensioning.

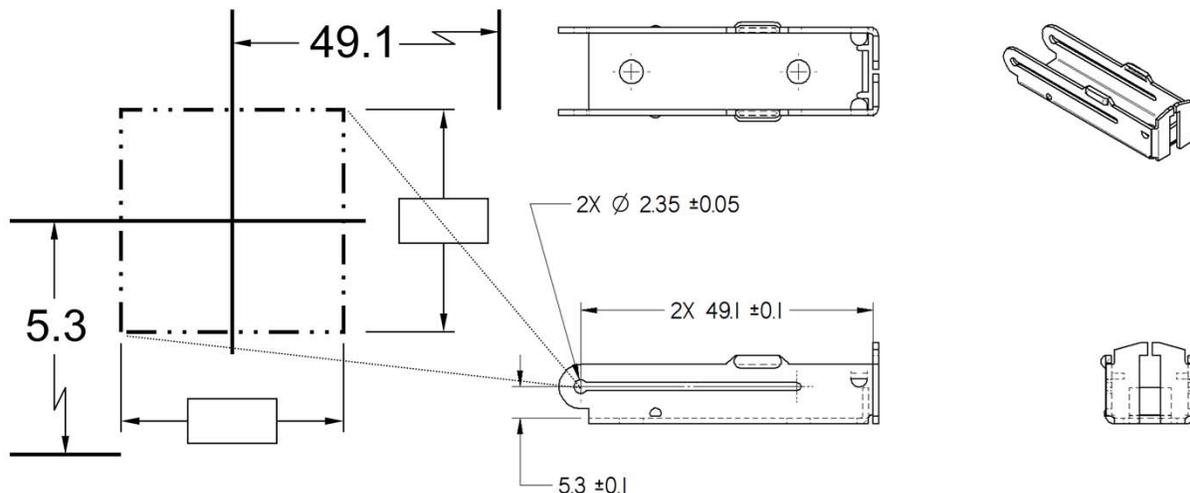
Exercise: Apply functionally appropriate dimensions to the magazine drawing views above using X.X.

The ITR GD&T Application Approach

Below is a more expanded version of the general steps for applying dimensioning and tolerancing to new designs and drawings, and for reviewing and improving existing drawings and designs:

1. Define overall design requirements for the finished product, including performance, safety, durability, etc.
2. Identify governing Part-to-Part, Part-to-Process, and Feature-to-Feature Relationships
3. Define datums - contact and constraint relationship with respect to the 6 degrees of freedom for each part
4. Define dimensions - Part-to-Part, Part-to-Process, and Feature-to-Feature nominal size, location, and orientation requirements. Orientation and location relationships are related to datums established in Step 3.
5. Define proposed manufacturing and assembly processes and capabilities, including DFM/DFA requirements.
6. Calculate and apply tolerances based on information from Steps 1-5 and tolerance analysis and allocation methods.
7. Confirm initial datum, dimensioning, and tolerancing using Rules Based Design Guidelines, GD&T formulas, and drawing review checklists.
8. Optimize dimensioning and tolerancing based on actual data from prototype and production experience, and tolerance analysis and Design for Six Sigma methods, including digital tolerance analysis tools.

Coordinate System Weaknesses $\int t_r$



1. _____ tolerance zones.
2. _____ tolerance values (no bonus).
3. _____ inspection instructions.

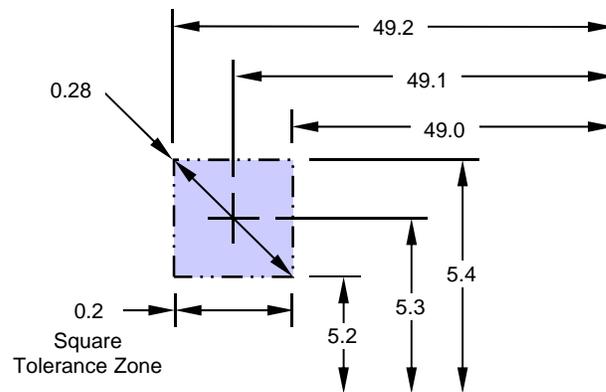
Word list: 0.2,0.2, illogical, fixed, ambiguous.

Coordinate dimensioning and tolerancing has been used on engineering drawings since the industrial revolution, and it continues to be the foundation of all dimensioning and tolerancing conventions worldwide. The size, location, orientation, and form are defined by dimensions, tolerances, and notes that have the *appearance* of precision and clarity, but are often open to interpretation. Coordinate tolerancing is appropriate for many applications including defining feature size, certain orientation relationships, chamfers, radii, and even locating certain part features. But there are several inherent weaknesses in using the coordinate dimensioning and tolerancing system exclusively.

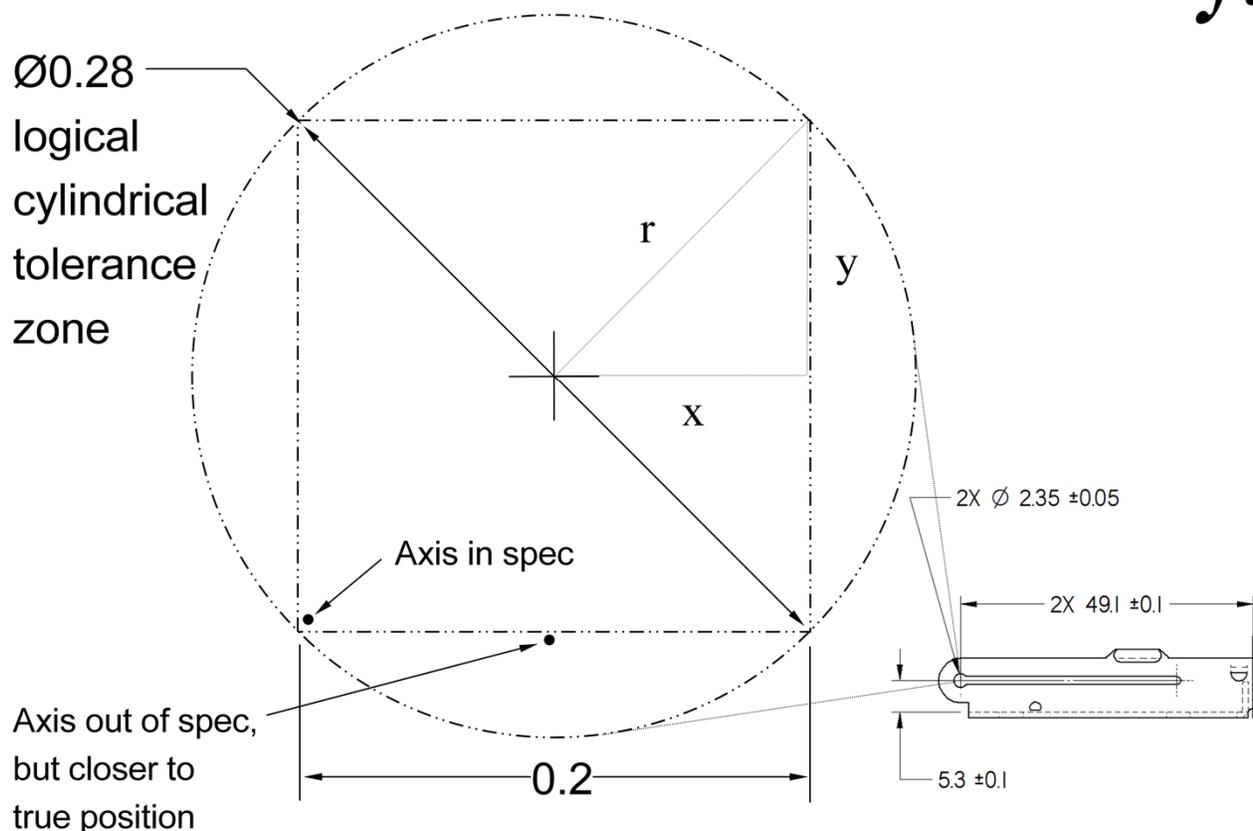
The 2.35 ±0.05 hole is located by two dimensions, 49.1 ±0.1 and 5.3 ±0.1, which yield planar limits at 5.2 and 5.4 from the bottom of the part, and 49.0 and 49.2 from the right side of the part. These four planar limits taken together create a 0.2 square tolerance zone.

This tolerance zone allows deviation of 0.1 horizontally or vertically from true position as defined by the nominal dimensions, but allows deviation of 0.14 diagonally. This tolerance zone shape is illogical and inappropriate for this application where the design is concerned with radial deviation of the axis from the target in all directions. As is the case for most diametral features of size, a cylindrical position tolerance is more logical and appropriate, but is not possible using coordinate tolerancing alone.

Illogical tolerance zones is the first major weakness of the coordinate dimensioning and tolerancing system.



Square vs. Cylindrical Zones

$$\int t_r$$


Copyright 1998-2015 ITR

GD&T 1-25

Integrated Training Resources, Inc.

The current interpretation of the print is that the axis of the hole must be no more than 0.1 away from true position as measured from the right side and bottom of the part. The resulting tolerance zone is a square that measures 0.2 x 0.2. A square tolerance zone does not accurately communicate the design requirements in this case because the specification is really concerned with radial variation from true position in all directions, not just along the X and Y axes. Note that both part axes depicted above logically meet the design requirement, but only the left axis is technically within the illogical square tolerance zone formed by the coordinate dimensioning and tolerancing system. If an equal radial tolerance is appropriate based on the distance of the corners of the current tolerance zone from true position, we can logically derive a cylindrical tolerance zone centered on true position. Use the root sum square (RSS) formulas to find the radius and diameter of the cylindrical tolerance zone:

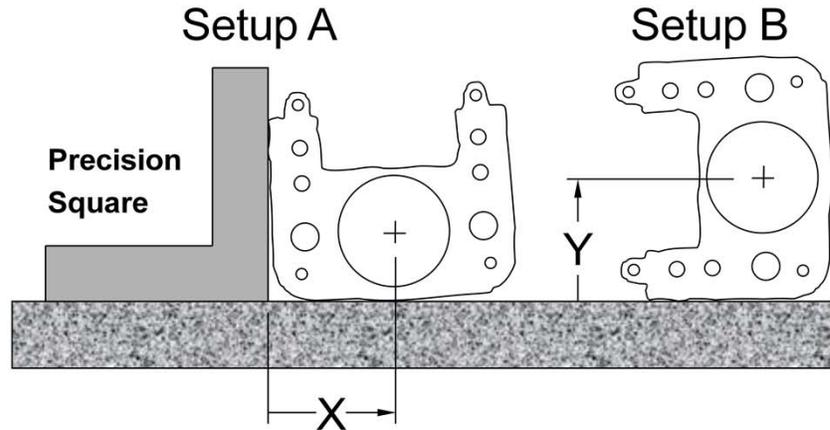
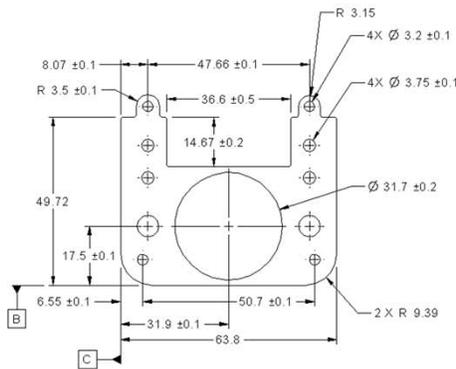
1. Radius = $\text{SQRT}(x^2 + y^2)$
2. Diameter = $2(\text{SQRT}(x^2 + y^2))$

If $x = y$, the following shortcut formula can also be used:

3. Diameter = $2x(1.414)$

Additionally, coordinate tolerances are *fixed in size*; they cannot grow take advantage of the relationship between size deviation and location deviation. For example, if the functional requirement of the hole is to assemble with the worst case mating part, the hole location is most critical when the hole is at its smallest size. As the actual hole size increases within the limits of size, the position tolerance can be increased by a corresponding amount (see Bonus Tolerance). Fixed size tolerances is the second weakness of the coordinate system.

Ambiguous Inspection Setups $\int t_r$



- _____ datums and datum reference frames create multiple possible inspection setups.
- In the examples above, measurements X _____ Y; both interpretations may be considered correct.
- Consequences: in-spec parts may be _____; out-of-spec parts may be accepted.

Word list: implied, ≠, rejected.

Finally, the coordinate dimensioning and tolerancing system is unable to fully describe many inspection requirements, especially those related to location and orientation dimensions and tolerances.

For example, to gage or measure the location of the center hole from the left side of the lamination stack part above, the inspector can choose between Setup A and Setup B. The measurement results from the two different setup will not be the same; this phenomenon is known as *measurement error*, and it is one of the major sources of variation in any manufacturing process.

Even if the inspector correctly guesses that the bottom and left side of the part might be used as starting points for measurements, or datums, the order or sequence of contact that these datums should have with the gage, fixture, or inspection equipment is still left open to interpretation.

To summarize, the three major shortcoming of the coordinate system are:

1. Illogical Tolerance Zones
2. Fixed-size Tolerance Zones
3. Ambiguous Inspection Setups

Here are just a few of the negative consequences of these weaknesses:

- Ambiguous specifications result in drawing misinterpretation, and engineering and manufacturing errors.
- Engineering, design, and manufacturing time is wasted on discussions, debates, and explanations.
- Good parts are rejected; bad parts are accepted.
- Manufacturing processes are not optimized to yield maximum quality and productivity.

Introduction to GD&T

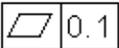


- GD&T is an _____ symbolic language that complements traditional coordinate dimensioning and tolerancing.
- GD&T addresses many of the inherent _____ of the coordinate system.
- GD&T encourages _____ dimensioning and tolerancing that comprehends design, manufacturing, and inspection requirements.

Word list: international, weaknesses, functional

Geometric Dimensioning and Tolerancing is an international engineering language that has evolved over the last half century to address the needs for clear, precise, and concise communication of dimensioning and tolerancing requirements on engineering drawings. The system is used to varying degrees in the majority of industrialized countries, including countries in North America, South America, Europe, Asia, and Australia. To the degree that mechanical engineering designs have been exported from North America and Europe to any other region of the world, GD&T has also been exported.

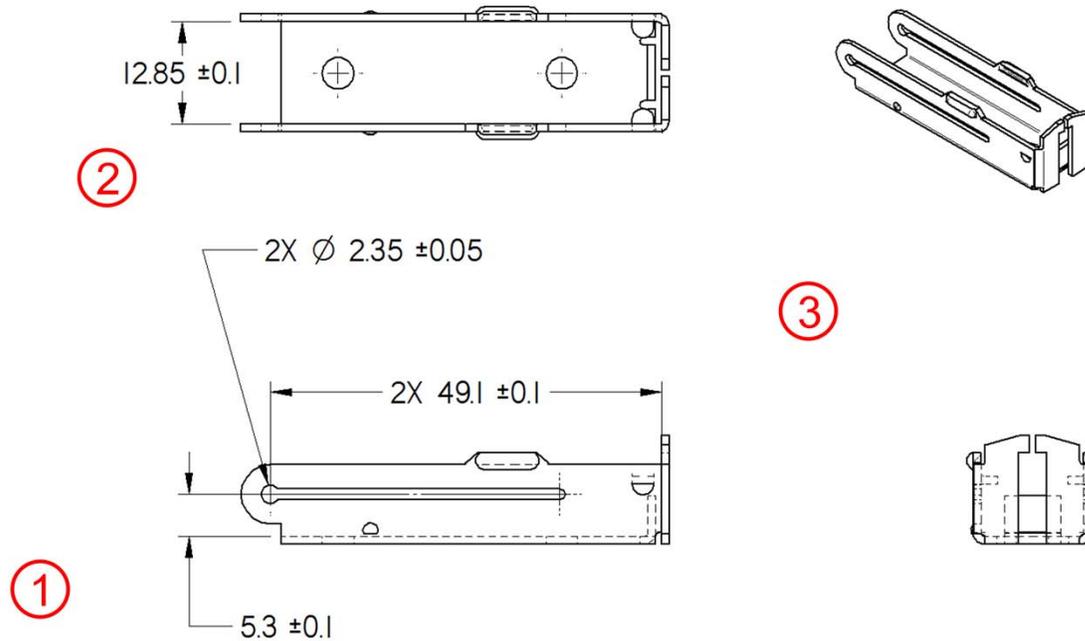
As with any other language, GD&T has its own symbols, syntax, and semantics. But unlike the coordinate dimensioning and tolerancing system, it does not rely primarily on notes to communicate detailed specifications. It can therefore be used in any country, regardless of the spoken or written language used in that country. For example, instead of using a note in English that reads “All points on this surface must lie within a tolerance zone formed by two parallel planar limits 0.1 apart with the upper limit being established from the surface high point plane,” the designer can simply add the following feature control frame:

GD&T provides any numb , mathematically defined tolerancing options that are far superior to the limited range of tolerance shapes afforded by the coordinate system alone. And while GD&T is not the solution to all engineering and design problems, it encourages dimensioning and tolerancing applications that reflect how the part performs or functions with respect to the final assembly, other parts, tools, and gages. It helps designers and product engineers focus on the *relationships* involved, on the “voice of the design.”

Drawings with GD&T contain three additional elements that are not found on drawings without GD&T:

1. Feature Control Frames; 2. Datums; and 3. Basic Dimensions. (See definitions in the glossary.)

Conversion to GD&T – Part 1 $\int t_r$



The following exercise will allow you to convert a drawing from the coordinate system to one that uses appropriate GD&T to do the following:

- Create explicit datums and datum sequences.
- Create logical tolerance zones.
- Create variable-sized tolerance zones, if appropriate, to provide additional tolerance.
- Create a feature control frame to communicate the requirements in the above bullets.

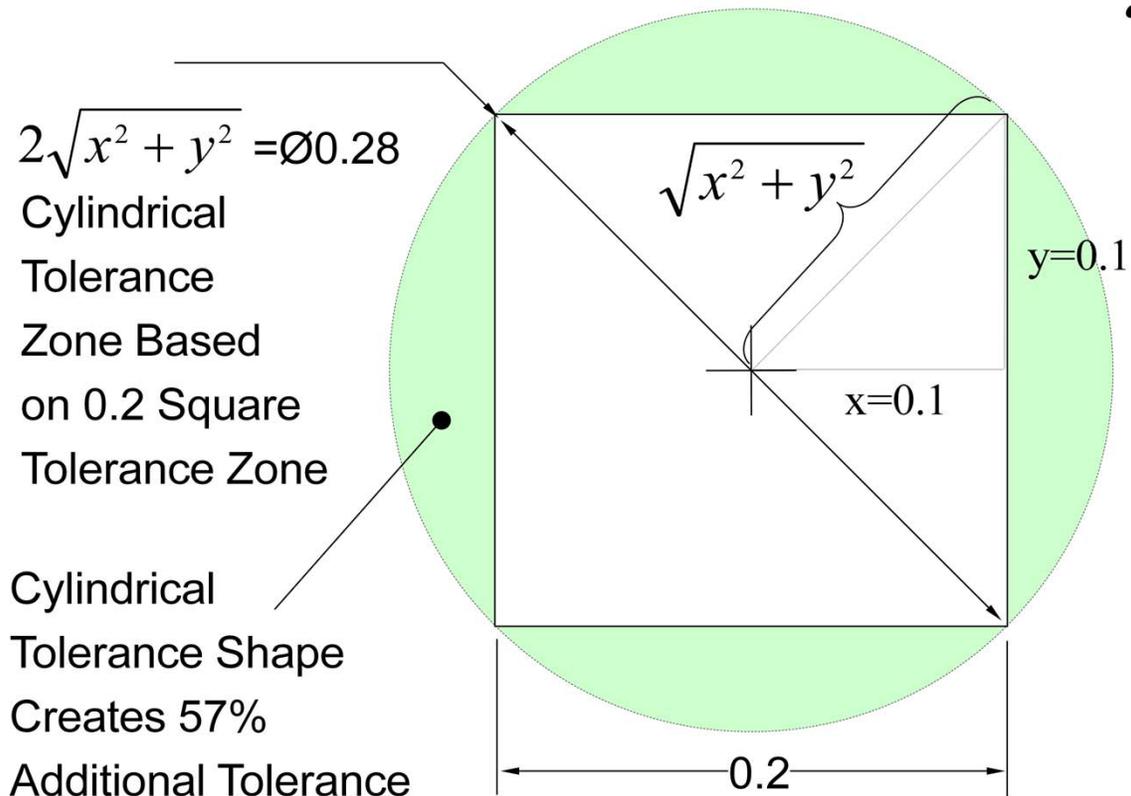
Creating Explicit Datums

The first step in this conversion process is to replace the implied datums with explicit datums. Selecting datum features requires an analysis of the assembly, performance, and manufacturing relationships, a series of skills that will be discussed in the chapters on datums. In this exercise we will simply study the drawing above within the overall context of the stapler assembly to establish explicit datum features. Remember that the best datum candidates are based on part-to-part contact in the application.

Draw in the following datum feature symbols on the drawing views above:

1. Add a datum feature symbol identifying the lower surface of the 5.3 dimension in the front view as datum A. (This surface is used as the primary mounting surface for the staples and the follower.)
2. Add a datum feature symbol identifying the center plane of the 12.85 width in the top view as datum B. (This width also interfaces with the staples and the follower.)
3. Add a datum feature symbol identifying the right surface of the 49.1 dimension in the front view as datum C. (This surface interfaces with the hammer in the final assembly.)

Conversion to GD&T - Part 2 $\int t_r$



Creating a Logical Cylindrical Tolerance Zone

The second step is to convert the 0.2 square tolerance zone value to a cylindrical tolerance value by using the following formula. This value will be added to the drawing on the next page.

$$\text{Ø} = 2\sqrt{x^2 + y^2}$$

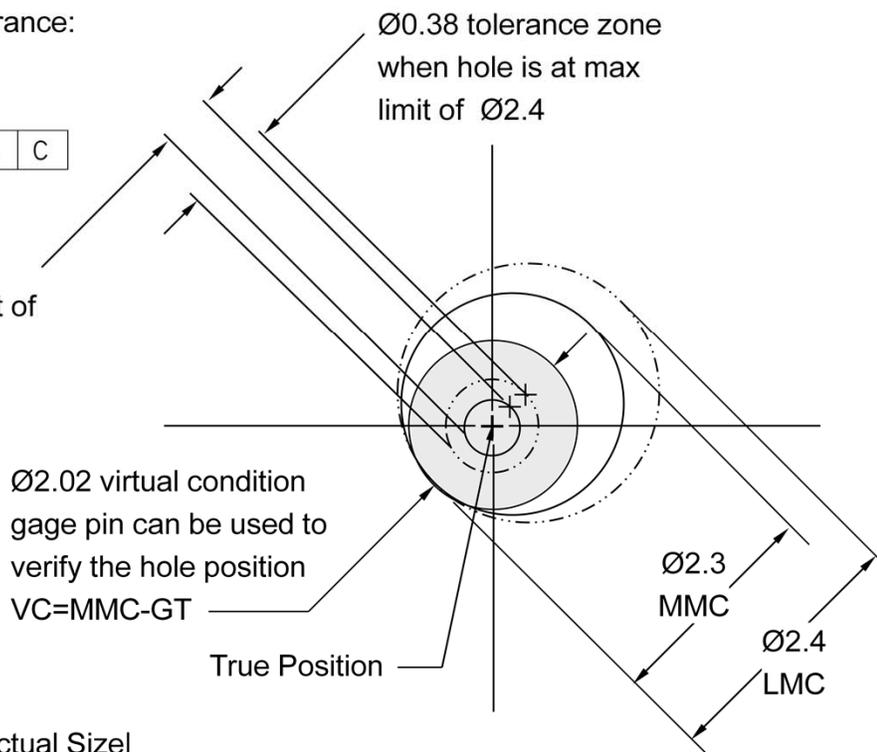
Conversion to GD&T - Part 3 *It_r*

Converted Position Tolerance:

Ø2.35±0.05



Ø0.28 tolerance zone when hole is at min limit of Ø2.3, signified by the MMC modifier (M)



Bonus = |MMC Limit – Actual Size|

Creating a Variable-Sized Tolerance Zone

The next step is to specify that this tolerance zone can increase in size proportionately as the hole size increases. The additional tolerance is referred to as *bonus tolerance* (see Bonus Tolerance).

If the design requires that the Ø2.35 hole have a worst case inner boundary of not less than Ø2.02, the smallest hole size, Ø2.3, can accommodate no more than a Ø0.28 position tolerance. The smallest hole results in a part with the maximum material; the smallest limit of size for an internal feature is therefore referred to as the “**Maximum Material Condition**”, or MMC.

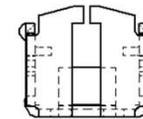
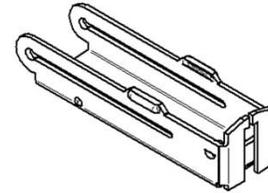
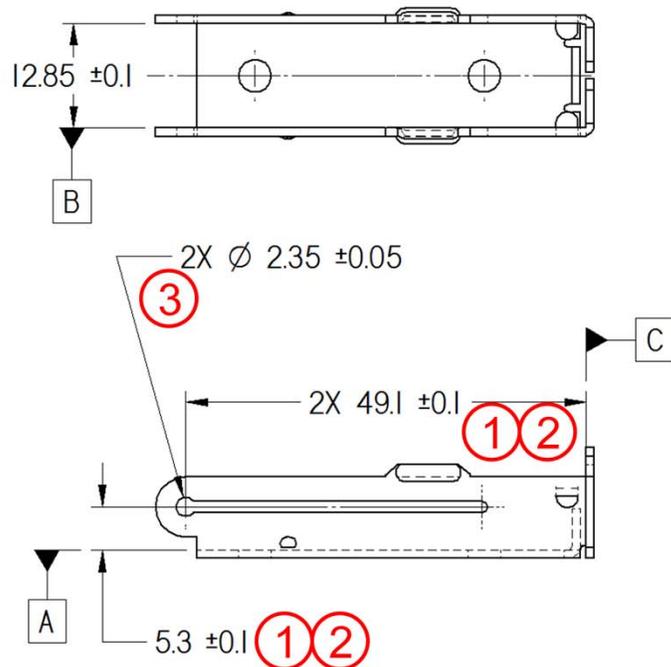
If the hole is made at the nominal size of Ø2.35, however, the position tolerance can be increased to Ø0.33 and still meet the worst case inner boundary requirement of Ø2.02.

Finally, if the hole is produced at its maximum limit of size, Ø2.4, it can have the maximum position tolerance of Ø0.38 and still meet the design requirement. The largest hole creates the part with the minimum material; the largest limit of size for an internal feature is therefore referred to as the “**Least Material Condition**”, or LMC.

The constant worst case inner boundary created by this specification is referred to as the **virtual condition**, and can be simulated by a gage pin of the same size. This phenomenon is illustrated in the graphic above and can also be seen in the table to the right.

	Actual Mating Envelope	Stated Position Tolerance	Bonus Position Tolerance	Total Position Tolerance	Virtual Condition Boundary
MMC	2.300	0.28	0.000	0.280	2.02
	2.325	0.28	0.025	0.305	2.02
	2.350	0.28	0.050	0.330	2.02
	2.375	0.28	0.075	0.355	2.02
LMC	2.400	0.28	0.100	0.380	2.02

Conversion to GD&T – Part 4 $\int t_r$



UNLESS OTHERWISE SPECIFIED
DIMENSIONING AND TOLERANCING
PER ASME Y14.5M-2009.

1 PLACE DIMS.	+/-0.3
2 PLACE DIMS.	+/-0.15
3 PLACE DIMS.	+/-0.05
ANGLES +/-	30°

Creating a Feature Control Frame

The final steps in the drawing conversion require that the coordinate tolerances are replaced with a geometric tolerance that communicates the elements of the new specification. This is accomplished with two additional steps:

- Convert the location dimensions to *basic dimensions* (see Basic Dimension).
- Create a feature control frame under the $\text{Ø}2.35\pm0.05$ hole specification.

The coordinate tolerances on the 5.3 and 49.1 location dimensions must be removed for the geometric tolerance to apply. But simply removing the local tolerances would indicate that the title block tolerances apply. To prevent this potential misinterpretation, these dimensions are boxed indicating that their tolerances are related to a particular feature. Boxed dimensions are referred to as *basic dimensions*.

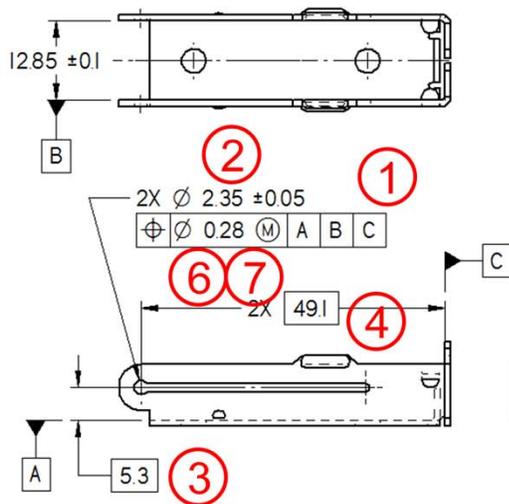
A special box can now be created to communicate the following specification:

“The axis of the $\text{Ø}2.35$ hole must be positioned within a cylindrical tolerance zone that measures $\text{Ø}0.28$ when the hole is produced at its minimum limit of size; the position tolerance increases proportionately as the actual hole size increases. The tolerance zone is located at “true position” as established by the applicable basic dimensions with respect to a datum reference frame established by primary datum A for location, secondary datum B for orientation, and tertiary datum C for location.”

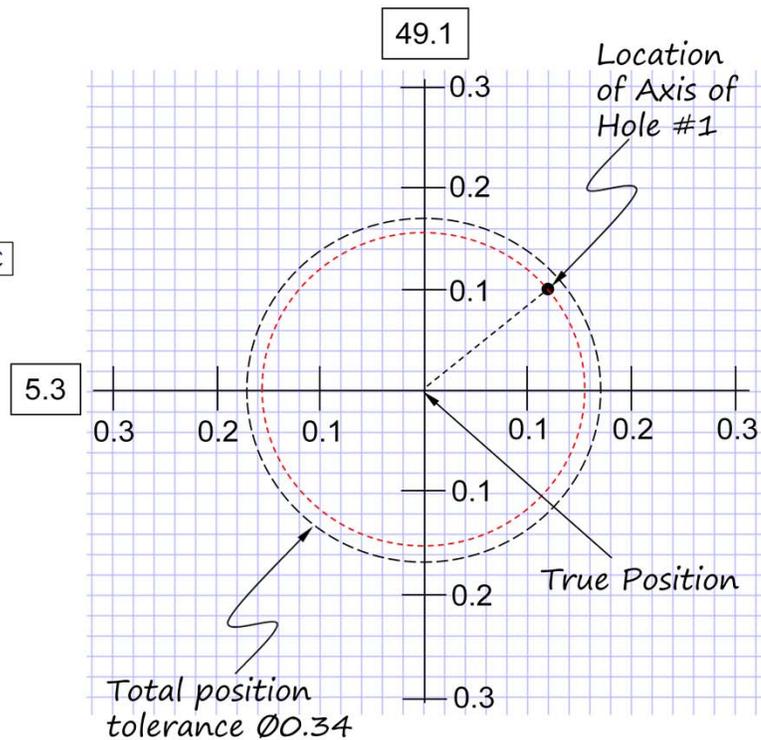
Perform the following steps to complete the conversion process:

1. Cross out the tolerances after the 5.3 and 49.1 location dimensions.
2. Box the 5.3 and 49.1 dimensions to make them basic.
3. Draw the necessary feature control frame under the $\text{Ø}2.35$ hole specification. Refer to the previous page for an example of the required feature control frame.

Inspection of Position w/ MMC $\int t_r$



Measurements	Deviation
Hole #1 $\text{Ø}2.36$	0.06
From A 5.40	0.10
From C 48.98	0.12
Radial deviation	0.1562
Diameter deviation	0.3124

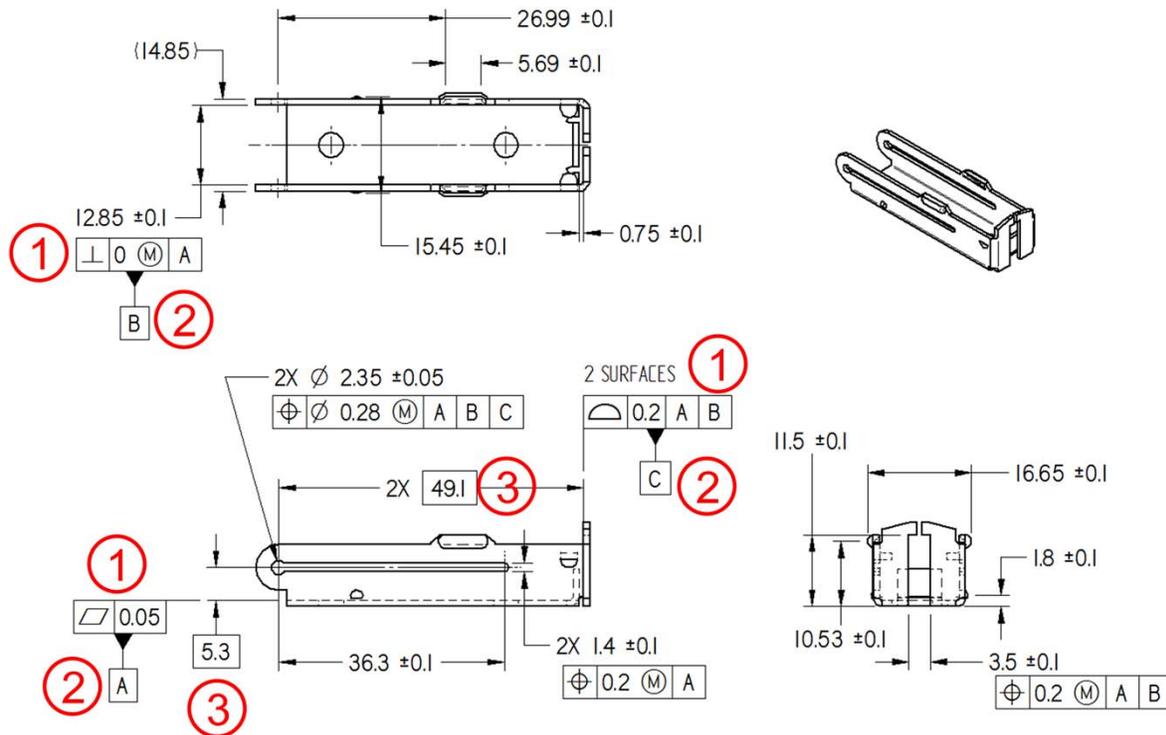


Inspection of Position with the MMC Modifier

The following steps would be required for the inspection of the position of the 2.35 holes:

1. Simulate the A, B, C datum reference frame
2. Measure the actual size of the 2.35 holes. In this example the first hole measures 2.36.
3. Measure the actual distance from datum A. In this example the distance from datum A is 5.40, for a deviation from true position in the Y axis of 0.10.
4. Measure the actual distance from datum C. In this example the distance from datum C is 48.98, for a deviation from true position in the X axis of 0.12.
5. Calculate the radial deviation (0.1562) and then express that deviation as a diameter (0.3124)
6. Compare the diameter of deviation with the stated tolerance of 0.28. If the diameter of deviation is equal to or less than 0.28, the hole is within the position tolerance. If not, proceed to step 7.
7. Calculate the actual bonus tolerance based on the following formula: $\text{Bonus} = |\text{MMC} - \text{Actual Size}|$. In this case the actual bonus is 0.06, for a total tolerance of 0.34. $0.34 > 0.3124$ therefore the hole is within the total position tolerance, including the bonus.

Major Elements of GD&T



The 3 Major Elements of GD&T

The drawing above shows the results of the conversion process. The drawing now contains the three elements that set drawings with GD&T apart from drawings with coordinate dimensioning and tolerancing alone:

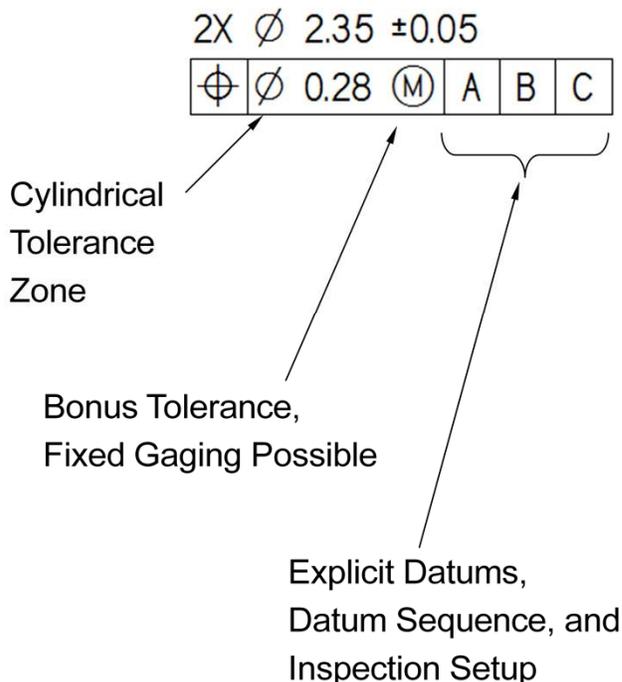
1. Feature Control Frames
2. Datums
3. Basic Dimensions

Circle the examples of each in the drawing above. (Other feature control frames and basic dimensions have been added to make the drawing more complete.)

Advantages of GD&T



- tolerance zones providing maximum tolerance.
- tolerance zones to provide bonus tolerance.
- datum references.



Word list: logical, flexible-size, explicit.

Advantages of GD&T

To summarize, GD&T is able to address the major shortcomings of the coordinate system by providing:

1. Logical tolerance zones.
2. Variable-sized tolerance zones, if appropriate, to provide additional tolerance.
3. Explicit datums references and datum sequences.

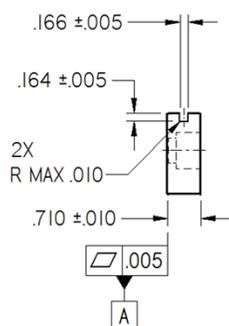
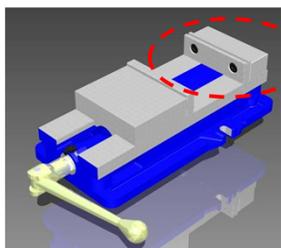
The disadvantages of using GD&T are minor in comparison to the advantages. It improves communication by providing a common, standardized language for all stakeholders, and by reducing ambiguous datums and gaging setups. It leads to better designs by allowing designers to say exactly what they mean, and by encouraging designs based on how components and assemblies function in the context of the overall design. Finally, it provides increased tolerances by creating logical tolerance zones and extra (bonus) tolerance.

GD&T, when applied and interpreted correctly, technically should not increase part costs, but rather decreases part costs and typically increases tolerances. When GD&T is applied incorrectly or is not understood by the drawing users, it can have the effect of increasing part cost. Learning to correctly interpret and apply GD&T requires investment in mastering the symbology, syntax, logic, and appropriate uses of GD&T. As with any language, fluency with GD&T is only obtained with study, practice, and real-world application. Understanding GD&T callouts is just the first step toward fluency. Those who must apply GD&T to the print in the first place, such as design and engineering personnel, must learn the concepts, applications, and variety of options to a relatively deeper level. The “disadvantage” of GD&T, therefore, is the need for ongoing analysis, study, training, application, and interpretation of the concepts; but that should be the case for all engineering and design disciplines and skills.

Appropriate Mix of Coordinate and Geometric Tolerances

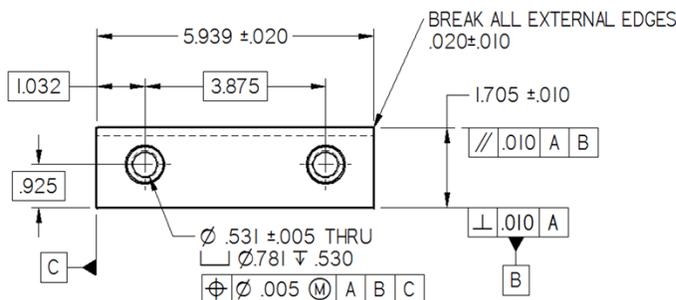
Coordinate Dimensioning and Tolerancing

-
-
-
-
-
-
-
-
-
-



Geometric Dimensioning and Tolerancing (GD&T)

-
-
-
-
-
-
-
-
-
-



While GD&T offers many benefits and advantages over coordinate and/or polar dimensioning and tolerancing alone, it was never intended to, nor should it be used to replace the coordinate system entirely.

Coordinate tolerancing is still the best option for defining many features, relationships, and specifications including:

- Size (diameters, widths)
- Location of select surfaces
- Chamfer
- Radius
- Draft
- Depth
- Length

Geometric Dimensioning and Tolerancing can, in theory, be used to define all the part features and specifications above; but GD&T should usually be used to complement coordinate dimensioning and tolerancing as appropriate in the following areas, and should typically not be used to define features of size:

- Location (especially of the axis or center plane of a feature of size)
- Orientation (especially for critical-to-quality (CTQ) feature perpendicularity or other orientation relationships)
- Form (especially on CTQs and datum features)
- Profile (especially on CTQs and datum features)
- Runout

GD&T Application Guidelines *It_r*

- **Necessary**
- **Technically Correct**
- **Appropriate**
- **Beneficial**
- **Understood**

Integrated Training Resources, Inc.

Necessary – Don't use it unless it addresses real quality or engineering issues, or provides a quantifiable benefit. It should typically be used as a supplemental strategy.

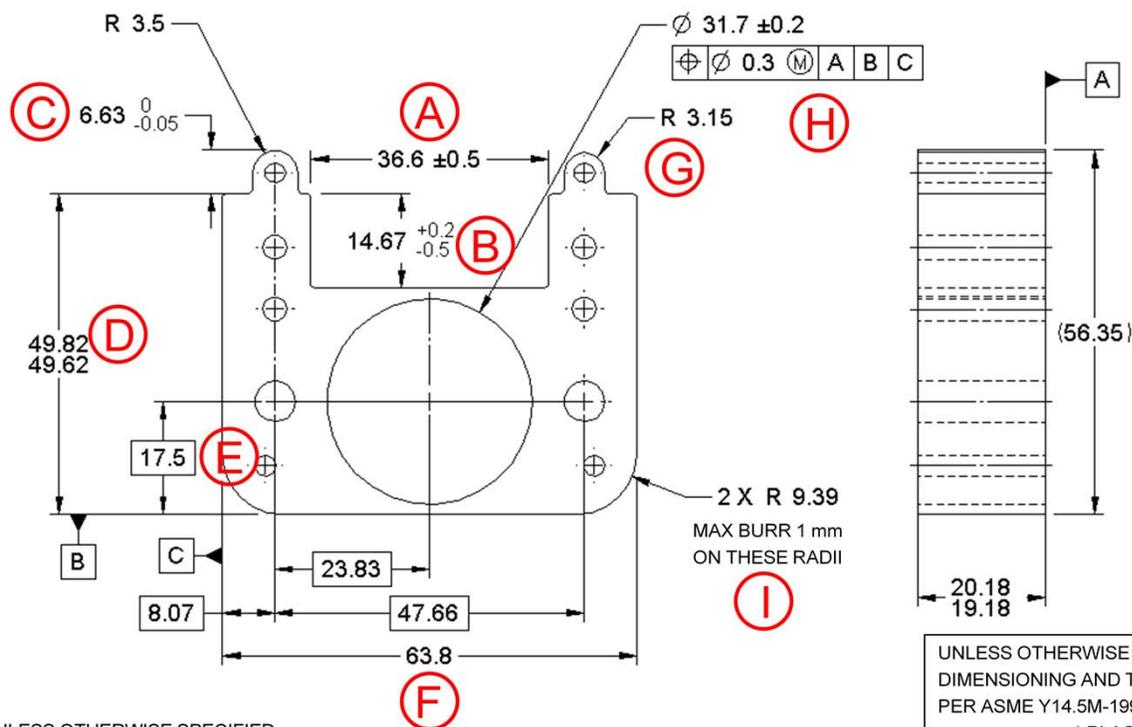
Technically Correct – It should be applied in a "legal" fashion, in accordance with applicable international standards (e.g. ASME or ISO) whenever possible, not only in terms of meaning, but also graphic representation and appearance.

Appropriate – GD&T is not an all or nothing proposition; it should be applied as appropriate, focusing on critical product characteristics. It should typically not be used to define size, length, depth, chamfer, and radius specification where coordinate tolerancing will provide adequate definition.

Beneficial – GD&T should never have the effect of penalizing manufacturing and inspection as compared with coordinate dimensioning and tolerancing practices. It should typically have the effect of achieving the same or greater tolerances that would otherwise be available without GD&T.

Understood – When properly applied to drawings, GD&T should be understood by all stakeholders, including design/product engineering, manufacturing, inspection/quality, and any other key stakeholders. Drawing reviews should be conducted early and often prior to release to ensure dimension and tolerancing meets the above criteria.

Exercise 1-1



UNLESS OTHERWISE SPECIFIED
 RADII MAX 0.5
 BURRS MAX 0.5
 Refer to 3545276 for plating specifications.

UNLESS OTHERWISE SPECIFIED
 DIMENSIONING AND TOLERANCING
 PER ASME Y14.5M-1994.

J

1 PLACE DIMS. +/-0.3
 2 PLACE DIMS. +/-0.15
 3 PLACE DIMS. +/-0.05
 ANGLES +/- 30'

Choose the best terms to complete the following statements based on the above drawing.

1. Dimension A is an example of an _____ tolerance.
2. Dimension B is an example of a _____ tolerance.
3. Dimension C is an example of a _____.
4. Dimension D is an example of a _____.
5. Dimension E is an example of a _____.
6. Dimension H is an example of a _____.
7. J is an example of _____ tolerances.

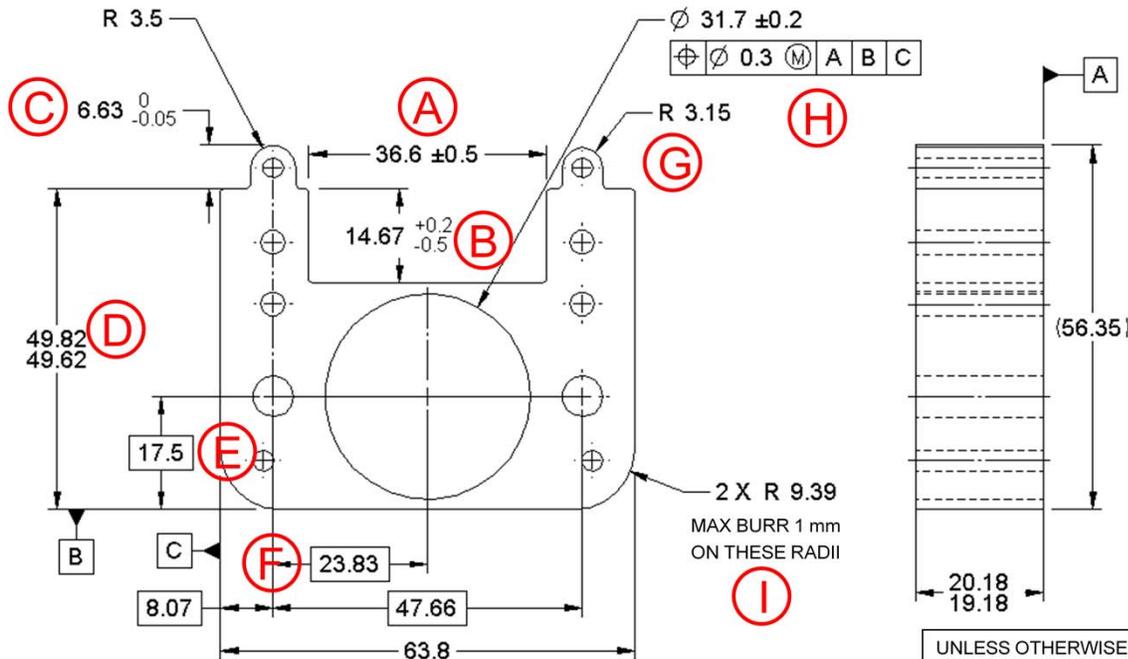
Calculate the total tolerance values for the following dimensions.

8. The total tolerance on dimension A is _____.
9. The total tolerance on dimension B is _____.
10. The total tolerance on dimension C is _____.
11. The total tolerance on dimension D is _____.
12. The total tolerance on dimension E is _____.
13. The total tolerance on dimension F is _____.
14. The total tolerance on dimension G is _____.
15. The *minimum* tolerance from feature control frame H is _____.
16. The *maximum* tolerance from feature control frame H is _____.

Terms for Statements 1-7

- Limit dimension
- Unequal bilateral
- Basic dimension
- Equal bilateral
- General (Title Block)
- Unilateral tolerance
- Feature Control Frame (Geometric Tolerance)

Exercise 1-2



UNLESS OTHERWISE SPECIFIED
 RADII MAX 0.5
 BURRS MAX 0.5
 Refer to 3545276 for plating specifications.

UNLESS OTHERWISE SPECIFIED
 DIMENSIONING AND TOLERANCING
 PER ASME Y14.5M-1994.

J
 1 PLACE DIMS. +/-0.3
 2 PLACE DIMS. +/-0.15
 3 PLACE DIMS. +/-0.05
 ANGLES +/- 30'

Copyright 1998-2015 ITR

GD&T 1-38

Integrated Training Resources, Inc.

Circle IN or OUT to complete each statement.

1. A measurement of 36.100 for dimension A is IN / OUT of specification.
2. A measurement of 14.88 for dimension B is IN / OUT of specification.
3. A measurement of 6.579 for dimension C is IN / OUT of specification.
4. A measurement of 49.820 for dimension D is IN / OUT of specification.
5. For position tolerance H, with measurements of 17.70 for dimension E, 31.90 for distance from datum C, and 31.77 as the measured size of the hole, the position of the hole is IN / OUT of specification.
6. A measurement of 3.301 for dimension G is IN / OUT of specification.
7. A measurement of 0.2 for dimension I is IN / OUT of specification.
8. Dimension are used to describe the _____, _____, _____, and _____ of a part or part features.
9. The principle dimensioning and tolerancing standard used in North America is _____.
10. The _____ series of dimensioning and tolerancing standards is used outside North America.
11. For millimeter expressions less than one millimeter, the decimal is preceded by a _____.
12. For inch expressions less than one inch, no _____ precedes the decimal point and the dimension and tolerance expressions should both have the same number of _____.
13. For whole number metric dimensions no _____ is used.
14. Three benefits to using GD&T include _____

15. A drawing with GD&T will likely have the following three elements: _____

Exercise 1-3

Define the following terms (see the glossary for definitions):

1. ISO 1101 – _____

2. ASME Y14.5M – _____

3. Basic Dimension (Theoretically Exact Dimension) – _____

4. Bonus Tolerance – _____

5. Coordinate Dimension – _____

6. Datum – _____

7. Dimension – _____

8. Engineering Drawing – _____

9. Feature Control Frame (Tolerance Frame) – _____

10. Geometric Tolerance – _____

11. Maximum Material Condition (MMC) – _____

12. Tolerance – _____

13. True Position – _____

14. Virtual Condition – _____

Exercise 1-4

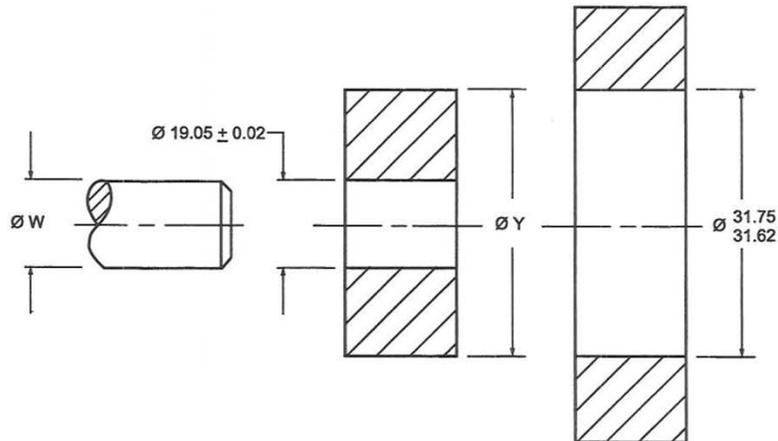
Complete each of the fundamental rules from ASME Y14.5-2009. See pages 1-19 and 1-20.

- (a) Each dimension shall have a _____, 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 ASME Y14.1 and ASME Y14.1M.
- (b) Dimensioning and tolerancing shall be complete so there is full understanding of the characteristics of each _____. Values may be expressed in an engineering drawing or in a CAD product definition data set. See ASME Y14.41. Neither scaling (measuring 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, provided the necessary control dimensions are specified.
- (c) Each _____ 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 _____ and mating relationship of a part and shall not be subject to more than _____ interpretation.
- (e) The drawing should define a part without specifying _____ 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) Nonmandatory _____ dimensions shall be identified by an appropriate note, such as "NONMANDATORY (MFG DATA)." Examples of nonmandatory data are processing dimensions that provide for finish allowance, shrink allowance, and other requirements, provided the final dimensions are given on the drawing.
- (g) Dimensions should be arranged to provide required information for optimum _____. 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 _____ 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 2D orthographic drawing at _____ angles and no angle is specified. See para. 2.1.1.3.
- (j) A _____ basic angle applies where center lines of features in a pattern or surfaces shown at right angles on a 2D orthographic drawing are located or defined by basic dimensions and no angle is specified. See para. 2.1.1.4.
- (k) A _____ basic dimension applies where axes, center planes, or surfaces are shown coincident on a drawing, and geometric tolerances establish the relationship among the features. See para. 2.1.1.4.
- (l) Unless otherwise specified, all dimensions and tolerances are applicable at _____ (68°F) in accordance with ANSI/ASME B89.6.2. Compensation may be made for measurements made at other temperatures.
- (m) Unless otherwise specified, all dimensions and tolerances apply in a _____ condition. For exceptions to this rule see paras. 4.20 and 5.5.
- (n) Unless otherwise specified, all tolerances apply for full depth, _____, and width of the feature.
- (o) Dimensions and tolerances apply only at the drawing level where they are _____. A dimension specified for a given feature on one level of drawing (e.g., a detail drawing) is not mandatory for that feature at any other level (e.g., an assembly drawing).
- (p) Where a _____ system is shown on the drawing, it shall be right-handed unless otherwise specified. Each axis shall be labeled and the positive direction shall be shown.

Exercise 1-5

It_r

1. Dimension shaft (W) to have:
 - a) a minimum clearance of 0.03 with the mating bushing, and;
 - b) a unilateral tolerance of 0.08
2. Dimension bushing (Y) to have:
 - a) a minimum interference of zero with the large hole, and;
 - b) an equal bilateral tolerance with a total tolerance of 0.04



Notes

Notes

Notes