Geometric Dimensioning and Tolerancing for Mechanical Design

A SELF-TEACHING GUIDE TO THE ASME Y14.5M-1994 STANDARD

- ✓ Interprets and applies the ASME Y14.5M-1994 Standard
- ✓ Covers datums and form, orientation, and profile controls
- ✓ Detailed explanation of the many applications of the position control
- ✓ Practice questions at the end of each chapter

Gene R. Cogorno

Geometric Dimensioning and Tolerancing

Course Objectives

1. Introduction
2. Dimensioning and Tolerancing Fundamentals
3. Symbols, Terms, and Rules
4. Datums
5. Form controls
   A. Flatness
   B. Straightness
   C. Circularity
   D. Cylindricity
   E. Free state variation
6. Orientation
   A. Parallelism
   B. Perpendicularity
   C. Angularity

7. Position, General
   A. Specifying the position tolerance
   B. Regardless of feature size
   C. Maximum material condition
   D. Shift tolerance
   E. Boundary conditions
   F. “0” positional tolerancing at MMC
8. Position, Location
   A. Fasteners
   B. Projected tolerance zones
   C. Multiple patterns of features
   D. Composite positional tolerancing
   E. Two single-segment feature control frames
   F. Nonparallel holes
   G. Counterbored holes
   H. Noncircular features at MMC
   I. Symmetrical features at MMC
9. Position, Coaxiality
10. Concentricity & Symmetry
11. Runout
12. Profile

Introduction to Geometric Dimensioning and Tolerancing

Chapter Objectives

After completing this chapter, you will be able to:

- **Explain** the definition of GD&T
- **Explain** when to use GD&T
- **Identify** three advantages of GD&T over coordinate tolerancing
  1. The cylindrical tolerance zone
  2. The maximum material condition
  3. Datums specified in order of precedence

What is GD&T?

Geometric Dimensioning and Tolerancing:

- **Is a symbolic language**
- **Is a design tool**
- **Communicates design intent**

When Should GD&T be Used?

Designers should tolerance parts with GD&T when:

- Drawing delineation and interpretation need to be the same
- Features are critical to function or interchangeability
- It is important to stop scrapping perfectly good parts
- It is important to reduce drawing changes
- Automated equipment is used
- Functional gaging is required
- It is important to increase productivity
- Companies want across the board savings

In short, used GD&T to **locate all SIZE FEATURES**.

The Coordinate Tolerancing System

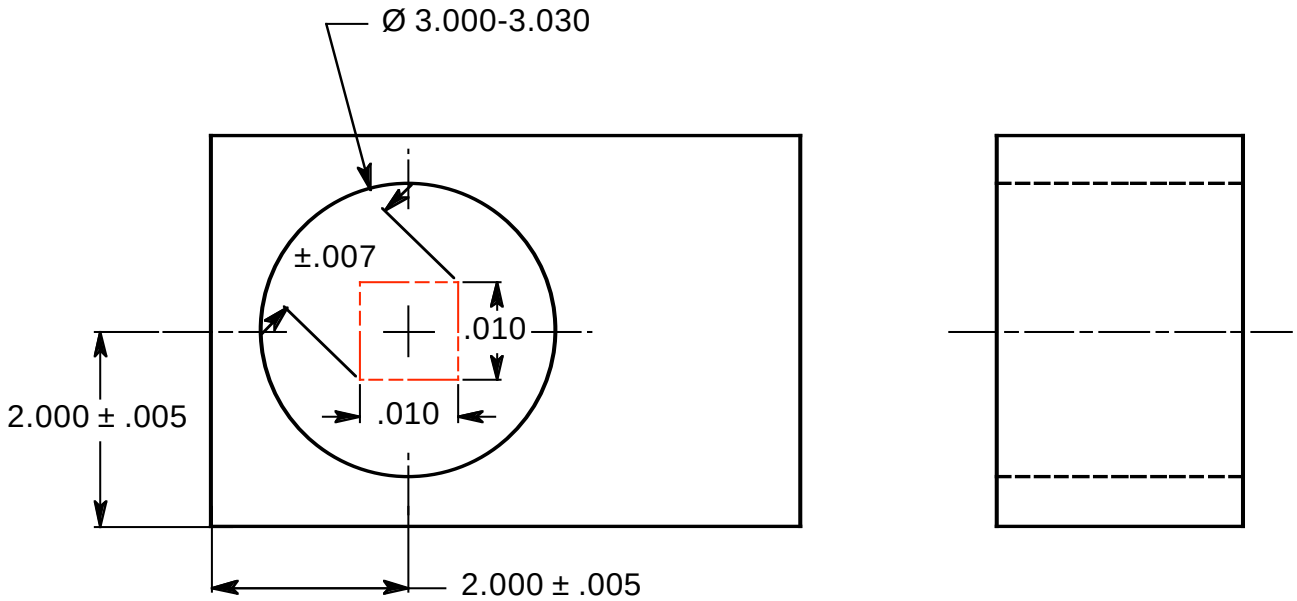


Figure 1-1 The traditional plus or minus tolerancing system

- A plus or minus tolerance generates rectangular tolerance zones.
- Size features can only be specified at RFS.
- Datums are usually not specified.

Advantages of GD&T over the coordinate tolerancing system:

- The cylindrical tolerance zone
- The maximum material condition
- Datums specified in order of precedence

The Cylindrical Tolerance Zone

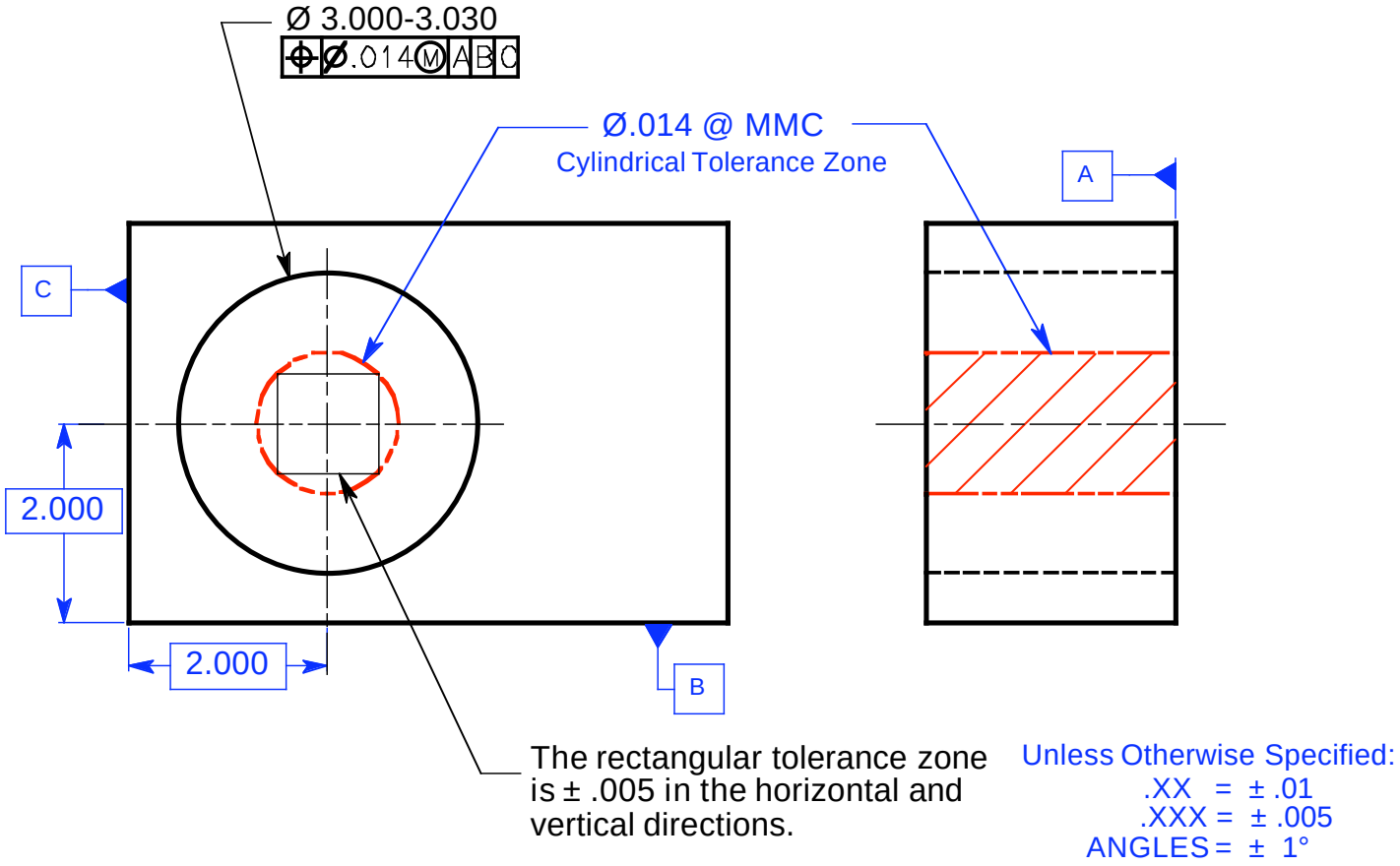
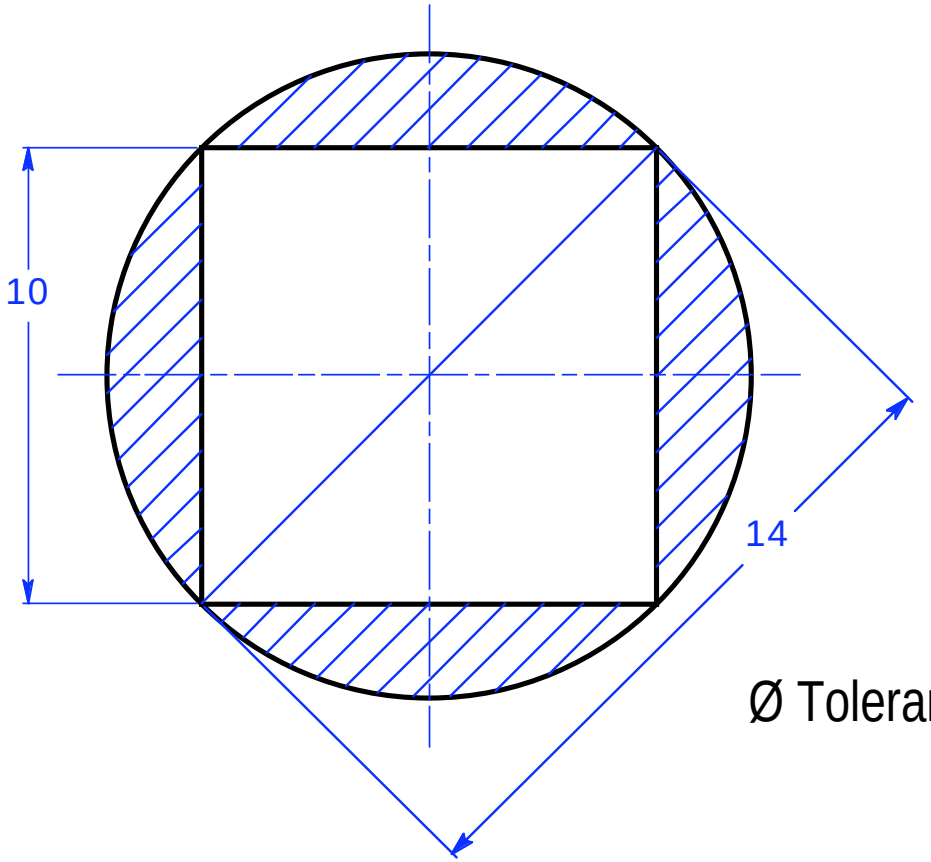


Figure 1-2 The cylindrical tolerance zone compared to the rectangular tolerance zone



$$\text{Ø Tolerance Zone} = \sqrt{.010^2 + .010^2} \approx .014$$

Figure 1-3 A cylindrical tolerance zone provides a uniform distance from the axis of the hole to the edge of the tolerance zone. It contains 57% more tolerance.

The Maximum Material Condition

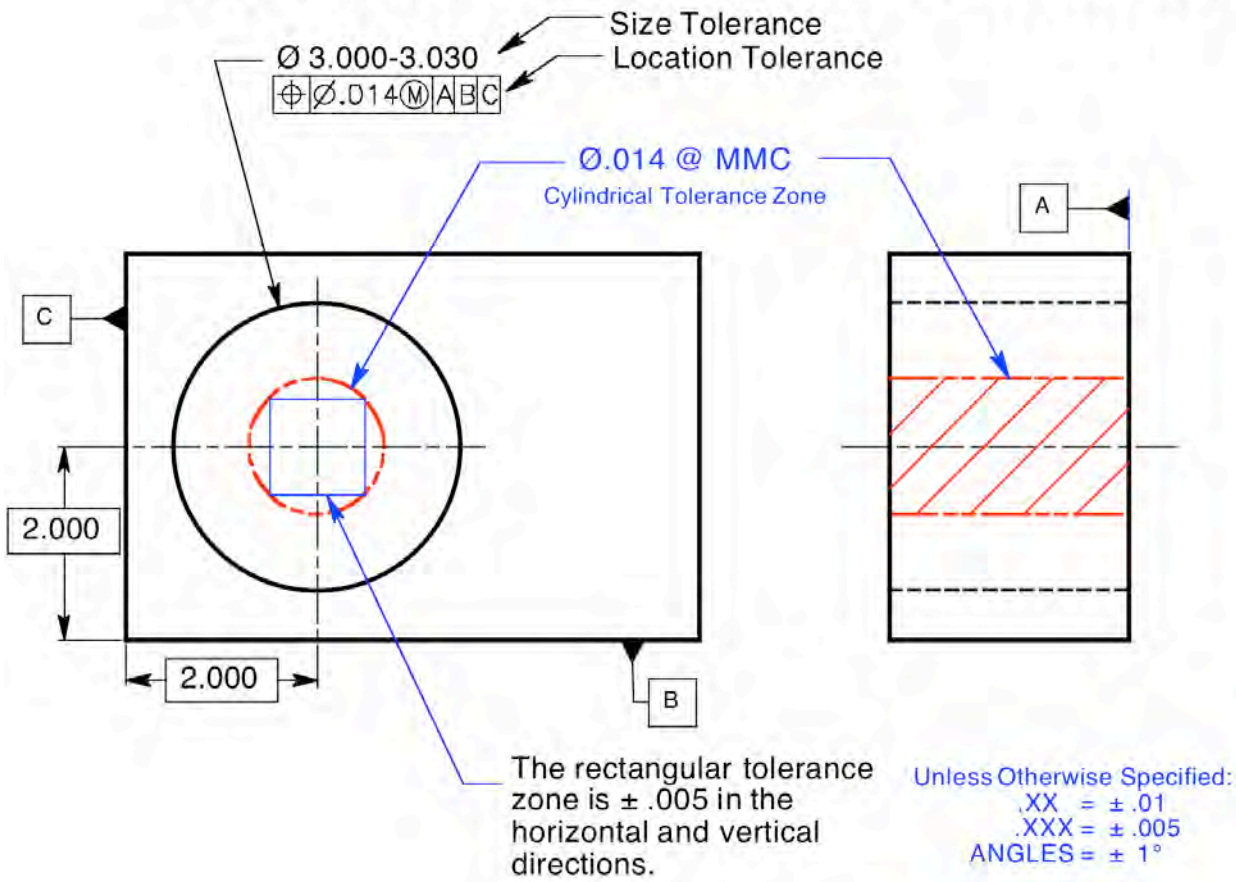


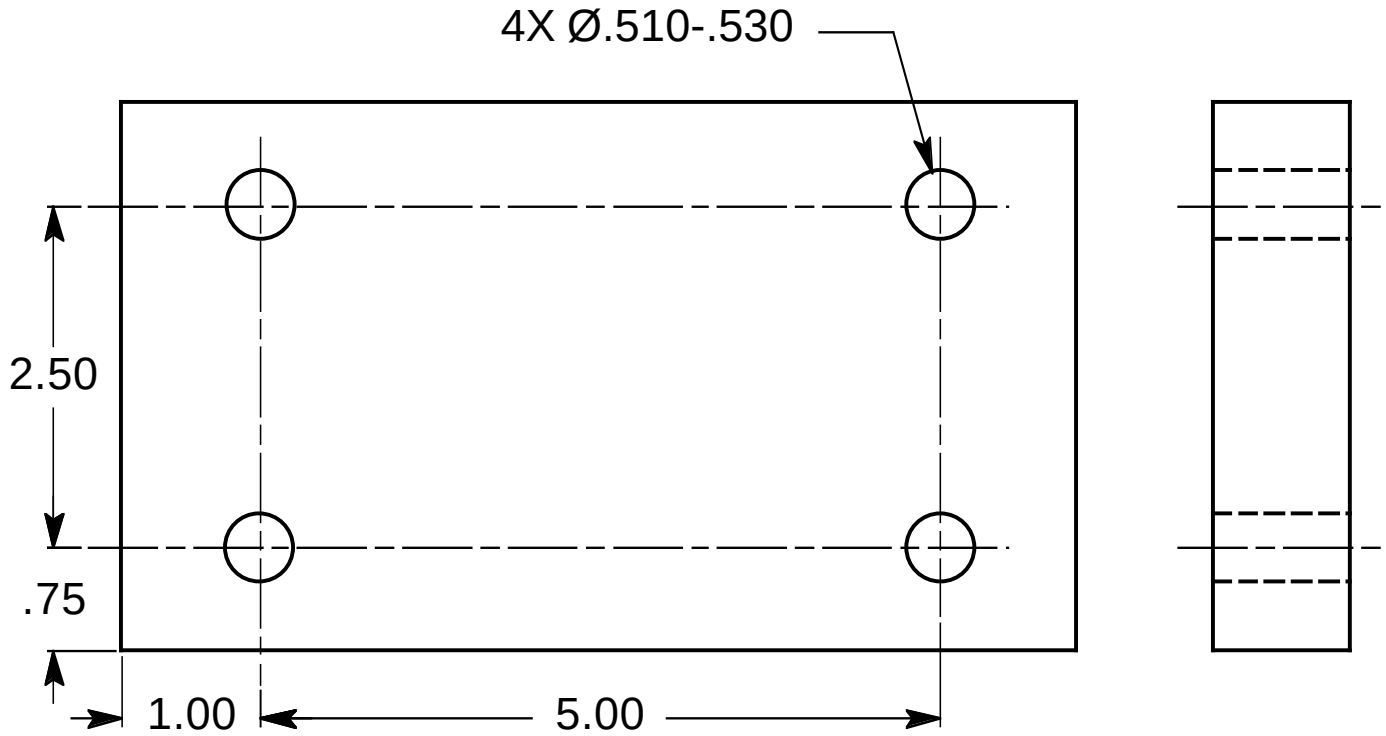
Figure 1-4A The size, size tolerance, and feature control frame for the hole location

As the hole diameter increases, the location tolerance increases.

In Figure 1-4A, if the **actual hole diameter is 3.020**, the **total positional tolerance is a diameter of .034.**

Actual feature size	3.020	Bonus tolerance	.020
Minus the MMC	–3.000	**Plus** the geometric tolerance	+.014
Bonus tolerance	**.020**	**Equals the Total tolerance**	**.034**

Datums Specified in Order of Precedence



Unless Otherwise Specified:
.XX: = ± .01
ANGLES: = ± 1°

Figure 1-5 No datums are specified on this drawing

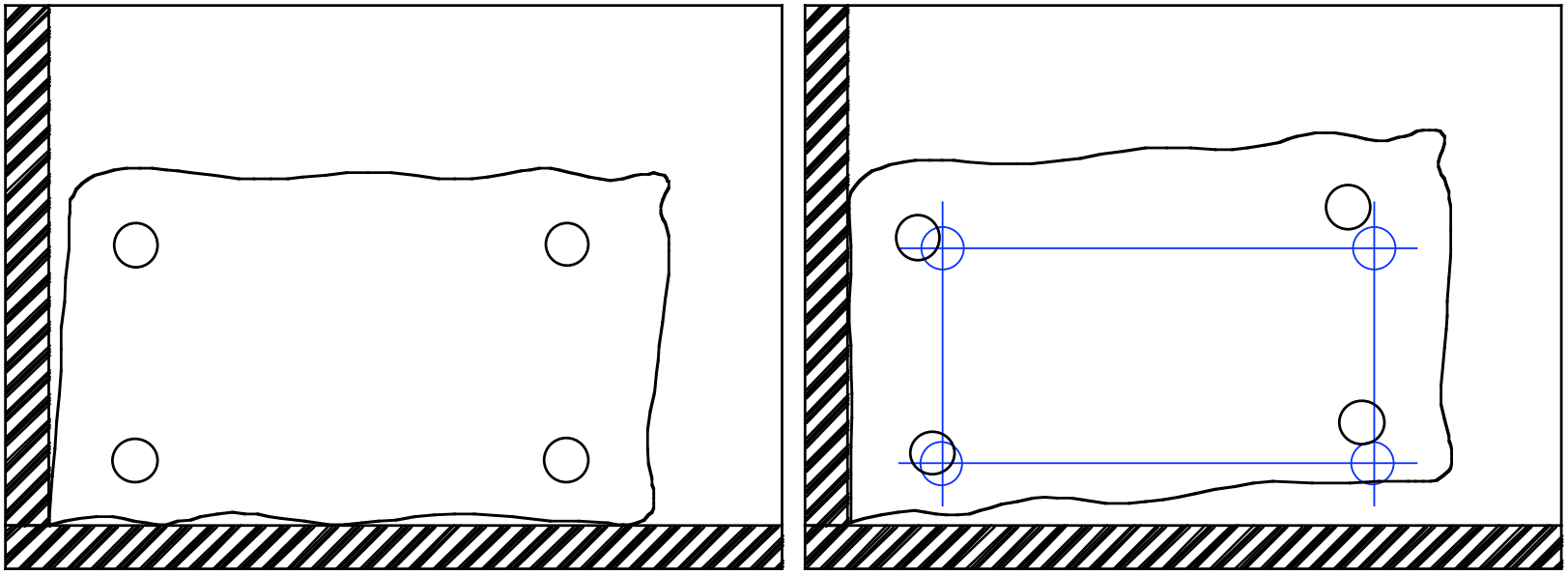


Figure 1-6 Two possible datum interpretation of the part in Figure 1-5

Summary

- GD&T is a symbolic language used to specify the size, shape, form, orientation, and location of features on a part.
- GD&T was created to insure the proper assembly of mating parts, to improve quality, and to reduce cost.
- GD&T is a design tool that communicates design intent.
- This text is based on the standard, *Dimensioning and Tolerancing* ASME Y14.5M–1994.

- The cylindrical tolerance zone defines a uniform cylindrical tolerance zone boundary.
- The MMC symbol is a modifier that allows a bonus tolerance.
- All datums must be specified in order of precedence.