

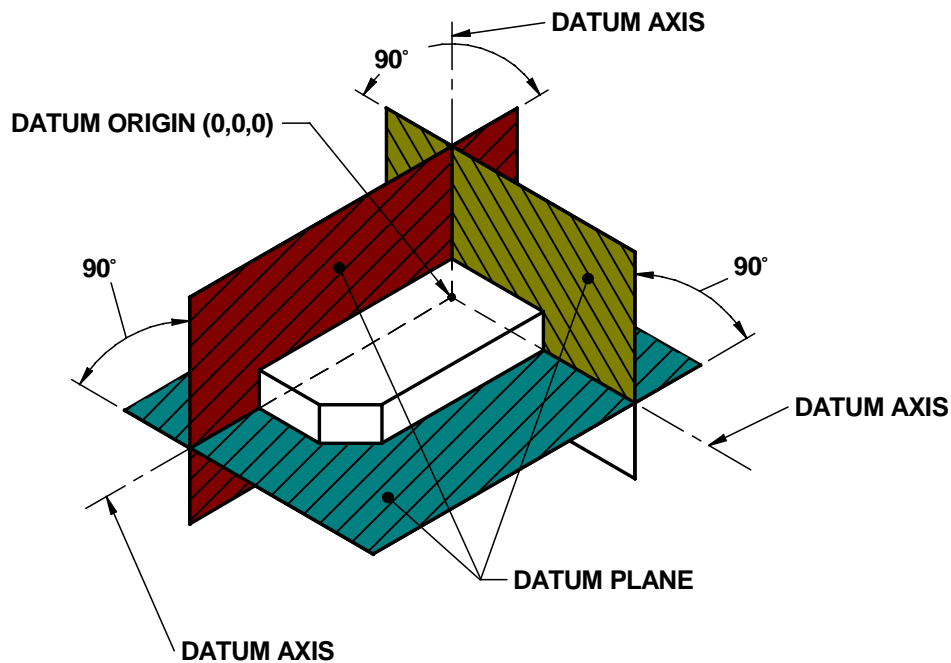
Geometric Boundaries

*Interpretation and Application
of Geometric Dimensioning and Tolerancing
(Using the Customary Inch System)
Based on ASME Y14.5M-1994*

Written and Illustrated by
Kelly L. Bramble

Published by:
Engineers Edge
510 N. Crosslane Road
Monroe, Georgia 30656
www.engineersedge.com

Copyright © All Rights Reserved



Preface

This book is written for those individuals within the design, drafting, engineering and manufacturing fields that desire a practical guide for the interpretation and application of geometric dimensioning and tolerancing.

I have deliberately directed my efforts for technical professionals applying geometric dimensioning and tolerancing and attempted to comprehensively cover the concepts and applications that are and will be the most relevant within industry today and the future. The choice of examples are those which represent typical applications and may be combined as applicable to create products.

Much of the text material has been organized so that the topics appear and build the necessary knowledge required to proceed to the next subject matter.

The book is dedicated to my children, Nathan and Heather.

Kelly L. Bramble

Copyright 2002, 2003, 2004, 2005, 2006, 2007, 2008 All right reserved. No part of this book may be reproduced or transmitted in any form or by any means, electronic or mechanical, including photocopying, recording, or any information storage and retrieval system, without permission in writing from the publisher.

Revision L

Acknowledgments

The following documents have been used as reference material (cited and not cited).

ASME Y14.5.M-1994, Dimensioning and Tolerancing.
ANSI Y14.5M-1982, Dimensioning and Tolerancing.
ANSI Y14.5M-1973, Dimensioning and Tolerancing
ANSI Y14.5M-1966, Dimensioning and Tolerancing
International Standards Institute, ISO/R1101-1969, & Associated Documents
ANSI B4.2-1978, Preferred Metric Limits and Fits
ANSI B5.10-1981, Machine tapers – Self Holding and Steep Taper Series
ANSI/ASME B46.1-1985, Surface Texture (Surface Roughness, Waviness, and Lay)
ANSI B89.3.1-1972, Measurement of Out-of-Roundness
ANSI B92.1-1970, Involute Splines and Inspection, Inch Version
ANSI B92.2M-1980, Metric Module, Involute Splines
ANSI/ASME B94.6-1984, Knurling
ANSI B94.11M-1979, Twist Drills
ANSI Y14.1-1980, Drawing Sheet Size and Format
ASME Y14.2M-1992, Line Conventions and Lettering
ASME Y14.5.1M-1994, Mathematical Definition of Dimensioning and Tolerancing Principles.
ANSI Y14.6.1-1978, Screw Thread Representation
ANSI Y14.6.2-1981, Screw Thread Representation (Metric Supplement)
ANSI Y14.7.1-1971, Gear Drawing Standards – Part 1: For Spur, Helical, Double Helical, and Rack
ANSI Y14.7.2-1978, Gear and Spline Drawing Standard – Part 2: Bevel and Hypoid Gears
ASME Y14.8M-1989, Castings and Forgings
ANSI Y14.36-1978, Surface Texture Symbols
ANSI/IEEE 268-1992, Metric Practice
ANSI/ASME B1.2-1983, Gages and Gaging for Unified Inch Screw Threads
ANSI B4.4M-1981 (R1987), Inspection of Workpieces
ASME Y1.1-1989, Abbreviations
ASME Y14.3M-1994, Multiview and Sectional View Drawings
Engineers Edge 2000 - 2007, Solutions by Design, Kelly Bramble,
Design for Manufacturing 2006 - 2007, Kelly Bramble

Table of Contents

- 1.2 Preface
- 1.3 Acknowledgments
- 1.4 Table of Contents
- 1.9 Introduction
- 1.10 How the Geometric Dimensioning and Tolerancing System Works
- 1.11 Tolerances, Features and Characteristics
- 1.12 Feature Control Frame, Rule #2 & 2a
- 1.13 Position and Limit Tolerance General Overview and Contrast
- 1.14 Common Symbols ASME Y14.5M - 1994
- 1.15 Dimensioning and Tolerancing General Rules
- 1.17 Workshop
- 2.1 **Limit**
- 2.2 General – Dimensioning System Limit Tolerancing
- 2.3 Limit tolerancing, Square - Round Tolerance Zone Contrast and Advantage
- 2.4 Implied 90 Degree Angle, Expressing Limit Tolerancing
- 2.5 Dimension and Tolerance Expression – Millimeter Tolerances and Dimensions
- 2.7 Inch Tolerances and Dimensions
- 2.9 Angular Dimensioning and Tolerancing
- 2.10 Slotted holes, Arcs, Countersink hole
- 2.11 Counter bored Holes, Countersink on curved surface
- 2.12 Internal External Chamfers, Keyseats
- 2.13 Rectangular Coordinate Dimensioning
- 2.14 Rectangular Coordinate Dimensions in Tabular Form, Polar Dimensioning
- 2.15 Repetitive Features
- 2.16 Flat Taper, Conical Taper
- 2.17 Statistical Tolerancing
- 2.18 Dimension Origin, Screw Threads Tolerance Application
- 2.19 Angular Surface Defined by Limit and Angle Dimension
- 2.20 Gears and Splines, Controlled Radius, Radius Tolerance
- 2.21 Features With and Without Size
- 2.22 Material Condition, MMC, LMC
- 2.23 Rule #1, Envelope Principle
- 2.24 Rule #1 Limitations, Exceptions
- 2.26 Workshop
- 3.1 **Datums**
- 3.2 Datum Reference Frame, DRF - General
- 3.3 Immobilization of component and measurement
- 3.4 Datum symbols and identification Datum identification features without size
- 3.5 Datum identification features with size
- 3.6 Datum associated with feature control frame
- 3.7 Datum feature, simulated datum, and theoretical datum plane
- 3.8 Setup and inspection of datums, datum and dimensional measurement equipment
- 3.10 Sequence of datum features
- 3.11 Sequence of datum features relates part to datum reference frame
- 3.12 Cylindrical datum feature
- 3.13 Parts with angular orientation

Table of Contents

3.14	Development of datum reference frame	5.4	Flatness applied on a unit basis
3.15	Orientation of two datum planes through a hole	5.5	Flatness applied on unit basis with overall control
3.16	Development of datum reference frame	5.6	Straightness
3.17	Partial datum surface as datum features	5.7	Straightness Per Unit Basis
3.18	Multiple datum features, single datum	5.8	Straightness applied in two directions
3.19	Inclined datum features	5.9	Straightness of a surface (Cylindrical)
3.20	View oriented datum reference frame	5.10	Straightness of a feature of size @ RFS
3.21	Workshop	5.11	Straightness of a feature of size @ MMC
4.1	Datum Targets	5.12	Straightness per unit length With specified total straightness
4.2	General	5.13	Cylindricity
4.3	Datum target point symbol, application	5.14	Circularity (Roundness)
4.4	Datum target area	5.15	Workshop
4.5	Datum target line	6.1	Orientation
4.6	Dimensioning datum targets	6.2	General
4.7	Primary datum plane established by three datum target areas	6.3	Perpendicularity - Surface
4.8	Primary datum plane established by two datum target points and one datum target line.	6.4	Perpendicularity – Center plane
4.9	Step datum feature	6.5	Perpendicularity at MMC Internal Feature – Center Plane
4.10	Datum target lines and areas	6.6	Perpendicularity – External feature of Size Axis
4.11	Primary Datum Axis Established by Datum Target points on a Single Cylindrical Feature	6.7	Perpendicularity – Internal feature of size axis
4.12	Equalizing Datum	6.8	Perpendicularity –Threaded Hole or Inserts Projected Tolerance Zone
4.14	Secondary Datum Axis	6.9	Parallelism
4.15	Workshop	6.10	Parallelism Control of Two Hole Features
5.1	Form	6.11	Parallelism Hole Relative to Plane
5.2	General	6.12	Angularity Overview and Surface to Surface
5.3	Flatness	6.13	Angularity Surface to Surface with Location Control
		6.14	Angularity Hole to Planar Datum
		6.15	Workshop

Table of Contents

Tolerances of Location	
7.3	General
7.4	Fundamental explanation of positional tolerancing
7.5	Differences between position tolerancing and limit tolerancing
7.6	Modifiers
7.7	Maximum material condition MMC
7.8	Least material condition LMC
7.9	External feature of size position tolerance boundaries with MMC modifier
7.10	Internal feature of size position tolerance boundaries with MMC modifier
7.11	External feature of size position tolerance boundaries with LMC modifier
7.12	Internal Feature of Size Position Tolerance Boundaries with LMC Specification
7.13	Zero positional tolerance at MMC
7.14	Position Tolerance at RFS
7.15	Positional Tolerance Axis and Surface Interpretation – Surface Datum’s
7.16	Positional Tolerance Axis and Surface Interpretation – Surface Datum’s
7.17	Positional Tolerance Axis Interpretation - Surface Datum’s
7.18	Positional Tolerance Surface Interpretation - Surface Datum’s
7.19	Positional Tolerance Axis and Surface Interpretation - Thru Hole Datum’s
7.20	Positional Tolerance Axis Interpretation – Thru Hole Datum’s
7.21	Positional Tolerance Surface Interpretation – Thru Hole Datum’s
7.22	Rectangular Coordinate Method
7.23	Positional tolerance at MMC relative to hole and slot datum feature
7.24	Bi-directional positional tolerancing, polar coordinate method
7.25	Different positional tolerance at each surface
7.26	Circular pattern of holes
7.27	Positional tolerance at MMC relative to datum feature center planes
7.28	Positional tolerance at RFS of slots relative to surface datum features
7.29	Coaxial cylinders
7.30	Coaxial (Concentric) Control of Multiple Hole-Counterbore Holes
7.31	Coaxial control of cylinders
7.32	Hole pattern located perpendicular to cylindrical datum
7.33	Holes Not Normal to DRF
7.34	Hole Pattern Located at Angle to Datum Reference Frame
7.35	Positional Tolerance at MMC of Spherical Feature
7.36	Positional Tolerance of Coaxial Holes of Same Size
7.37	Least Material Condition Application – Cylinder Wall Thickness
7.38	Positional Tolerance for Coaxiality With Datum Feature referenced at MMC
7.39	Positional Tolerance for Coaxially with Feature Referenced at Zero MMC Relative to Datum Feature at MMC
7.40	Positional Tolerance - Thru Hole Datum’s at MMC

Table of Contents

7.41	Positional Tolerance - Thru Hole Datum Verification	10.1	Coaxial Tolerance Comparison
7.42	Composite Positional Tolerancing	11.1	Tolerance Analysis
7.45	Concentricity	11.2	Series Stack
7.48	Symmetry	11.3	Floating Fastener Condition
7.49	Workshop	11.4	Fixed Fastener Condition
8.1	Profile	11.5	Tolerance Compensation for Projected Tolerance Zone – Fixed Fastener condition
8.2	General	11.7	Two Mating Coaxial/Coplanar Features at MMC
8.3	Profile of surface, bilateral tolerance	11.8	Tolerance Compensation for Projected Tolerance Zone Fixed Fastener Condition
8.4	Profile of Surface, Bilateral Tolerance Rectangular Coordinate Dimensioning Without Dimension Lines	11.10	Three Mating Coaxial/Coplanar Features at MMC
8.4	Profile of Surface, unilateral (Inside) tolerance	11.11	Position Tolerance calculation and Hole Pattern Analysis
8.5	Profile of surface, unilateral (Outside) tolerance	11.12	Position Tolerance Verification
8.6	Profile of surface, bilateral unequal tolerance	11.13	Position Coordinate to Location Conversion Chart
8.7	Profile of surface, all around	11.15	Workshop
8.8	Profile of Surface, Independent Form Control	12.1	Critical Feature Drawing (Reduced Dimension Drawing)
8.9	Profile of Surface, Boundary Principle	12.2	General
8.10	Profile Tolerance for Coplanar Surfaces	12.3	Implementation Considerations Applicable documents, Overview
8.12	Profile of Line	12.4	Design Drawing Requirements Digital Model and Database
8.13	Composite Profile Tolerance	12.5	Quality and inspection requirements Manufacturing
8.14	Composite Profile Application	12.6	Change notice procedure
8.15	Workshop		
9.1	Runout		
9.2	General		
9.3	Circular runout		
9.4	Total runout		
9.5	Workshop		

Table of Contents

13.1 **Definitions and Terminology (Glossary)**

14.1 **Symbol Comparison**

14.2 Comparison of ASME and ISO Symbols (Geometric Characteristics)

14.3 Comparison of ASME and ISO Symbols (General)

Introduction

Geometric Dimensioning and Tolerancing (GD&T) is an engineering drawing language used to communicate the physical limit requirements of a product object in two or three dimensional space. The GD&T standard defines a collection of symbols and specific rules for defining specific characteristics, relationships, and feature controls.

The latest standard on the subject of GD&T defined and in practice is the American Society of Mechanical Engineers (ASME) Y14.5M – 1994 Dimensioning and Tolerancing. The GD&T standard used internationally is the International Institute Standard (ISO) 1101:2004, Technical Drawings - Geometrical Tolerancing and associated standards.

The following are ISO standards that define GD&T requirements:

ISO/129-	Technical Drawings General Principles
ISO/406-	Technical Drawing Linear and Angular Dimensions
ISO/1101-	Technical Drawings Geometrical Tolerancing
ISO/1660-	Technical Drawings Profiles
ISO/2692-	Technical Drawings Maximum Material Condition
IOS/2692:1998/DAM 1	Technical Drawings Least Material Condition
ISO/3040-	Technical Drawings Cones
ISO/5458-	Technical Drawings Positional Tolerancing
ISO/5959-	Technical Drawings Datums and Datum Systems
ISO/7083-	Technical Drawings Symbols Proportions
ISO/8015-	Technical Drawings Fundamental Tolerance Principle
ISO/10579-	Technical Drawings Non-Rigid Parts
IOS/10587-	Technical Drawings Projected Tolerance Zones

Declarations:

All illustration and drawings are depicted and interpreted per Figure 0.1

THIRD ANGLE PROJECTION	UNLESS OTHERWISE SPECIFIED DIM ARE IN INCHES TOL ON ANGLE $\pm 0^{\circ} 30'$.XX $\pm .03$.XXX $\pm .010$.XXXX $\pm .0001$ INTERPRET DRAWING AND TOL PER ASME Y14.5M-1994
------------------------	---

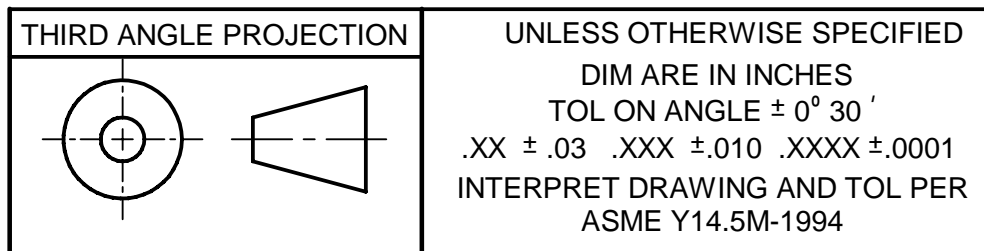


Figure 0.1

DIMENSIONING AND TOLERANCING

How the Geometric Dimensioning and Tolerancing System Works

Dimensioning and tolerancing is a means to communicate the geometry requirements of a particular part or assembly. Depending on the function, feature relationships, manufacturing or definition requirements, one will then define the level or extent of details for the part. Geometric Dimensioning and Tolerancing (GD&T) standard ASME Y14.5M-1994 is a defined system of rules, symbols, and explicit requirements to fully delineate an objects geometric requirements.

The following are the more common reasons to apply GD&T principles:

- Part features are critical to function or inter-changeability.
- When functional gauging techniques are desired.
- When a common reference (origin) or datum is required to ensure communication is consistent between design, manufacturing and inspection.
- When a standard interpretation or tolerance is not already implied.
- Simplify tolerance analysis.
- Replace complex or long geometry requirement description notes with a single geometric symbol.

Geometric Characteristics and Symbols

Geometric characteristic symbols are used to define a simple or complex feature requirement or relationship. GD&T characteristics and categories are:

	TOLERANCE TYPE	CHACTERISTIC	SYMBOL
FOR INDIVIDUAL FEATURES	FORM	FLATNESS	
		STRAIGHTNESS	
		CYLINDRICITY	
		CIRCULARITY	
FOR INDIVIDUAL OR RELATED FEATURES	PROFILE	PROFILE OF A SURFACE	
		PROFILE OF A LINE	
FOR RELATED FEATURES	ORIENTATION	PERPENDICULARITY	
		PARALLELISM	
		ANGULARITY	
	LOCATION	POSITION	
		CONCENTRICITY	
		SYMMETRY	
	RUNOUT	TOTAL RUNOUT	
CIRCULAR RUNOUT			

See outside-back cover of this book for an expanded geometric characteristics chart.