

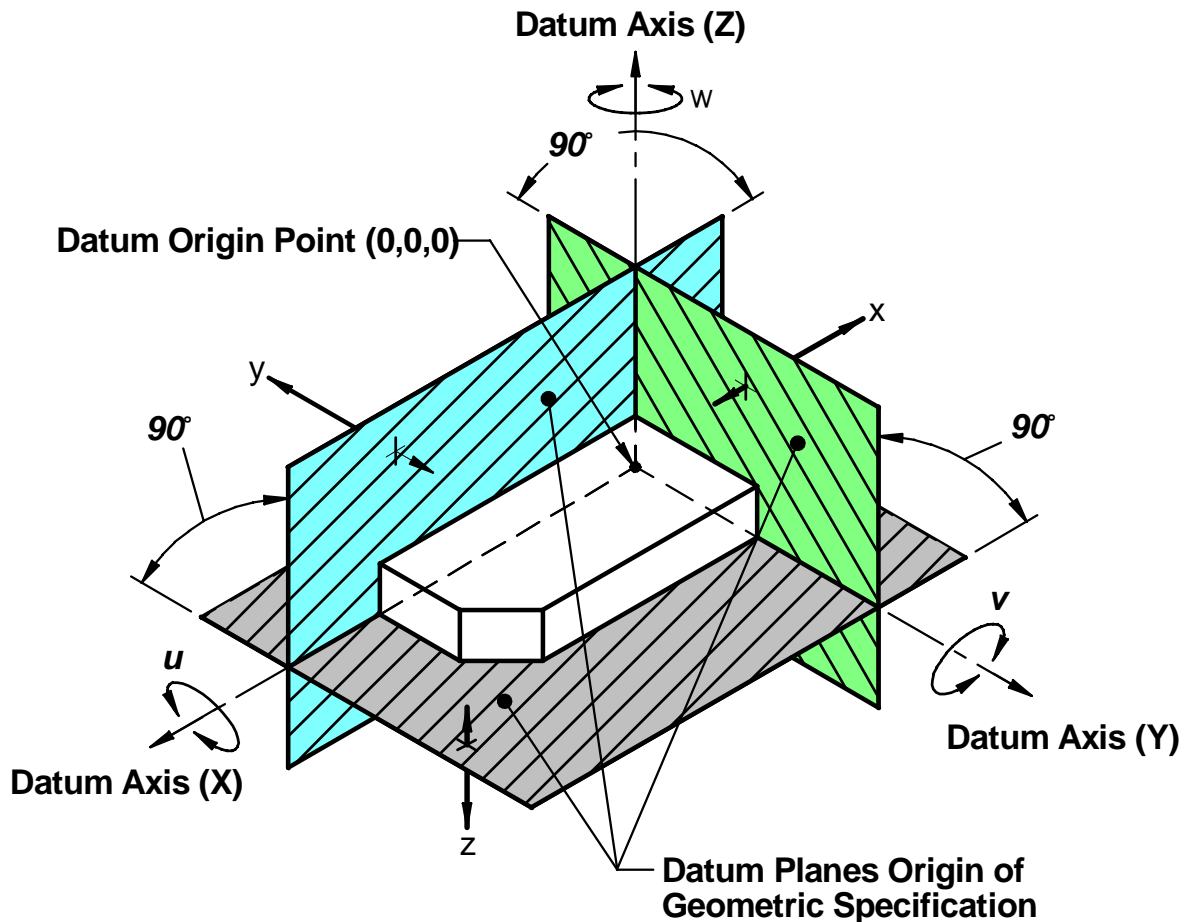
Geometric Boundaries II

*Interpretation and Application
of Geometric Dimensioning and Tolerancing
(Using the Inch and Metric Units)
Based on ASME Y14.5-2009 (R2004)*

Written and Illustrated by
Kelly L. Bramble

Published by:
Engineers Edge, LLC
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.

Kelly L. Bramble

Copyright 2009, 2010, 2011 All rights 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 J

Acknowledgments

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

Engineers Edge 2000 - 2009, Solutions by Design – www.engineersedge.com
Design for Manufacturing 2006 - 2009, Kelly Bramble
Geometric Boundaries Based on ASME Y14.5M-1994
Geometrical Boundaries Based on ISO 1101(E) 2004, Kelly Bramble
ANSI/ASME B94.6-1984 (R2003), Knurling
ANSI B4.2-1978 (R2004), Preferred Metric Limits and Fits
ASME Y14.5-2009, Dimensioning and Tolerancing.
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
ISO/R1101 (E)-2004, & 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/ASME B89.6.2-1973 (R2003), Temperature and Humidity Environment for Dimensional Measurement
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 Y14.1M-2005, Metric Drawing Sheet Size and Format
ASME Y14.2M-1992, Line Conventions and Lettering
ASME Y14.2-2008, Line Conventions and Lettering
ASME Y14.5.1M-1994, Mathematical Definition of Dimensioning and Tolerancing Principles.
ANSI Y14.6aM-1981 (R1998), Screw Thread - Representation (Metric Supplement)
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

Acknowledgments Continued

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

ANSI/IEEE 268-1992, Metric Practice

ANSI/ASME B1.2-1983, Gauges and Gauging for Unified Inch Screw Threads

ANSI B4.4M-1981 (R1987), Inspection of Workpieces

ASME B5.10-1994, Machine Tapers — Self Holding and Steep Taper Series

ASME Y1.1-1989, Abbreviations

ASME Y14.3M-1994, Multiview and Sectional View Drawings

ASME Y14.41-2003 (R2008), Digital Product Definition Data Practices

ASME Y14.43-2003 (R2008), Dimensioning and Tolerancing Principles for Gages and Fixtures

IEEE/ASTM SI 10-2002 ERRATA 2005, Standard for Use of the International System of Units (SI) — The Modern Metric System

Table of Contents

- 1.2 Preface
- 1.3 Acknowledgments
- 1.4 Table of Contents
- 1.12 Introduction
- 1.13 Standards Based on ISO or ASME Standards
- 1.15 Corporate Standards
- 1.16 How the Geometric Dimensioning and Tolerancing System Works
- 1.17 Tolerances, Features and Characteristics Overview
- 1.18 Tolerance Hierarchy
- 1.19 Feature Control Frame
- 1.20 Position and Limit Tolerance General Overview and Contrast
- 1.21 Common Symbols
- 1.22 Dimensioning and Tolerancing General Rules
- 2.1 **Limit Tolerancing**
- 2.2 General – Dimensioning System Limit Tolerancing
- 2.3 Limit tolerancing, Square Tolerance, Zone and Overview
- 2.4 Tolerance Application and Method, Basic Dimensions
- 2.5 Implied 90 Degree Angle, Dimensional Expression
- 2.6 Dimension and Tolerance Expression – Millimeter Tolerances and Dimensions
- 2.8 Inch Tolerances and Dimensions
- 2.10 Angular Dimensioning and Tolerancing
- 2.11 Slotted holes, Arcs, Countersink hole
- 2.12 Counter bored Holes, Countersink on curved surface
- 2.13 Internal External Chamfers, Keyseats
- 2.14 Rectangular Coordinate Dimensioning
- 2.15 Rectangular Coordinate Dimensions in Tabular Form, Polar Dimensioning
- 2.16 Repetitive Features
- 2.17 Flat Taper
- 2.18 Conical Taper
- 2.19 Statistical Tolerancing
- 2.20 Dimension Origin
- 2.21 Screw Threads Tolerance Application, Gears and Splines
- 2.22 Angular Surface Defined by Limit and Angle Dimension
- 2.23 Radius Tolerance
- 2.24 Controlled Radius Tolerance
- 2.25 Knurling, Rods and Tubing Specification, Surface Roughness or Texture
- 2.26 Features With and Without Size
- 2.27 Material Condition, MMC, MMB LMC, LMB
- 2.28 Rule #1, Envelope Principle
- 2.29 Exceptions to Rule #1, Independency Application
- 2.32 Rule #2 Material Condition Requirements
- 2.33 Continuous Feature
- 2.35 Limitation of the Limits of Size
- 3.1 **Datums**
- 3.2 Datum Reference Frame, DRF - General
- 3.3 Immobilization of component and measurement
- 3.4 Datum identification General
- 3.5 Datum symbols and identification Datum identification features without size
- 3.6 Datum identification features with size
- 3.7 Datum Identification Features with Size Alternative and Special Applications

Table of Contents

3.8 Datum associated with feature control frame	4.1 Datum Targets
3.9 Datum feature, simulated datum, and theoretical datum plane	4.2 General
3.10 Primary External Datum Diameter	4.3 Datum Target Point Symbol, Application
3.11 Primary internal Datum Diameter	4.4 Datum Target Area
3.12 Primary External Datum Width	4.5 Datum Target Line
3.13 Primary Internal Datum Width	4.6 Dimensioning Datum Targets
3.14 Primary Datum Features Reference	4.7 Primary Datum Plane Established by Three Datum Target Areas
3.15 Setup and Inspection of Datum's, Datum and Dimensional Measurement equipment	4.8 Primary Datum Plane Established by Two Datum Target Points and One Datum Target Line.
3.17 Sequence of Datum Features	4.9 Step Datum Feature
3.18 Sequence of Datum Features Relates Part to Datum Reference Frame	4.10 Datum Target Lines and Areas
3.19 Parts with Angular Orientation	4.11 Primary Datum Axis Established by Datum Target points on a Single Cylindrical Feature
3.20 Cylindrical Datum Feature	4.12 Equalizing Datum
3.22 Orientation of Two Datum Planes Through Hole Features	4.14 Secondary Datum Axis
3.24 Datum reference Frame With Translation Modifier	4.15 Spherical Movable Datum Target
3.26 Partial Datum Surface(s) as Datum Features	5.1 Form
3.27 Partial Contoured Datum Surface	5.2 General
3.28 Multiple Datum Features, Single Datum	5.3 Flatness
3.29 Planar Multiple Datum Features Offset	5.4 Flatness Applied on a Unit Basis
3.30 Contoured Datum Feature Constraining a Rotational Degree of Freedom, Contoured Datum Feature at MMB	5.5 Flatness Applied on Unit Basis With Overall Control
3.31 Planar Datum Feature Constraining Rotational Degree of Freedom	5.6 Flatness Applied to Derived Median Plane at RFS
3.32 Conical Datum Feature Constraining a Rotational Degree of Freedom	5.7 Flatness Applied to Derived Median Plane at MMC
3.33 Conical Datum Feature Constraining Rotational Degree of Freedom With secondary Datum Constraint	5.8 Straightness
3.34 Inclined Datum Features	5.9 Straightness Per Unit Basis
	5.10 Straightness Applied in Two Directions
	5.11 Straightness of a Surface (Cylindrical)
	5.12 Straightness of a Feature of Size @ RFS
	5.13 Straightness of a Feature of Size @ MMC

Table of Contents

- 5.14 Straightness Per Unit Length With Specified Total Straightness
- 5.15 Cylindricity
- 5.16 Circularity (Roundness)
- 5.17 Circularity of Cone
- 5.15 Circularity of Sphere
- 5.16 Circularity or Cylindricity Applied with Average Diameter

- 6.1 **Tolerance of Orientation**
- 6.2 General - Overview
- 6.3 Perpendicularity - Surface
- 6.4 Perpendicularity – Surface Two Datum's
- 6.5 Perpendicularity – Center plane
- 6.6 Perpendicularity at MMC Internal Feature – Center Plane
- 6.7 Perpendicularity – External Feature of Size Axis
- 6.8 Perpendicularity – Internal Feature of Size Axis
- 6.9 Perpendicularity – Threaded Hole or Inserts Projected Tolerance Zone
- 6.10 Parallelism
- 6.11 Parallelism Control of Two Hole Features
- 6.12 Parallelism Hole Relative to Plane
- 6.13 Angularity Overview and Surface to Surface
- 6.14 Angularity Surface to Surface with Location Control
- 6.15 Angularity Hole to Planar Datum
- 6.16 Secondary Datum Application

- 7.1 **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 and Boundary
- 7.8 Least Material Condition and Boundary
- 7.9 External Feature of Size Position Tolerance Boundaries with Maximum Material Condition MMC Specification and Datum's at RMB
- 7.10 Internal Feature of Size Position Tolerance Boundaries with Maximum Material Condition MMC Specification and Datum's at RMB
- 7.11 External Feature of Size Position Tolerance Boundaries with Least Material Condition LMC Specification and Datum's at RMB
- 7.12 Internal Feature of Size Position Tolerance with Least Material Condition LMC Specification and Datum's at RMB
- 7.13 Zero Positional Tolerance at MMC, Datum's at RMB
- 7.14 Position Tolerance at RFS and Datum's at RFB
- 7.15 Positional Tolerance at MMC Surface and Axis Interpretation of a Hole Feature
- 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

Table of Contents

- 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 (Concentric) Control of 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 MMB
- 7.39 Positional Tolerance for Coaxially With Feature Referenced at Zero MMC Relative to Datum Feature at MMB
- 7.40 Positional Tolerance - Thru Hole Datum's at MMC
- 7.41 Positional Tolerance - Thru Hole Datum Verification
- 7.42 Composite Positional Tolerancing
- 7.46 Composite Positional Tolerancing With Pattern Orientation Control
- 7.49 **Concentricity**
- 7.51 Concentricity Application
- 7.52 Concentricity of Sphere
- 7.53 **Symmetry**
- 8.1 **Profile**
- 8.2 General
- 8.3 Profile Surface Definition
- 8.4 Profile of surface, Bilateral tolerance
- 8.5 Profile of Surface, Bilateral Tolerance Rectangular Coordinate Dimensioning Without Dimension Lines
- 8.6 Profile of Surface, Unilateral (INSIDE) Tolerance
- 8.7 Profile of Surface, Unilateral (OUTSIDE) Tolerance
- 8.8 Profile of Surface, Bilateral Unequal Tolerance
- 8.9 Profile of Surface, All Around
- 8.10 Profile of Surface, All Over
- 8.11 Profile of Surface, Independent Form Control
- 8.12 Profile of Surface, Boundary Principle
- 8.13 Profile Tolerance for Coplanar Surfaces
- 8.14 **Profile of Line**
- 8.15 Profile of Line Tapered Shape
- 8.16 Profile of Line Unilateral Inside
- 8.17 Profile of Line and Size Control

Table of Contents

- 8.18 Composite Profile Tolerances
- 8.20 Composite Profile Tolerance With Rotation Control of PRC
- 8.21 Non-Uniform Zone
- 8.22 Non-Uniform Zone Application
- 8.23 Profile of Surface With Datum at MMC

- 9.1 **Runout**
- 9.2 General
- 9.3 Circular Runout
- 9.6 Total Runout
- 9.9 Run-Out With Plane as Datum
- 9.10 Run-Out Relative to Datum Surface and a Diameter
- 9.11 Run-Out With Two Datum Diameters
- 9.12 Multiple Cylinders Related to Each Other Runout Application
- 9.13 Runout Relating Separate Features to Each Other with Common Datum

- 10.1 **Coaxial Tolerance Comparison**

- 11.1 **Tolerance Analysis**
- 11.2 Tolerance Conversion Unilateral – Bilateral
- 11.3 Series Stack
- 11.4 Floating Fastener Condition
- 11.5 Fixed Fastener Condition
- 11.6 Tolerance Compensation for Projected Tolerance Zone – Fixed Fastener condition
- 11.8 Two Mating Coaxial/Coplanar Features at MMC
- 11.9 Tolerance Compensation for Projected Tolerance Zone Fixed Fastener Condition
- 11.10 Three Mating Coaxial/Coplanar Features at MMC
- 11.11 Position Tolerance Calculation and Hole Pattern Analysis

- 11.12 Coordinate to Position Tolerance Conversion Chart
- 11.13 Position Tolerance Verification Workshop
- 11.17 Generic Hole Verification Chart
- 12.1 **Critical Feature Drawing (Reduced Dimension Drawing)**
- 12.2 General
- 12.3 Implementation Considerations Applicable Documents, Overview
- 12.4 Design Drawing Requirements Digital Model and Database
- 12.5 Quality and Inspection Requirements Manufacturing
- 12.6 Change Notice Procedure

- 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)
- 14.4 Principal Changes and Revisions within ASME Y14.5-2009

- 15.1 **Appendix**
- 15.1 ANSI Standard Size (Inch) Drills
- 15.3 ISO Metric (mm) Size Standard Drill Sizes
- 15.4 ANSI External Screw Threads Sizes 0 - 3/8
- 15.5 ANSI External Screw Threads Sizes 3/8 – Larger
- 15.6 ACME Thread Forms – General Purpose External
- 15.7 ACME Thread Forms – General Purpose Internal

Table of Contents

- 15.8 ACME Thread Forms – Internal
Centralizing
- 15.9 ACME Thread Forms – External
Centralizing
- 15.10 Standard External Pipe Threads
- 15.11 Standard Internal Pipe Threads

- 16.1 Index

This page left blank intentionally.

Introduction

Geometric Dimensioning and Tolerancing (GD&T) is an engineering drawing language used to communicate the physical 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 Y14.5 – 2009 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
ISO/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
ISO/10587-	Technical Drawings Projected Tolerance Zones

Declarations:

All illustrations and drawings are depicted and interpreted per Figure 0.1

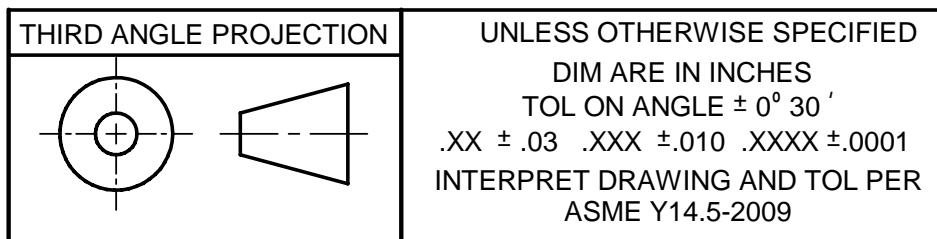


Figure 0.1

Actual Local Size, 2.31, 2.34, 5.6, 5.7, 5.14, 13.1
Angle Tolerances, 2.9, 2.10, 2.11, 2.12, 2.15, 2.16, 2.22
All Around, 1.20, 5.15, 7.51, 8.9, 14.3
All over, 1.20, 8.10, 13.5
ASME, 1.13, 1.16, 1.19
Angles, 2.5, 2.25
Angularity, 6.2, 6.13, 6.14, 6.15, 6.16, 13.1 14.2
ANSI, 1.19, 15.1
Arc Length, 1.20, 2.11, 14.3
Average Diameter, 5.19, 13.3
Arcs, 2.11, 2.23, 2.24, 7.24, 8.3
Axis, 1.22, 2.2, 3.2, 3.6, 3.7, 3.8, 3.10, 3.11, 3.14, 3.22, 4.15, 5.11
Basic Dimension, 1.19, 2.4, 2.5, 2.7, 2.8, 7.4, 13.1
Bilateral Tolerance, 8.4, 8.5, 13.1,
Center Lines, 1.22, 2.5,
Chamfers, 2.13,
Chords, 2.11
Circularity, 5.2, 5.16, 5.17, 5.18, 13.1
Circular Runout, 9.2, 9.3, 9.4, 13.7, 14.2
Coaxial Features, 9.2
Composite Positional Tolerancing, 7.42, 7.46
Composite Profile Tolerance, 8.18, 8.20
Concentricity, 7.3, 7.49, 7.51, 7.52
Contoured Datum Feature, 3.30
Controlled Radius, 1.20, 2.24, 14.3
Coordinate Dimension, 2.14, 2.15, 8.5, 13.2, 13.4
Continuous Feature, 2.33
Coplanarity, 13.2,
Conical Datum, 3.33
Conical Taper, 2.18
Conterbored, 2.12
Counterdrilled, 2.11
Countersunk, 2.11, 2.12

Critical Feature Drawing (CFD), 12.1
Cylindricity, 6.1, 5.15, 13.2, 14.2
Datum, 3.2
Datum Feature Simulator, 3.9, 3.10, 3.11, 3.12, 3.13
Datum Symbol, 3.5
Datum Target, 4.2
Datum Translation, 3.24,
Datum, Permanent, 3.3
Datum, Primary, 1.18
Datum, Secondary, 1.18,
Datum reference Frame (DRF), 3.2
Datum, Temporary, 3.3
Datum, Tertiary, 1.18
Degrees of Freedom, 3.2, 3.32, 3.33
Dimension Origin, 2.20
Envelope Principle (Rule #1), 2.28, 2.29, 2.30
Extension Lines, 1.21, 2.14, 2.33, 5.3
Feature Control Frame (FCF), 1.18, 1.20, 2.5, 13.4, 14.2
Feature of Size (FOS), 2.27, 13.4
Fixed Fastener Condition, 11.4, 11.5
Finish, 1.17,
Flatness, 2.31, 5.2, 5.3, 5.4, 5.5, 5.6, 5.7, 13.4
Floating fastener Condition, 11.3
Flat Tapers, 2.17
Form Tolerance, 5.1
Free State Condition, 1.22,
Fundamental Rules Dimensioning, 1.21
Gears, 2.21
Geometric Tolerance, 13.6, Back Cover
Implied 90 Degree Angle, 2.5
Independency, 1.20, 2.31
ISO, 14.2
Knurling, 2.25
Least Material Boundary (LMB), 1.20, 2.27, 7.6, 13.5
Least Material Condition (LMC), 1.18, 1.20, 2.27, 7.6, 13.5
Machining Centers, 2.6
Maximum Material Condition (MMC), 1.18, 1.20, 2.27, 7.6, 13.5
Maximum Material Boundary (MMB), 1.20, 2.27, 7.6, 13.5
Order of Precedence, 3.3, 3.17, 3.18, 3.20, 7.17
Orientation, 1.15, 1.16, 1.17, 2.5, 6.2

Parallelism, 1.15, 6.2, 6.5, 6.10, 6.11
Perpendicularity, 1.15, 6.4, 6.6, 6.7, 6.10, 6.1, 6.16
Plus and minus Tolerancing (Limit), 2.2, 2.3, 2.9
Position Tolerancing, 7.2, 13.6
Profile Tolerancing, 8.2, 13.6
Profile of Line, 8.2, 8.14, 8.15, 8.16, 8.17,
Profile of Surface, 8.4, 8.5, 8.6, 8.7, 8.8, 8.9, 8.10, 8.11
Projected Tolerance Zone, 6.9, 11.5, 11.8,
Radius, 1.20, 2.23, 2.24,
Reference Dimension, 1.20, 1.20, 13.6, 14.3,
Regardless of Material Boundary (RMB), 2.27, 13.6
Regardless of Material Condition (RFS), 2.27, 13.6
Repetitive Features, 2.16
Resultant Condition, 6.6, 6.7, 6.8, 7.9, 7.10, 7.11, 7.12, 7.13, 7.35, 13.6
Roundness, see Circularity
Rule #1 – see Envelope Principle
Rule #2, 1.18
Runout, 1.15, 9.2, 9.3
Screw Threads, 2.21, 13.4, 15.4, 15.5
Slotted Holes, 2.11
Statistical Tolerancing, 2.19
Straightness, 5.8, 5.9, 5.10, 5.11, 5.12, 5.13, 5.14
Symmetry, 1.15, 7.3, 7.53, 13.7, 14.2
Splines, 2.21,
Spotface, 1.20, 14.3
Square Symbol, 1.20
Tangent Plane, 1.20, 7.6, 13.7, 14.2
Tolerance Analysis Stacks, 11.1
Total Runout, 1.15, 1.16, 9.2, 9.6, 13.8
True Position, 13.7
Virtual Condition, 5.7, 5.13, 5.14, 6.6, 6.7, 6.8, 7.9, 7.10, 7.11, 7.12, 7.13

ENGINEERS EDGE TRAINING

Engineers Edge, LLC training seminar, workshops, and lectures are available on an in-plant basis on geometric dimensioning and tolerancing. Training programs are based on current best industry practice and standards. These training programs can be conducted in either the customary (inch) system or SI units (metric).

Available classes:

- GD&T Intermediate (Interpretation -Application Combined)
- GD&T Basics (Interpretation)
- GD&T Custom
- Tolerance Analysis & Stacks
- Design for Manufacturing (DFM)

Other books available:

Engineering Design for Manufacturing

Geometric Boundaries ASME Y14.5M-1994, Imperial – Interpretation and Application

Geometric Boundaries SI – Interpretation and Application ASME Y14.5M-1994

Geometrical Boundaries – ISO1101 (E) – 2004, Interpretation and Application

For detail information – please visit www.engineersedge.com