



Applied Geometrics Inc.

Fundamentals of Geometric Dimensioning & Tolerancing (GD&T) Per ASME Y14.5-2009



No part of this document may be reproduced in any form
in an electronic retrieval system or otherwise,
without prior written permission of Applied Geometrics, Inc.

Copyright © 2018
Applied Geometrics, Inc.
All rights reserved
Printed in U.S.A.

Table of Contents

Forward	7
----------------------	----------

Biographies	8
--------------------------	----------

Chapter 1

INTRODUCTION - GD&T OVERVIEW	11
---	-----------

1.0 Introduction.....	11
1.1 Course Objectives.....	11
1.2 Fundamental Terms.....	11
1.3 Functional Relationship	11
1.4 Geometric Dimensioning and Tolerancing	12
1.4.1 History	12
1.5 Simplest Drawing in the World	13
1.5.1 Rule #1	14
1.5.2 Other Ambiguity	14
1.6 When to Use GD&T	17
1.6.1 Quality versus Tight Tolerances.....	17
1.6.2 Cost of Size & Position Tolerance.....	18
1.6.3 Why use GD&T - Because GD&T:.....	19
1.6.4 Advantages of Using Symbols	19
1.7 Example Drawing Without GD&T.....	20
1.8 The GD&T Language.....	21
1.8.1 The Geometric Characteristic Symbols	24
1.8.2 Four Ways to Control a Feature	24
1.8.2.1 The Rememberall.....	24
1.8.2.2 Size Controls	25
1.8.2.3 Form Controls.....	25
1.8.2.4 Orientation Controls	25
1.8.2.5 Location Control	26
1.8.2.6 Combination Controls	26
1.8.3 Terms & Symbols.....	26
1.8.3.1 Repetitive Information	28
1.8.3.2 Symbols Summary	28
1.8.3.3 BASIC Dimensions	29
1.8.3.3.1 Three Methods to Show Basic Dimensions.....	29
1.8.3.3.2 Basic Dimensions with Datums.....	30
1.8.3.3.3 Implied Basic Dimensions	31

Chapter 2

Flange Scenario.....	32
-----------------------------	-----------

2.0 Flange Scenario	32
2.1 Flange: Hole Pattern Measurement	33
2.2 Flange: Hole Pattern Controls	33
2.2.1 Exercise: Sketch Position Tolerance Zone	37
2.3 Exercise: Is this Drawing Complete?.....	39
2.3.1 Is this a Complete Solution?.....	40
2.3.2 This is a Complete Solution.....	41

Chapter 3

Definitions & Concepts42

3.0 Features 42

3.1 Type of Features. 42

3.1.1 Exercise: Features of Size and Total Features 43

3.1.2 Features of Size 44

3.2 Not All Dimensions Are Features 44

Chapter 4

Three Big Concepts45

4.0 The Three Big Fundamental Concepts. 45

4.1 Big Concept #1: Datums 45

4.1.1 Feature Control Frames 45

4.1.2 Check List to Read Feature Control Frames. 47

4.1.3 Spherical Diameter Symbol 48

4.2 Big Concept #2: Rule #1. 48

4.3 Big Concept #2: Material Condition Modifiers 48

Chapter 5

Material Condition Modifiers.49

5.0 Material Condition Modifier Symbols in Feature Control Frames 49

5.1 Maximum Material Condition 50

5.2 Least Material Condition. 57

5.2.1 Least Material Condition Application 59

5.3 Regardless of Feature Size 61

5.4 Material Condition Exercises 63

Chapter 6

Material Boundaries65

6.0 Maximum & Least Material Boundaries. 65

6.1 Virtual Condition. 66

6.1.2 MMC Virtual Condition External Feature of Size - “Largest Plus” 66

6.1.3 MMC Virtual Condition Internal Feature of Size - “Smallest Minus” 67

Chapter 7

Floating & Fixed Fastener Formulas70

7.0 Formulas for Interchangeability 70

7.1 Exercise: Floating Fastener. 70

7.1.1 Exercise: Calculate Virtual Condition 71

7.2 Exercise: Fixed Fastener 71

7.2.1 Exercise: Calculate Virtual Conditions 72

Chapter 8

Rule #1 Material Boundaries.73

8.0 Rule #1 By Default 73

8.1 Rule #1 - LMC Specified 75

8.2 Rule #1 Exceptions or Not Applicable 76

8.2.1 Independency 77

8.2.2 Continuous Feature 77

Chapter 9

Other General Rules79

9.0 Rule #2 - Material Condition Rule 79

9.1 Pitch Diameter Rule for Threads 79

9.2 Pitch Diameter Rule For Gears & Splines 80

Chapter 10

Datums: Theory to Reality81

10.0 Technical Terminology 81

10.1 Definitions 81

10.2 Datum Feature Symbol 82

10.2.1 Application of the Datum Feature Symbol 83

10.3 The Datum Reference Frame 85

10.3.1 Degrees of Freedom 85

10.3.1.1 Planar Datum Features 85

10.3.1.2 Cylindrical Datum Features (Holes & Pins) 86

10.3.1.3 Width Datum Features Width (Slots & Tabs) 86

10.2.1.4 Spherical Datum Features (Balls & Sockets) 86

10.3.2 Can / May / Must 87

10.4 Datum Precedence 88

10.4.1 Datum Feature Simulators 88

10.5 How Datum Theory Really Works 90

10.6.1 Cylindrical & Width Datum Features - (Primary & Secondary) 97

10.6.2 Datum Feature of Size Precedence Examples 100

10.7 Datum Feature Simulators for |A|D|E| 102

10.8 Datum Feature Design Review Scenario 104

10.9 Coordinate System Display 108

10.10 Datum Target & Area Symbols 111

10.10.1 Movable Datum Targets 111

Chapter 11

Position Tolerance113

11.0 Directly Toleranced Linear Dimension Tolerance Zone 113

11.0.1 Cylindrical Tolerance Zones 114

11.1 Position with Bonus Tolerance 115

Chapter 12

Form Controls118

12.0 Straightness 118

12.1 Surface Straightness On A Planar Face 119

12.1.1 Surface Straightness On A Cylinder 120

12.1.2 Derived Median Line Straightness (DML) - RFS 121

12.1.3 Derived Median Line Straightness (DML) - MMC 122

12.2 Flatness 123

12.2.1 Surface Flatness 123

12.2.2 Derived Median Plane Flatness - (DMP) 125

12.3 Circularity 126

12.4 Cylindricity 128

Chapter 13

Orientation Controls.129

13.0 Orientation Controls 129

13.1 Perpendicularity of a Surface 129

13.2 Perpendicularity of a Width Feature of Size (Slot/Tab) 130

13.3 Perpendicularity of a Cylindrical Feature (Total Width Tolerance Zone) 131

13.4 Perpendicularity of a Cylindrical Feature (Cylindrical Tolerance Zone) 132

13.5 Angularity of a Surface 133

13.6 Parallelism of a Surface to a Datum Plane 134

13.7 Parallelism of an Axis to a Datum Plane (Total Width Tolerance Zone) 135

13.8 Parallelism of an Axis to a Datum Axis 137

Chapter 14

Profile.....138

14.0 Profile 138

14.1 Profile of a Surface..... 138

14.1.1 Profile of a Surface Unilateral & Unequally Disposed Tolerance Zones 140

14.1.2 Profile Between Two Points 141

14.1.3 Profile All Around 142

14.1.4 Profile All Over 143

14.1.5 General Profile Note 143

14.1.6 Profile to Control Co-Planarity..... 144

Chapter 15

Runout145

15.0 Runout (Circular and Total) 145

15.1 Circular Runout..... 146

15.2 Total Runout 148

Chapter 16

Concentricity.....150

16.0 Concentricity..... 150

16.1 Concentricity vs. Coaxiality 150

16.2 Concentricity Specification 151

Chapter 17

Symmetry153

17.0 Symmetry..... 153

17.1 Symmetry Specification 153

Chapter 18

AGI Geometric Design Process.....155

18.0 AGI Geometric Design Process..... 155

18.1 Datum Recognition Criteria 155

18.2 Datum Feature Selection and Design Criteria 155

18.3 Datum and Datum Feature Selection Exercise..... 156

Appendix162

Forward

This text covers the fundamental concepts and rules of GD&T per the ASME Y14.5-2009 standard. What consists of fundamentals versus advance material is subjective. Applied Geometrics covers what many will consider the fundamentals such as the 14 geometric symbols, shapes of tolerance zones, and material modifiers for tolerance zones. Naturally, basic understanding of datum theory is also covered. This text also covers some significant details in each of these areas not always found in a GD&T fundamentals course. Applied Geometrics provides extensive discussion about the reality of GD&T with special regard to datum features, common mistakes made when applying GD&T and most importantly why.

Applied Geometrics has written this text to use the same language found when reading the ASME Y14.5-2009 standard. This greatly facilitates reading the actual standard for further study after completing this program.

Ultimately, three fundamental concepts are what Applied Geometrics wants each reader to understand in detail. Those concepts are understanding datum theory and reality, rule #1 (perfect form at MMC or LMC), and the material modifiers (MMC/MMB, LMC/LMB, and RRS/RMB) including the how, when and why of each concept.