Geometric Dimensioning and Tolerancing

Applications, Analysis & Measurement

[per ASME Y14.5-2009]

James D. Meadows

Table of Contents

Chapter Number	Chapter Title	Pages
1	**Symbols, Rules, Charts**	**1**
	-Geometric Characteristics	2
	-Symbols	3
	-New Symbols	5
	-Old Symbols, New Meanings	5
	-Charts-Food Chains of Symbology	6
	-A Few Basic Definitions, Formulas and Guidelines	8
	-New Rule Regarding the Use of Regardless of Feature Size	10
	-Maximum Material Boundary, Least Material Boundary and Regardless of Material Boundary	10
	-Actual Minimum Material Envelope vs. Actual Mating Envelope	11
	-Flatness of the Derived Median Plane	12
	-Types of Controls	13
	-Tolerances	13
	-General Rules for Tolerances	15
2	**Selecting a Tolerancing Approach**	**20**
	-Datums and Datum Features	21
	-Defining, Tolerancing and Qualifying Datum Features	23
	-Fixed Fastener Assembly Tolerancing Formula	24
	-Simultaneous Requirement Rule	25
	-Reading a Feature Control Frame	28
3	**Datum Feature Simulators**	**29**
	-Datum Feature Simulators: Physical and Imaginary	30
	-Fixtures, Gages and Virtual Condition Boundaries	31
4	**Boundaries and Material Condition Symbols, MMC, LMC & RFS**	**35**
	-Dimensioning and Tolerancing Overview	36
	-Rule #1: Size Tolerance and Form Tolerance are Interdependent	37
	-Exceptions to Rule #1	38
	-New Principle of Independency Symbol	38
	-GO Gages	40
	-Brief Comparison of Concentricity, Circular Runout, Total Runout and Position Tolerancing	42
	-Introduction of Orientation on Mating Parts	42
	-Material Condition Symbols and Concepts Explained	45
	-Regardless of Feature Size	45
	-Least Material Condition	48
	-Maximum Material Condition	49
	-Inner and Outer Boundary Calculations	50

5 *Major Concepts of Geometric Dimensioning and Tolerancing* **58**
--Converting from Plus and Minus Tolerance to Geometric Tolerance 58
-Position 63
-Profile 64
-Selecting Datum Features 64
-Size Tolerance Controls Form Tolerance (Rule #1), GO Gages 66
-Flatness 67
-Perpendicularity 68
-Mating Part Tolerancing 71
-Reading the Feature Control Frames as a Language 74
-Functional Gages 74
-Calculating Inner and Outer Boundaries 75
-Virtual Condition 75
-Resultant Condition 75
-Practical Absolute Gage Tolerancing 76
-Bonus Tolerancing Formulas 78
-Allowed vs. Actual Deviation from True Position Calculations 82
-Conversion Chart Inches 84
-Conversion Chart Millimeters 85
-Tolerance Zone vs. Boundary Verification 88
-Another Difference between Bonus Tolerance (Growth) and Datum Shift (Movement) of Tolerance Zones 89

6 Form **93**
-Flatness 95
-Straightness 100
-Cylindricity 109
-Circularity (Roundness) 113
-Spherical Diameters Controlled with Circularity 118
-Average Dimensions 119

7 Orientation **120**
-Parallelism 122
-Parallelism of a Tangent Plane 128
-Perpendicularity 129
-Angularity 136
-Angularity of a Tangent Plane 138
-Angularity as a Refinement of Position 140
-Shifting vs. Growing Tolerance Zones 142

8 Profile **146**
-Profile of a Surface 147
-New Symbol for Unequal or Unilateral Profile Tolerancing 151
-Profile of a Line 160
-The Power and Versatility of Profile (Mating Parts) 163

-Tolerancing Mating Part Profiles 163
-Composite Profile 168
-Composite vs. Two Single Segment Profile Controls 173
-Profiling Patterns of Features Using 3 Levels of Profile Tolerances 176
-Coplanarity 177
-Continuous Feature of Size Symbol 177
-Dimension Origin Symbol 180
-Locating Offset Surface with Profile of a Surface 184
-Conicity 187

9 Runout **191**
-Circular Runout 192
-Total Runout 195
-Comparison of Perpendicularity and Total Runout on a Planar Surface 203

10 Concentricity and Symmetry **207**
-Concentricity 208
-Comparison of Coaxiality Controls 210
-Symmetry 215

11 Datums **217**
-How They are Selected and What They Mean 219
-Specifying Degrees of Freedom 223
-Datum Feature Simulation 224
-Designating Degrees of Freedom on the Part Drawing 227
-Establishing a Valid Datum Plane 232
-Effects of Differing Datum Precedence on Part Acceptance 237
-Curved Surface as a Datum Feature 238
-Conical Datum Features 239
-Datum Feature Pattern Referenced Regardless of Material Boundary 240
-Inclined Datum Feature 241
-Constant Cross-Sections and Complex Datum Features 242
-Specifying Degrees of Freedom in the Feature Control Frame 243
-Multiple Datum Reference Frame Identification 245
-Correct Material Boundary Size Specified Next to the Datum Feature 246
-Correct Material Boundary Calculations 247
-Using the Translation Modifier 248
-Basic or BSC Spelled Out in a Feature Control Frame 250
-Planar Datum Feature Simulated at Regardless of Material Boundary (RMB) 252
-Planar Datum Feature Simulated at Maximum Material Boundary (MMB) 253
-Offset Datum Features of Size Simulated at RMB and MMB 255
-Profiled Datum Features Simulated at RMB and MMB 256
-Irregular Datum Features of Size 263

12 Centerplane Datums **264**
-An Overview 265

-Centerplane Datums on Mating Parts in a Fixed Fastener Assembly 267

13 Position with Fixed Fastener Assemblies and Projected Tolerance Zones 279
-Tolerancing Mating Parts in a Fixed Fastener Assembly 280
-Projected Tolerance Zones and How they are Measured 286
-Datum Feature Shift/Pattern Shift 289
-Alternate Method Using Chain Lines to Show Projected Tolerance Zones 292
-Calculating Clearance Hole Sizes Needed Without Projected Tolerance Zones 293

14 Tolerancing Mating Parts in a Floating Fastener Assembly 294
-Floating Fastener Assembly Condition (Radial Hole Patterns) 295
-Assigning Datum Features to Mating Parts with Radial Hole Patterns 296
-Calculating Position Tolerance 298
-Two Single Segment Position Tolerancing 300
-Calculating Minimum Wall Thicknesses 301
-Accumulative Datum Shift on Mating Parts in an Assembly 303
-Tolerance Zones and Their Movement with Two Single Segment Position 304

15 Direct vs. Indirect Relationships 305
-Overview 306
-Tolerancing Mating Parts Holding Function Directly and Indirectly 308
-Switching Datums in Mid-Stream 311
-Unique Effects of Utilizing the LMC and LMB Concepts 311
-Wall Thickness Calculations 314

16 Datum Targets 321
-Flexible Parts, Datum Targets and Partial Datum Features 322
-Sheet Metal Panels and GD&T Sheets 327
-Equalizing Datums 329
-Moveable Targets, Finding the Datum Planes and Fixturing 333
-Datum Target Symbols for Spherical Diameters 335
-Centerplane Datums 336
-Spherical Tolerance Zones 337

17 Datum Feature Scheme Choices 338
-Datum Feature Patterns and Profile 339
-Simultaneous Requirements 342
-Compound Datum Features of Size 345
-Secondary and Tertiary Datum Features of Size 347
-Finished Machining Requirements for a Cast Part 350

18 Flexible Parts 352
-Flexible Parts and Inspecting Them in the Way They Work 353
-Temporary Datum Features 354
-Common Misconceptions 355
-Free State Variation in Sheet Metal Parts 356

-Specifying Restrained State Inspection 358
-Fixturing Sheet Metal Parts 359
-Profile ALL OVER Controls and What They Mean 363

19 Position Boundary Concept 366
-Position Boundary 367
-Elongated Holes 367
-Functional Gages and Virtual Condition Boundaries 370
-Tolerancing Hoses, Pipes and Tubing with Positional Boundary 371
-Tolerancing Oddly Configured Features with Positional Boundary 373
-Oddly Configured Datum Features and How to Represent them in Gages 374
-Tolerance Zones vs. Boundary Concept Explanation 376
-Rectangular Tolerance Zones for Round Holes 378
-Bi-directional Position Tolerancing, Polar Coordinate Method 379

20 Why Use GD&T 381
-Multiple Interpretations of Simple Plus and Minus Tolerances 383
-Converting from Plus and Minus to Composite Position Tolerancing 386
-Calculating the Position Tolerance for a Composite Position Control 387
-Minimum Wall Thickness Calculation for Composite Position Tolerances 391
-Composite Tolerancing for Coaxial Hole Patterns 393
-Minimum Wall Thickness Calculations for Coaxial Hole Patterns 393
-Composite Position Tolerancing with 3 Levels of Control 396
-Differentiating Between Features of Similar Size and Shape 397

21 Composite vs. Two Single Segment Positional Tolerancing 398
-Composite vs. Two Single Segment Positional Tolerancing 399
-Similarities 401
-Differences 402
-One Level Tolerancing vs. Composite Tol. and Simultaneous Requirements 405
-Two Single Segment Position Controls 411
-Refining Geometric Controls to be More Cost Effective 414

22 Dimensioning and Tolerancing of Gages 423
-Dimensioning and Tolerancing of Gages per ASME Y14.43-2003 424
-GO Gages 424
-NOGO Gages 426
-Functional Gages 427
-Calculating to Determine Good Parts Rejected or Bad Parts Accepted 430
-Steps in the Development of a Dimensional Inspection Plan 436

23 Tolerance Stack-Up Analysis 442
-Tolerance Stack-Up Analysis for a Fixed Fastener Assembly 443
-Rules 444
-Calculating Gaps; Working the Route 445
-Calculating Inner and Outer Boundary Means and Their Tolerances 448

-Calculating Statistical Tolerancing 451
-Root Sum Squares 451
-Bender Factor 457
-Reintegrating the Statistical Tolerancing into the Assembly 458
-A Simpler Way to Reintegrate the Statistical Tolerance 461
-More Statistical Formulas and Symbols 462
-Glossary of Statistical Terms 464

24 How to be Specific in Calculating and Specifying Statistical Requirements for Size and Geometric Tolerancing 466
-Some Useful Definitions When Geometric Tolerances are Used 469
-Symbology for SPC Formulas 471
-Arithmetic Mean; Normal Distribution of Tolerance and the Standard Deviation; Statistical Probability for Tolerance Stack-Up Analysis for Positional Geometric Tolerances 474
-Calculating a Standard Deviation 476
-Predicting the Amount of Tolerance to be Consumed by Manufacturing 477
-Charts and Tables 478

25 Tolerance Stack-Up Analysis in a 5-Part Assembly 481
-Determining a MIN GAP in a Rotating Assembly 482
-Factors vs. Non-factors 483
-Alignment 485
-Dealing with Threaded Features 486
-Calculating the Pertinent Numbers 490
-Simplifying the Assembly Drawing 491
-Creating a Line Graph with Numbers to Calculate the Minimum Clearance 492
-Adding the Negative and Positive Designations 492
-Wall Thickness Calculations and Choosing the Pertinent Tolerances 493
-Single Part Analysis 496
-Using Profile Tol. and Separate Requirements for Accumulated Error 499

26 Tolerance Stack-Up Created during Manufacture due to Changing Set Ups 502
-Where the Tolerance Accumulation Comes From 503
-Proportions and Trigonometry 504

27 GD&T as a Language 507
-To Properly Read a Drawing 508
-Reading the Feature Control Frames as Sentences 512
-Profile 513
-Tolerance Zones and Pattern Shift Zones 513
-Reading Two Single Segment Controls 514
-Using Gages to Visualize a Geometric Tolerance's Meaning 517
-Reading a GD&T Sheet 526
-Optional Tolerancing Approaches for Similar Results 529
-Gears 530

-Pattern Shift, Where it Comes From and How it Effects the Workpiece 532
-Bonus Tolerance, Virtual Condition and Zero Positional Tolerances 535
-Threads, Gears and Splines 536
-Sequential Tolerancing Using the Simultaneous Requirement Rule 537

28 Definitions **539**

Index **568**

Bibliography **573**

Other GD&T Course Materials Written by James D. Meadows **574**