

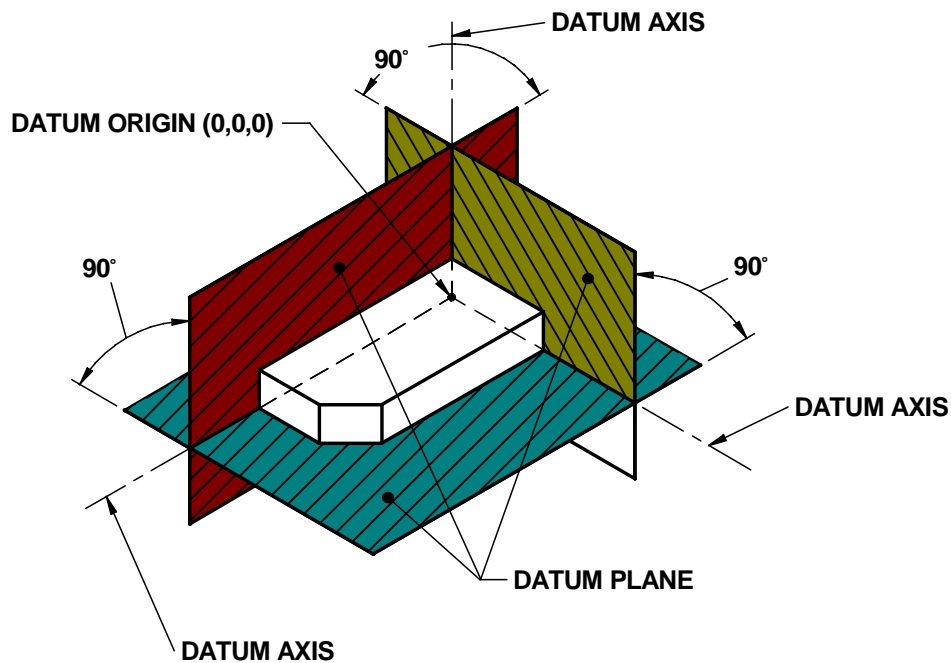
Geometrical Boundaries

*Interpretation and Application
of Geometrical Product Specifications (GPS)
(Using SI Units)
Based on ISO 1101:2004*

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 Geometrical tolerancing.

I have deliberately focused my efforts for technical professionals applying geometrical 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 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 B

Acknowledgments

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

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 Requirement
ISO 2692:1998/DAM 1 - Technical Drawings Least Material Requirement
ISO 3040 - Technical Drawings Cones
ISO 5458 - Technical Drawings Positional Tolerancing
ISO 5959 - Technical Drawings Datum's and Datum Systems
ISO 7083 - Technical Drawings Symbols Proportions
ISO 8015:1985 - Technical Drawings Fundamental Tolerance Principle
ISO 10578:1992 – Technical Drawings - Tolerancing of orientation and location –
Projected tolerance zone
ISO 10579:1993 - Technical Drawings Non-Rigid Parts
ISO 10587 - Technical Drawings Projected Tolerance Zones
ISO/TS12180-1:2003, Cylindricity – Part 2

ASME Y14.5.M-1994 - Dimensioning and Tolerancing.
Engineers Edge 2000 - 2008, Solutions by Design, Kelly Bramble
Engineering Design for Manufacturing 2006 - 2008, Kelly Bramble

Table of Contents

- 1.2 Preface
- 1.3 Acknowledgments
- 1.4 Table of Contents
- 1.9 Introduction
- 1.10 How the Geometrical Dimensioning and Tolerancing System Works
- 1.11 Tolerances, Features and Characteristics
- 1.12 Tolerance Frame
- 1.13 Tolerance Frame Association and Application
- 1.14 Common Symbols ISO 1101:2004
- 1.15 Position and Limit Tolerance General Overview and Contrast

- 2.1 **Limit Tolerancing**
- 2.1 Limit Tolerancing
- 2.2 Dimensioning System –Limit Tolerancing
- 2.3 Limit Tolerance, Square - Round Tolerance Zone Contrast and Advantage
- 2.4 Implied 90 Degree Angle
- 2.5 Dimension Origin
- 2.6 Gears, Splines and Screw Threads
- 2.7 Feature Definition – Size & Without Size
- 2.8 Materials Requirements, MMR, LMR
- 2.9 Principal of Independency
- 2.10 Envelope Requirement

- 3.1 **Datums**
- 3.2 Datum coordinate system (DCS) overview
- 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 Tolerance Frame
- 3.7 Simulated datum, and Theoretical Datum Plane
- 3.8 Primary External Datum Diameter
- 3.9 Primary Internal Datum Diameter
- 3.10 Primary External Datum Width
- 3.11 Primary Internal Datum Width
- 3.12 Setup and Inspection of Datum's
- 3.14 Sequence of datum features
- 3.15 Sequence of datum features relates part to datum coordinate system
- 3.16 Cylindrical datum feature
- 3.17 Parts with angular orientation
- 3.18 Development of a Datum Coordinate System
- 3.19 Orientation of Two Datum Planes Through a Hole
- 3.20 Partial Datum Surfaces as Datum Features
- 3.22 Multiple Datum features, Single Datum
- 3.23 Inclined Datum Features

Table of Contents

4.1	Datum Targets	5.8	Straightness
4.2	Datum Targets Overview	5.9	Straightness Per Unit Basis
4.3	Datum Target Point Symbol, Application	5.10	Straightness Tolerance Applied in Two Directions
4.4	Datum Target Area	5.11	Straightness of a Surface (Cylindrical)
4.5	Datum Target Line	5.12	Straightness of a Feature of Size at RFS
4.6	Dimensioning Datum Targets	5.13	Straightness of a Feature of Size at MMR
4.7	Primary Datum Plane Established by Three Datum Target Areas	5.14	Straightness Per Unit Length With Specified Total Straightness
4.8	Primary Datum Plane Established by Two Datum Target Points and One Datum Target Line.	5.15	Cylindricity
4.9	Datum Step Feature	5.16	Roundness
4.10	Datum Target Lines and Areas		
4.11	Primary Datum Axis Established by Datum Target points on a Single Cylindrical Feature	6.1	Orientation
4.12	Equalizing Datum	6.2	General
4.14	Secondary Datum Axis	6.3	Perpendicularity – Surface
5.1	Form	6.4	Perpendicularity Surface Multiple Datum's
5.2	General	6.5	Perpendicularity – Center plane
5.3	Flatness	6.6	Perpendicularity at MMR Internal Feature – Center Plane
5.4	Flatness Applied to Multiple Unique Surfaces With the Same Value	6.7	Perpendicularity – External feature of Size Axis
5.5	Flatness Applied to Multiple Unique Surfaces as a Common Tolerance Zone	6.8	Perpendicularity – Internal feature of size axis
5.6	Flatness Applied on a Unit Basis	6.9	Perpendicularity – Threaded Hole or Inserts Projected Tolerance Zone
5.7	Flatness Applied on Unit Basis with Overall Control	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

Table of Contents

Tolerances of Location			
7.3	General	7.21	Positional Tolerance at MMR Relative to Hole and Slot Datum Features
7.4	Fundamental Explanation of Positional Tolerancing Feature of Size	7.22	Position Tolerance Applied to a Flat Surface
7.6	Definitions and Modifiers	7.23	Bi-Directional Positional Tolerancing, Polar Coordinate Method
7.7	Maximum Material Requirement	7.24	Different Positional Tolerance at Each Surface
7.8	Least Material Requirement LMR	7.25	Coaxial (Concentric) Control of Cylinders
7.9	External Feature of Size Position Tolerance Boundaries with MMR Specification	7.26	Coaxial (Concentric) Control of Multiple Hole-Counterbore Holes
7.10	Internal Feature of Size Position Tolerance Boundaries with MMR Specification	7.27	Hole Pattern Located Perpendicular to Cylindrical Datum
7.11	External Feature of Size Position Tolerance Boundaries with LMR Specification	7.28	Hole Pattern Located Perpendicular to Cylindrical Datum
7.12	Internal Feature of Size Position Tolerance Boundaries with LMR Specification	7.29	Holes Not Normal to DCS
7.13	Zero positional tolerance at MMR	7.30	Hole Pattern Located at Angle to Datum Coordinate System
7.14	Position Tolerance at RFS	7.31	Positional Tolerance at MMR of Spherical Feature
7.15	Positional Tolerance Axis and Surface Interpretation – Surface Datum’s	7.32	Positional Tolerance of Coaxial Holes of Same Size
7.16	Positional Tolerance Axis Interpretation – Surface Datum’s	7.33	Least Material Requirement Application – Cylinder Minimum Wall Thickness
7.17	Positional Tolerance Surface Interpretation - Surface Datum’s	7.34	Positional Tolerance for Coaxiality With Datum Feature Referenced at MMR
7.18	Positional Tolerance Axis and Surface Interpretation - Thru Hole Datum’s	7.35	Positional Tolerance for Coaxially with Feature Referenced at Zero MMR Relative to Datum Feature at MMR
7.19	Positional Tolerance Axis Interpretation – Thru Hole Datum’s	7.36	Positional Tolerance - Thru Hole Datum’s at MMR
7.20	Positional Tolerance Surface Interpretation - Thru Hole Datum’s		

Table of Contents

7.37	Positional Tolerance - Thru Hole Datum Verification	9.1	Runout
7.38	Composite Positional Tolerancing	9.2	General
7.41	Coaxiality Tolerancing of an Axis	9.3	Circular Run-out
7.42	Concentricity Tolerance of Spherical Feature	9.4	Circular Run-out With Plane as Datum
7.48	Symmetry	9.4	Total Run-out
	Profile	10.1	Coaxial Tolerance Comparison
8.2	General	11.1	Tolerance Analysis
8.3	Profile any Surface, Bilateral Tolerance	11.2	Series Stack Calculations
8.4	Profile any Surface, Bilateral Tolerance Rectangular Coordinate Dimensioning Without Dimension Lines	11.3	Floating Fastener Condition
8.5	Profile any Surface, unilateral (Inside) Tolerance	11.4	Fixed Fastener Condition
8.6	Profile any Surface, Unilateral (Outside) Tolerance	11.5	Tolerance Compensation for Projected Tolerance Zone – Fixed Fastener condition
8.7	Profile any Surface, bilateral unequal tolerance	11.7	Two Mating Diameters Positional Calculation
8.8	Profile any Surface, All Around	11.8	Three Mating Diameters Positional Calculation
8.9	Profile any Surface, Independent Form Control	11.9	Position Tolerance Verification and Hole Pattern Analysis
8.10	Profile any Surface Tolerance Independent Form Control Not Related to Datum	11.11	Position Coordinate to Location Conversion Chart
8.11	Profile Tolerance for Coplanar Surfaces	11.12	Generic Hole Verification Chart
8.12	Profile any Line	12.1	Symbol Comparison
8.12	Profile any Line Without Datum Reference	12.2	Comparison of ASME and ISO Symbols (geometrical Characteristics)
8.13	Profile any Line Tapered Shape	12.3	Comparison of ASME and ISO Symbols (General)
8.14	Profile any Line Without Datum Reference		
8.15	Composite Profile Tolerance		
8.16	Composite Profile Application		

This Page Left Deliberately Blank

Introduction

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

The latest international standard on the subject of G&T defined and in practice is the International Standards Organization ISO 1101:2004, Geometrical Product Specifications (GPS)

Declarations:

All illustration and drawings are depicted using third angle projection drafting practices, see Figure .1. Please note that first angle projection could have been used equally well without prejudice to the principles established.

Except where noted, all dimensional data is given in SI units (mm).

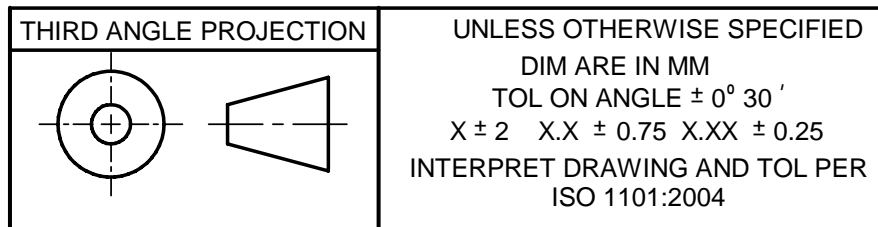


Figure .1

DIMENSIONING AND TOLERANCING

How the Geometrical 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. Geometrical Tolerancing (G&T) standard ISO 1101:2004 is a defined system of rules, symbols, and explicit requirements to fully delineate an objects geometrical requirements.

The following are the more common reasons to apply G&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 geometrical symbol.

Geometrical Characteristics and Symbols

Geometrical characteristic symbols are used to define simple or complex feature requirement or relationship. G&T characteristics and categories are:

	TOLERANCE TYPE	CHACTERISTIC	SYMBOL
FOR INDIVIDUAL FEATURES	FORM	FLATNESS	
		STRAIGHTNESS	
		CYLINDRICITY	
		ROUNDNESS	
FOR INDIVIDUAL OR RELATED FEATURES	PROFILE	PROFILE OF A SURFACE	
		PROFILE OF A LINE	
FOR RELATED FEATURES	ORIENTATION	PERPENDICULARITY	
		PARALLELISM	
		ANGULARITY	
	LOCATION	POSITION	
		COAXIALITY (for axis)	
		CONCENTRICITY (for center points)	
		SYMMETRY	
	RUNOUT	TOTAL RUN-OUT	
		CIRCULAR RUN-OUT	

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