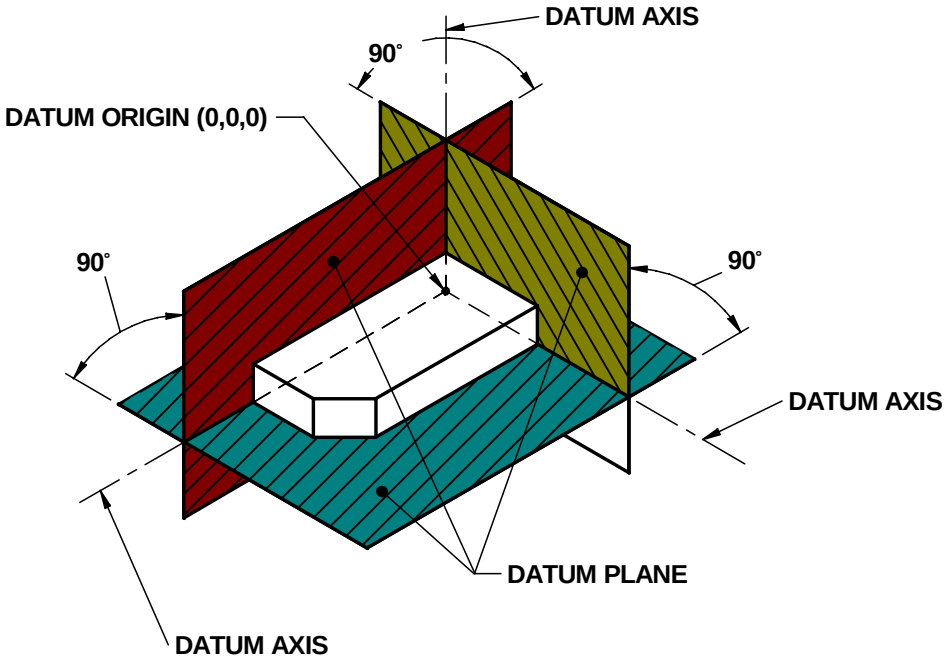
Geometrical Boundaries

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



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

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

Table of Contents

Table of Contents

Table of Contents

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).

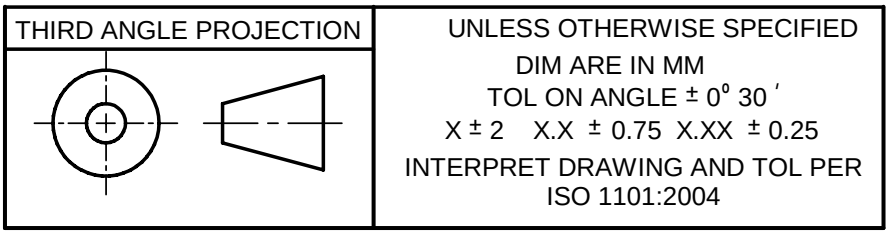


Figure .1

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.