Measurement and Tolerances

Alessandro Anzalone, Ph.D.

Hillsborough Community College, Brandon Campus

Measurement and Tolerances

Sections:

1. Meaning of Tolerance
2. Geometric Dimensioning and Tolerancing
3. Application of Geometric Tolerancing
4. Summary

Tolerance

Tolerance is the total amount that a specific dimension is permitted to vary. In dimensional metrology, tolerances are applied to both position (where) and size (how big) dimensions. Both types of dimensions must have tolerances for economical manufacture.

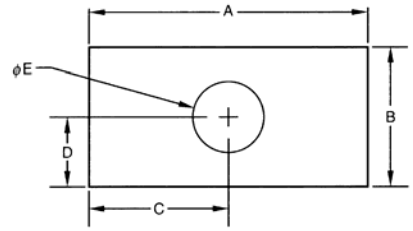


FIGURE 3–1 Tolerances apply to both dimensions of size (A, B, and E) and dimensions of location (C and D).

Geometric Dimensioning and Tolerancing

GD&T is a means of dimensioning and tolerancing a drawing with respect to the actual function or relationship of part features that can be most economically produced. The key words here are function and relationship. This type of dimensioning and tolerancing should be used when:

1. Features are critical to the functionality or interchange ability of the part.
2. Datum references are desirable to ensure consistency between design, manufacturing, and inspection.
3. Computerization techniques in design and manufacturing are being used or are desirable.
4. Standard interpretation or tolerance is not already implied.

Benefits of Geometric Dimensioning and Tolerancing

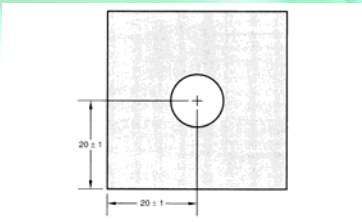


FIGURE 3–2 A square block with a hole drilled in the center, using conventional tolerancing.

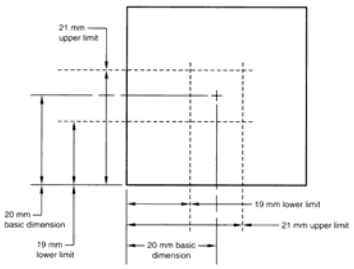


FIGURE 3–3 The conventional method prescribes a square (or rectangular) tolerance zone.

Benefits of Geometric Dimensioning and Tolerancing

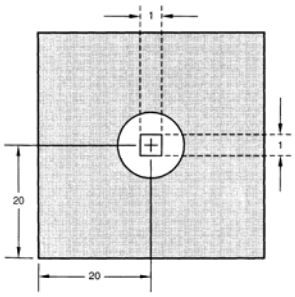


FIGURE 3–4 Coordinate location tolerance zone. The shaded area shows where the location of the hole could vary.

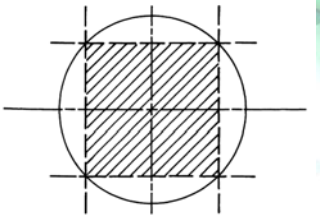


FIGURE 3–5 The conventional method requires that the drill center be within the shaded area, yet the additional spaces prescribed by the circle are also within the true tolerance required by the desired fit but not allowed.

Benefits of Geometric Dimensioning and Tolerancing

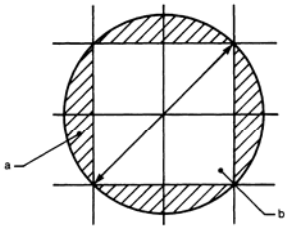


FIGURE 3–6 Point b is technically within true tolerance, whereas a is out of tolerance. Obviously, a will produce equally acceptable parts.

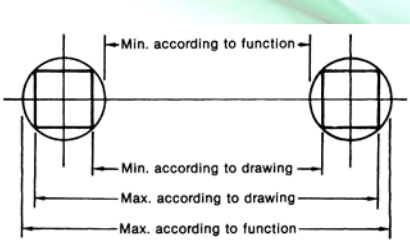


FIGURE 3–7 Geometric dimensioning and tolerancing allows us to reduce costs by not scrapping "good parts."

Benefits of Geometric Dimensioning and Tolerancing

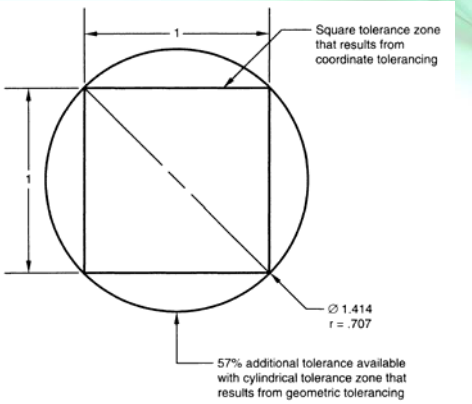


FIGURE 3–8 Area of a circle = πr^2 = 3.1416* (.707)^2 = 1.57; area of a square = s^2 = 1.0^2 = 1.0. Geometric tolerancing allows a diametrical (circular) tolerance zone, resulting in a 57% increase in the allowable tolerance.

Definitions

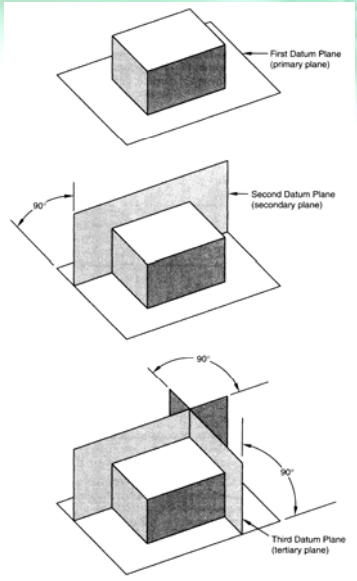
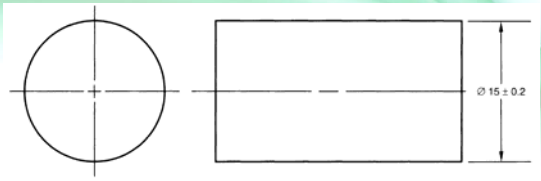
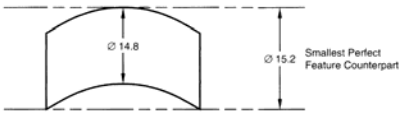


FIGURE 3–9 Datum reference frame

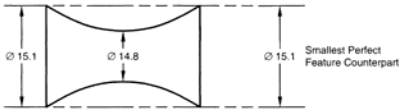
Definitions



a. Engineering drawing of a pin with a dimension of 15 ± 0.2



b. Actual mating envelope = Ø 15.2 for this part



c. Actual mating envelope = Ø 15.1 for this part

FIGURE 3–10 Actual mating envelope of an external feature of size

Definitions

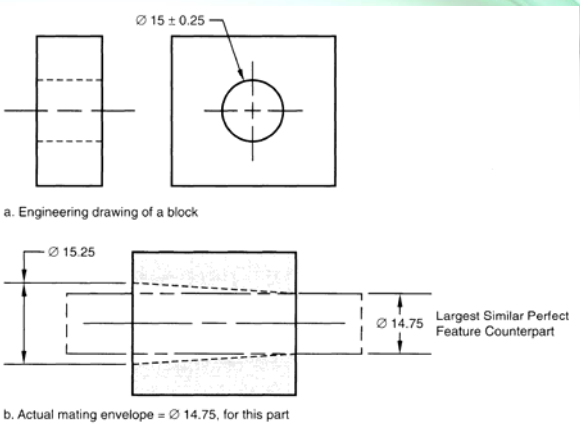


FIGURE 3–11 Actual mating envelope of an internal feature of size

Symbols and Modifiers

The language of geometric tolerancing is a set of symbols. These symbols are divided into five types of dimensioning control: form, profile, orientation, location, and runout.

1. **Form tolerance**. States how far an actual surface or feature is permitted to vary from the desired form implied by the drawing.
2. **Profile tolerance**. States how far an actual surface or feature is permitted to vary from the desired form on the drawing and/or vary relative to a datum.
3. **Orientation tolerance**. States how far an actual surface or feature is permitted to vary relative to a datum.
4. **Location tolerance**. States how far an actual size feature is permitted to vary from the perfect location implied by the drawing as related to a datum or other feature.
5. **Runout tolerance**. States how far an actual surface or feature is permitted to vary from the desired form implied by the drawing during full 360° rotation of the part on a datum axis.

Symbols and Modifiers

Geometric Dimensioning and Tolerancing Chart
(per ASME Y14.5-1994)

	Types of Tolerance	Symbol	Characteristic	Controls: Axis	Controls: Surface	Feature Modifiers	Uses a Datum Reference	Datum Modifiers	Notes
For Individual Features	Form	—	Straightness (of an axis)	X		Yes	NEVER	N/A	Rule #1 Overridden
		—	Straightness (of a surface element)		X	No		N/A	Rule #1 applies
		▱	Flatness		X	No		N/A	Tolerance must be less than size
		○	Circularity		X	No		N/A	Tolerance must be less than size
		⌭	Cylindricity		X	No		N/A	Tolerance must be less than size
For Individual or Related Features	Profile	⌒	Profile of a Line		X	No	SOMETIMES	Yes	Rule #1 applies
		⌓	Profile of a Surface		X	No		Yes	Rule #1 applies
For Related Features	Orientation	∠	Angularity	X	X	Yes	ALWAYS	Yes	Also controls flatness of surface
		⊥	Perpendicularity	X	X	Yes		Yes	Feature modifiers OK if diameter is used
		//	Parallelism	X	X	Yes		Yes	Also controls flatness of surface
	Location	⌖	Position	X	See Notes	Yes		Yes	Can control surface boundary
		◎	Concentricity	See Notes		No		No	Controls opposing median points
		⌯	Symmetry	See Notes		No		No	Controls opposing median points
	Runout	↗	Circular Runout		X	No		No	Rule #1 applies
		⌰	Total Runout		X	No		No	Rule #1 applies

Rule #1 summarized—Tolerance limits control the shape of an individual feature as well as the size. (See ASME Y14.5M-1994, section 2.7.1)

FIGURE 3–12 GD&T symbols

Material Conditions

MODIFIER	ABBREVIATION	SYMBOL
Maximum Material Condition	MMC	Ⓜ
Least Material Condition	LMC	Ⓛ
Regardless of Feature Size	RFS	None

FIGURE 3–13 Material condition modifiers

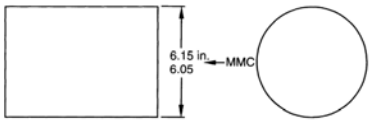


FIGURE 3–14 Maximum material condition (MMC) of an external feature like this pin equals the largest allowable diameter of 6.15 in.

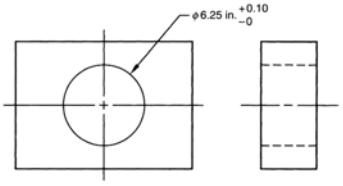


FIGURE 3–15 A hole at maximum material condition is the smallest allowable size, retaining the "most material." Here, the maximum material condition is 6.25 in.

Rules

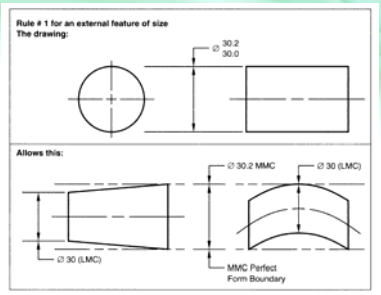


FIGURE 3–16 Rule #1—Variations in form as well as size are allowed.

Feature Control Frames

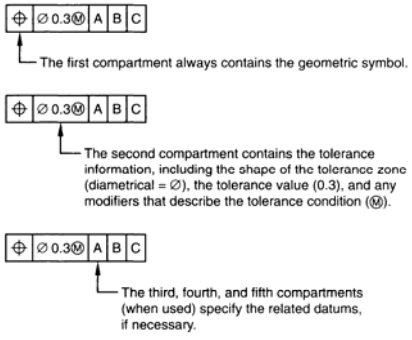


FIGURE 3–17 Feature control frames

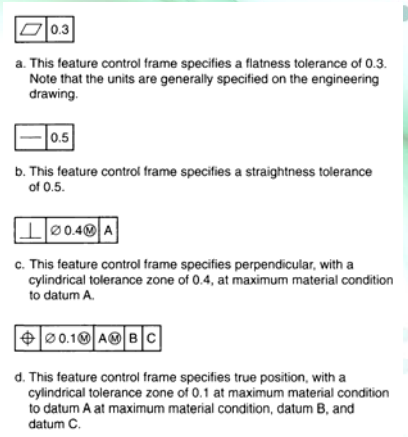
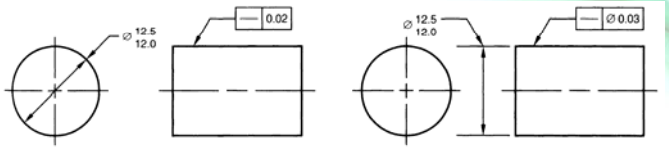


FIGURE 3–18 Examples of feature control frames

Application of Geometric Tolerancing



Straightness tolerance applied to a surface. Straightness tolerance applied to an axis.

Types of Tolerance	Symbol	Characteristic	Controls		Feature Modifiers	Uses a Datum Reference	Datum Modifiers	Notes
			Axis	Surface				
Form	—	Straightness (of an axis)	X		Yes	NEVER	N/A	Rule # 1 Overridden
	—	Straightness (of a surface element)		X	No		N/A	Rule # 1 applies

Note: The part must also be within the size limits.

FIGURE 3–19 Straightness tolerance

Application of Geometric Tolerancing

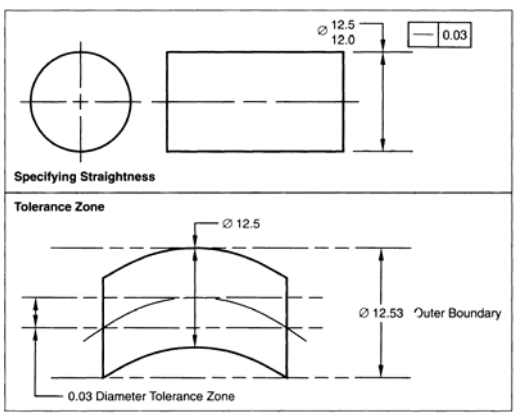


FIGURE 3–20 Straightness tolerance applied to an axis

Application of Geometric Tolerancing

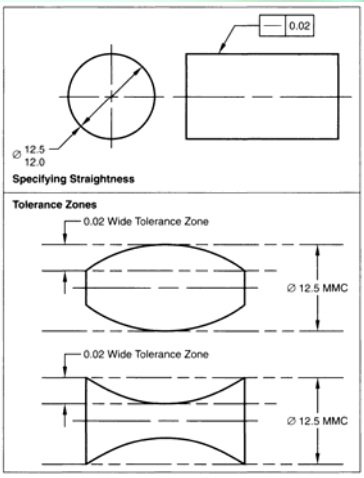


FIGURE 3–21 Straightness tolerance applied to a surface

Application of Geometric Tolerancing

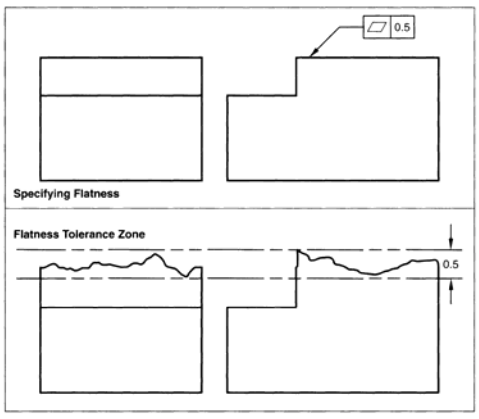
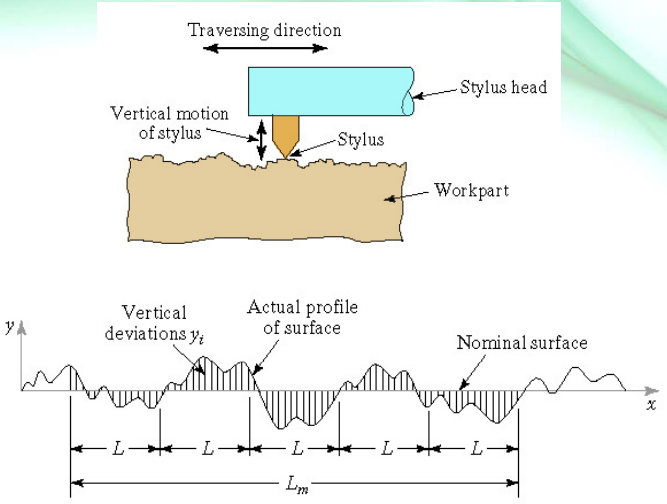


FIGURE 3–22 Flatness tolerance

Application of Geometric Tolerancing



Application of Geometric Tolerancing

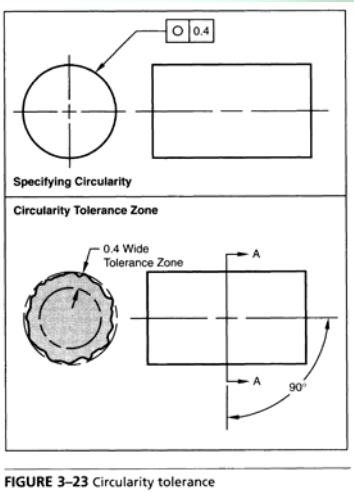


FIGURE 3–23 Circularity tolerance

Application of Geometric Tolerancing

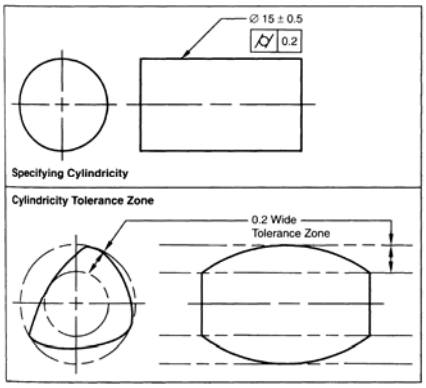


FIGURE 3–24 Cylindricity tolerance

Application of Geometric Tolerancing

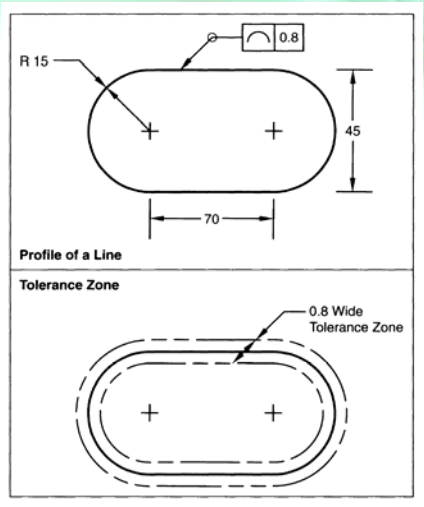
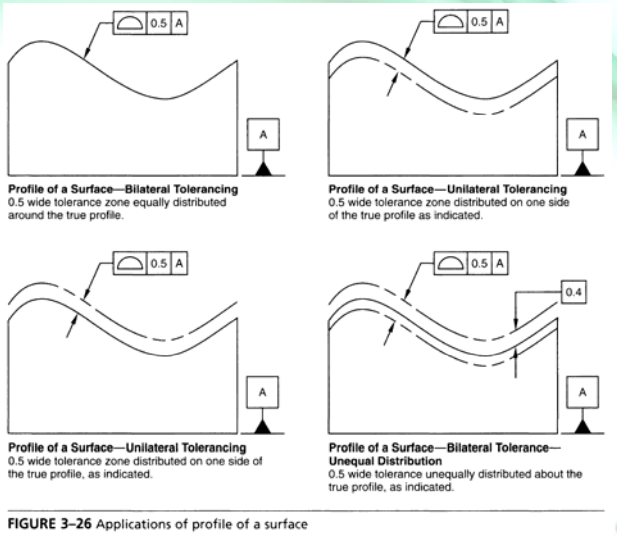


FIGURE 3–25 Profile of a line with all-around symbol

Application of Geometric Tolerancing



Profile of a Surface—Bilateral Tolerancing
0.5 wide tolerance zone equally distributed around the true profile.

Profile of a Surface—Unilateral Tolerancing
0.5 wide tolerance zone distributed on one side of the true profile as indicated.

Profile of a Surface—Unilateral Tolerancing
0.5 wide tolerance zone distributed on one side of the true profile, as indicated.

Profile of a Surface—Bilateral Tolerance—Unequal Distribution
0.5 wide tolerance unequally distributed about the true profile, as indicated.

FIGURE 3–26 Applications of profile of a surface

Application of Geometric Tolerancing

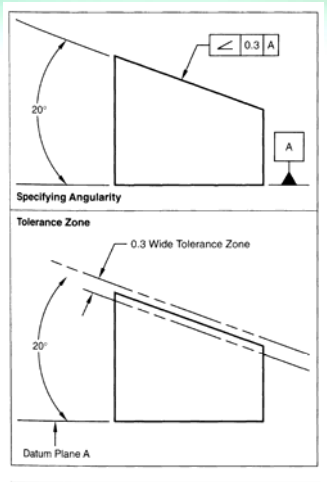


FIGURE 3–27 Angularity tolerance applied to a surface

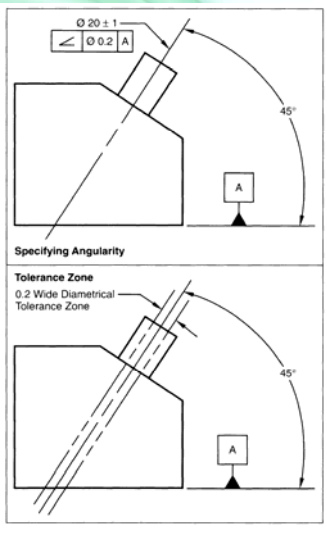


FIGURE 3–28 Angularity tolerance applied to an axis

Application of Geometric Tolerancing

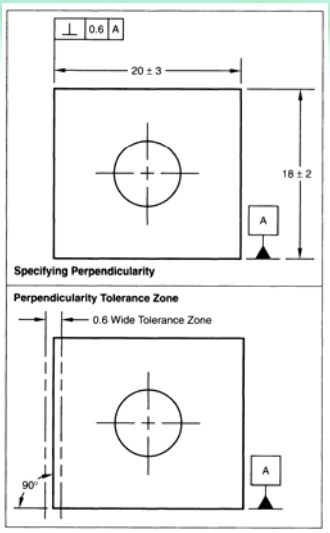


FIGURE 3–29 Perpendicularity tolerance applied to a surface

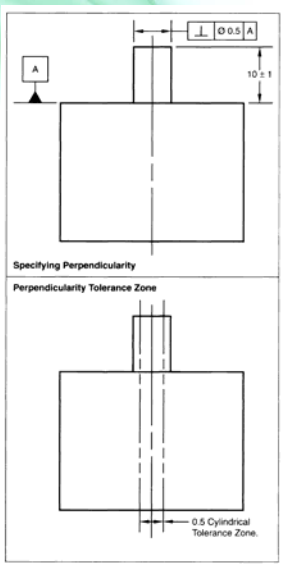


FIGURE 3–30 Perpendicularity tolerance applied to an axis

Application of Geometric Tolerancing

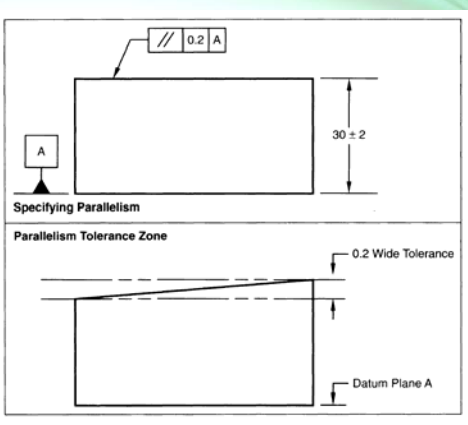


FIGURE 3–31 Parallelism tolerance applied to a surface

Application of Geometric Tolerancing

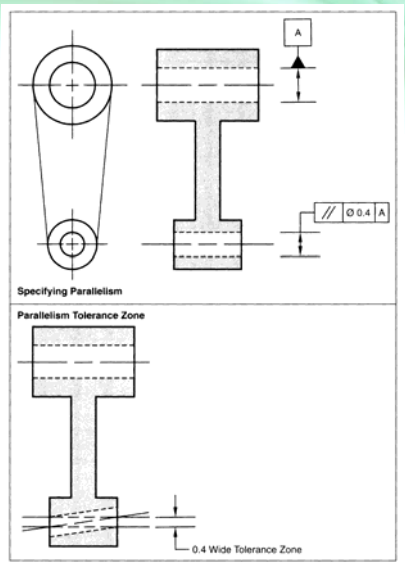


FIGURE 3–32 Parallelism tolerance applied to an axis

Application of Geometric Tolerancing

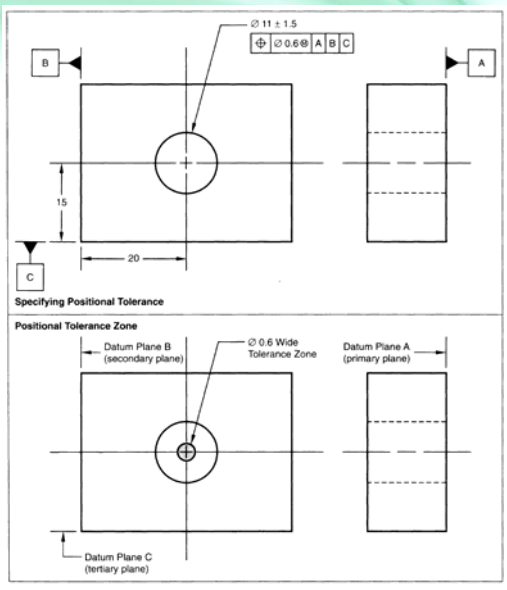
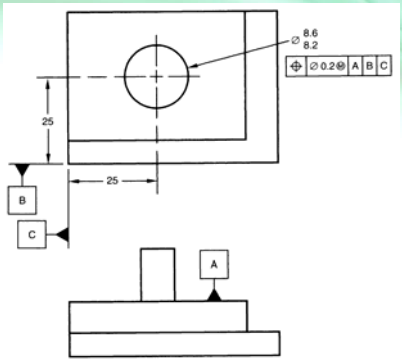


FIGURE 3–33 Position tolerance

Application of Geometric Tolerancing

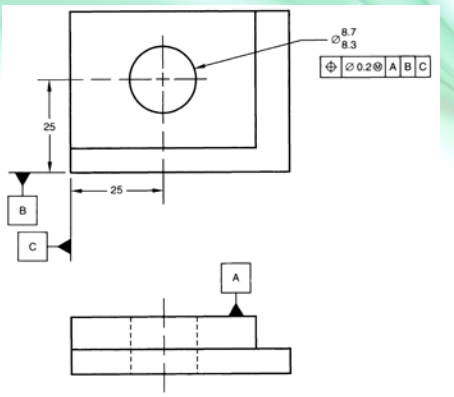


	Manufactured Size	Positional Tolerance	Bonus Tolerance (MMC-Actual Size)	Positional Tolerance Allowed
MMC	8.6	0.2	0	0.2
	8.5	0.2	0.1	0.3
	8.4	0.2	0.2	0.4
	8.3	0.2	0.3	0.5
LMC	8.2	0.2	0.4	0.6

The total positional tolerance is equal to the positional tolerance plus the bonus tolerance.

FIGURE 3–34 Positional tolerance at MMC for an external feature of size

Application of Geometric Tolerancing



	Manufactured Size	Positional Tolerance	Bonus Tolerance (Actual Size-MMC)	Positional Tolerance Allowed
MMC	8.3	0.2	0	0.2
	8.4	0.2	0.1	0.3
	8.5	0.2	0.2	0.4
	8.6	0.2	0.3	0.5
LMC	8.7	0.2	0.4	0.6

The total positional tolerance is equal to the positional tolerance plus the bonus tolerance.

FIGURE 3–35 Positional tolerance at MMC for an internal feature of size

Application of Geometric Tolerancing

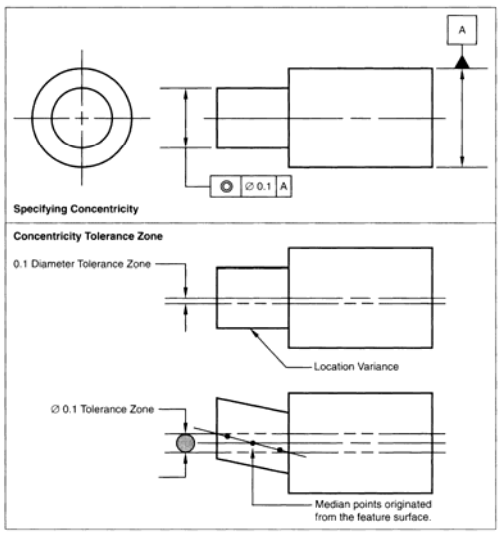


FIGURE 3–36 Concentricity tolerance. Concentricity is defined as the condition where all median points of diametrically opposed elements of a cylinder are congruent with the axis of a datum feature.

Application of Geometric Tolerancing

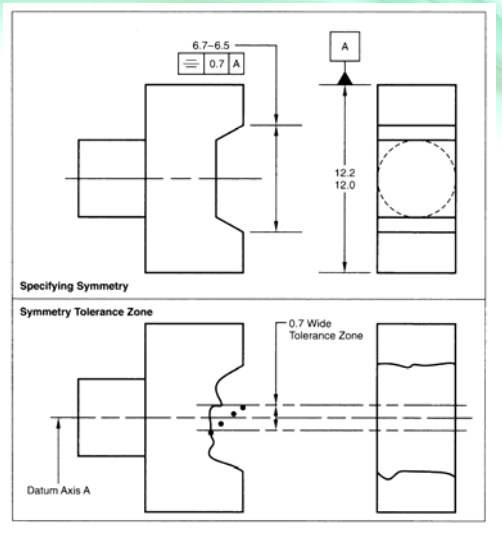


FIGURE 3–37 Symmetry tolerance

Application of Geometric Tolerancing

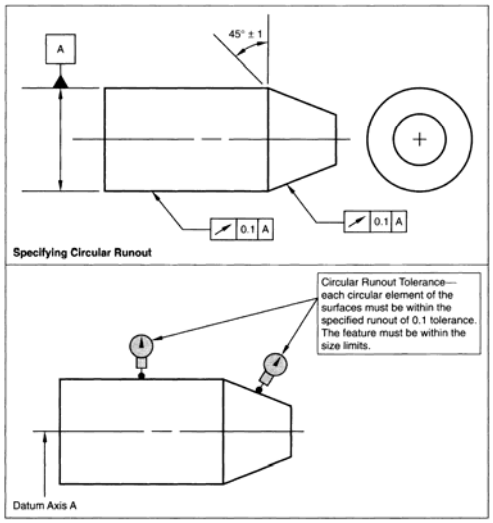


FIGURE 3–38 Circular runout

Application of Geometric Tolerancing

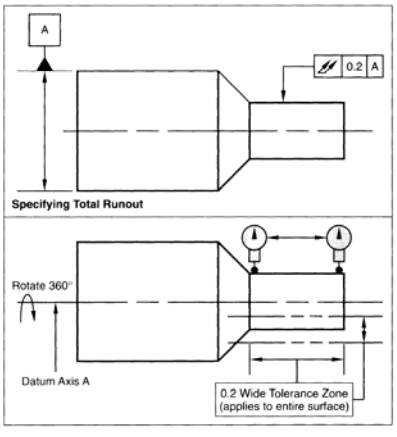


FIGURE 3–39 Total runout

Summary

- GD&T is a universal language of symbols used to convey design intent from the design stage, through manufacturing, to inspection of the final product.
- Machine drawings were conventionally dimensioned by Cartesian coordinates—vertical and horizontal dimensions. Cartesian coordinates prescribe a square tolerance zone. Geometric tolerancing allows a diametrical or cylindrical tolerance zone, increasing the size of the tolerance zone by approximately 57%.
- Geometric tolerancing provides manufacturing with increased tolerancing for mating parts, reducing scrap and lowering manufacturing costs.
- While this chapter provides a brief summary of GD&T concepts, the ASME Y14.5M-1994 Dimensioning and Tolerancing Standard should be made available to designers and inspectors for a complete set of definitions, fundamental rules, and practices. The standard establishes uniform practices for stating and interpreting engineering drawings.

References

http://www.thayer.dartmouth.edu/mshop/pdf/introdr.pdf
http://www.etinews.com/gdt_glossary.html