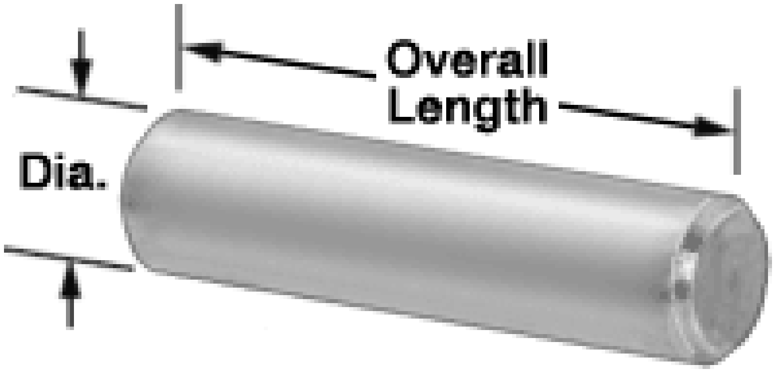
WHAT IS GEOMETRIC DIMENSIONING & TOLERANCING?

or

Help! My parts don’t fit

I Need a Part !

Example of a Simple Pin

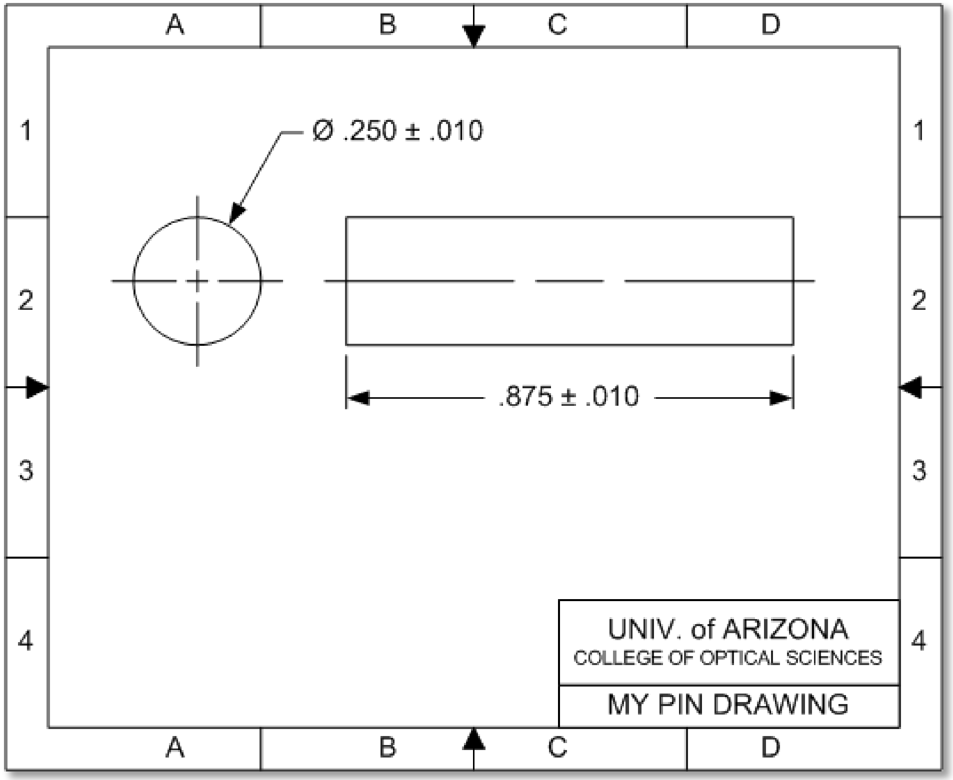


Pin must fit into a hole with a
Ø.260 inch mating envelope

The Produce Development Cycle
(In a Nutshell)

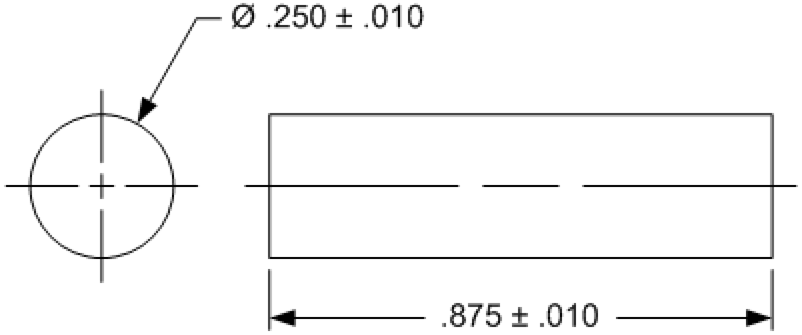
- Identify Need
- Requirements
- Design
- Manufacture
- Verification
- Need Satisfied ($$)

The Design

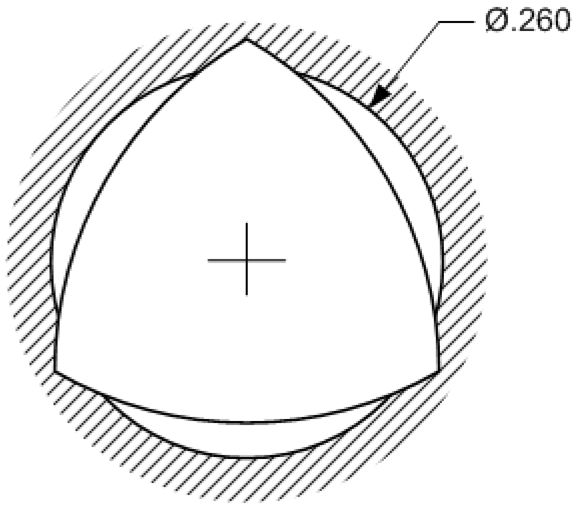

A
B
C
D
1
2
3
4
Ø .250 ± .010
.875 ± .010
UNIV. of ARIZONA
COLLEGE OF OPTICAL SCIENCES
MY PIN DRAWING

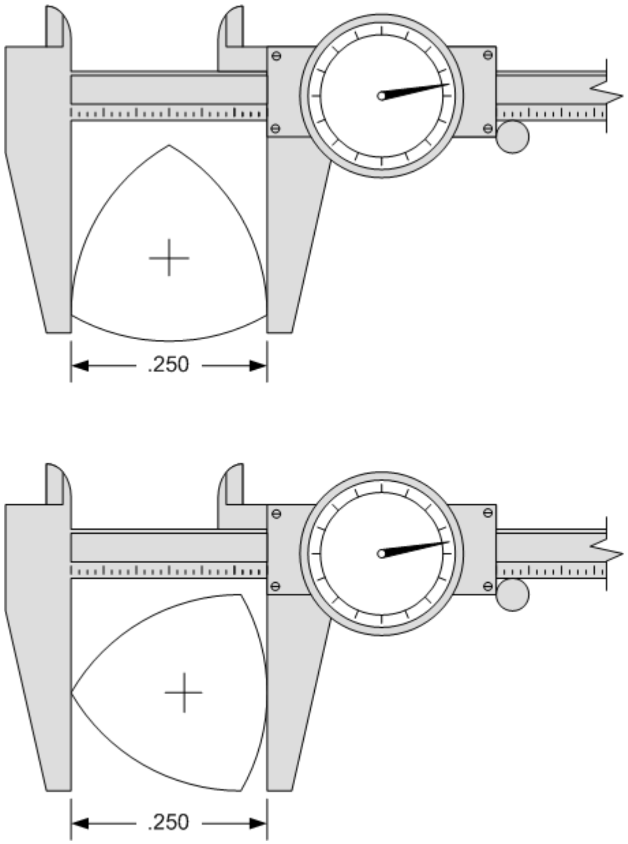
What Can Go Wrong?

- Limit dimensions were used to define part
  - Controls diametrically opposed elements only
- Pin can be lobed and still meet this requirement
- There is no form control for “roundness”
- There is also no form control for “straightness”



Help! My Part Doesn't Fit!

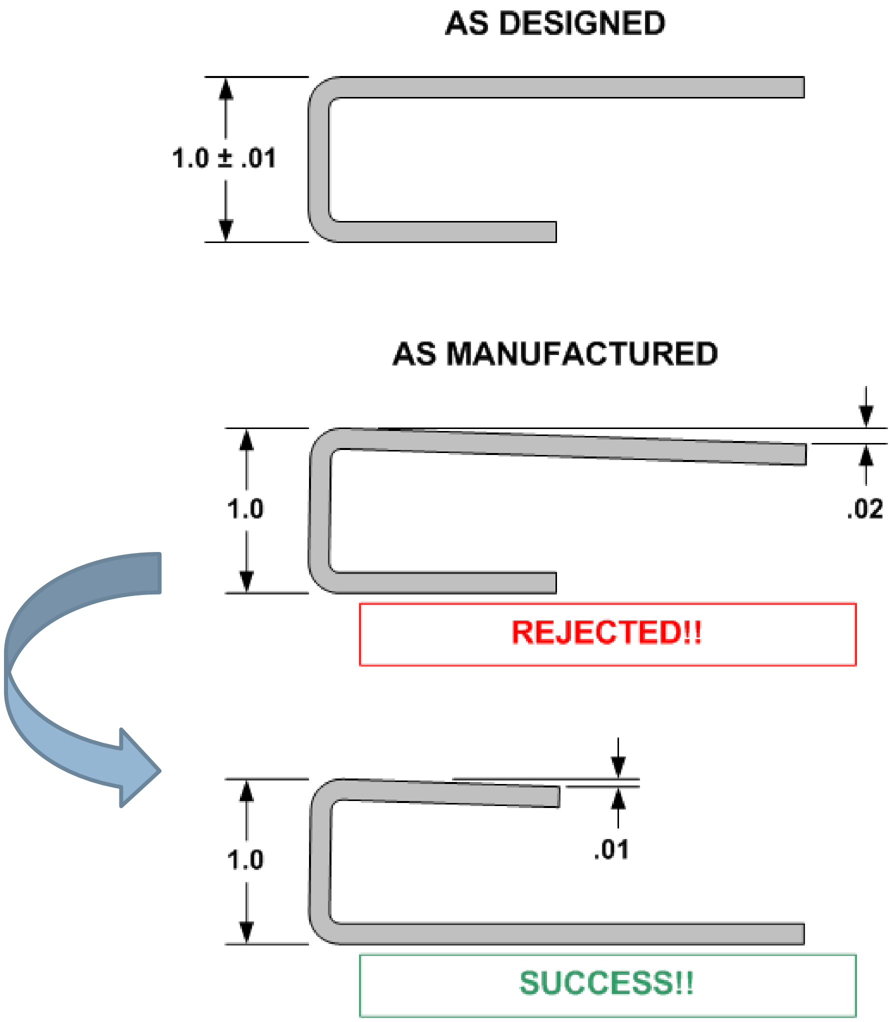




NO FORM CONTROL!

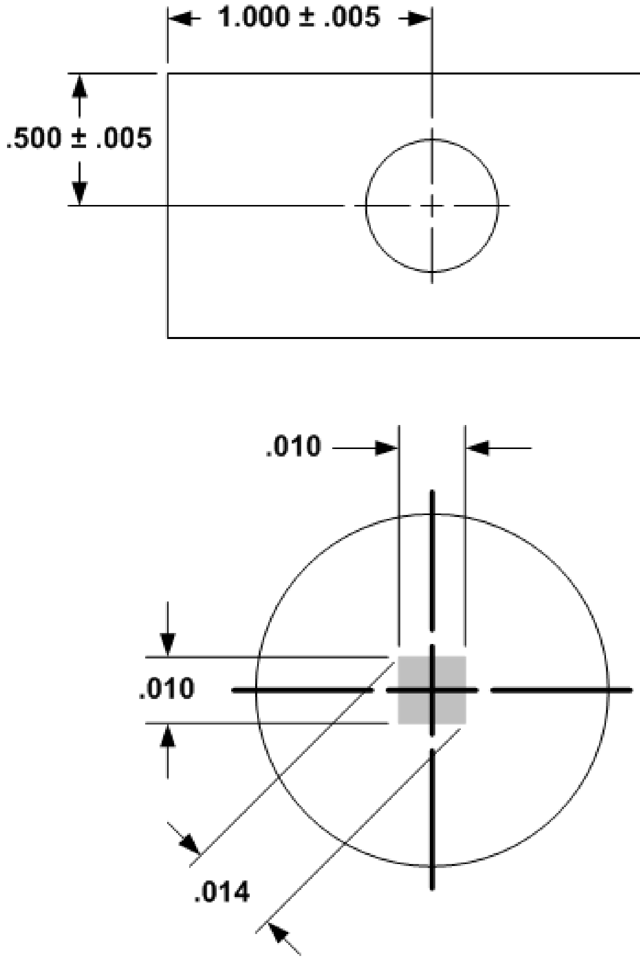
Other Limitations of Limit Dimensioning

- Inconsistent inspection results
- Dependent on orientation
- Relies on implied origins
- Not cool!



Other Limitations of Limit Dimensioning (con't)

- Overly Restrictive
- If requirement is axial alignment within .014
- Plus-minus tolerance must be ±.005
- Results in a square tolerance zone



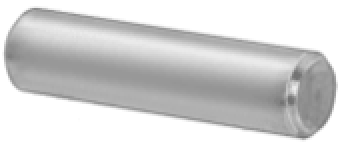
How Do I Get What I Want?

Using Limit Dimensions

- Build the part yourself
  - Use the mating part as a gage
  - Limited only by your machining skills and equipment
- Work closely with the manufacturer
  - Provide access to your setup
  - Communication is key
  - Limited by location and accessibility (are they on the same continent and speak English?)
- Find an alternate way to describe your part

Chances for Success

Using Limit Dimensions



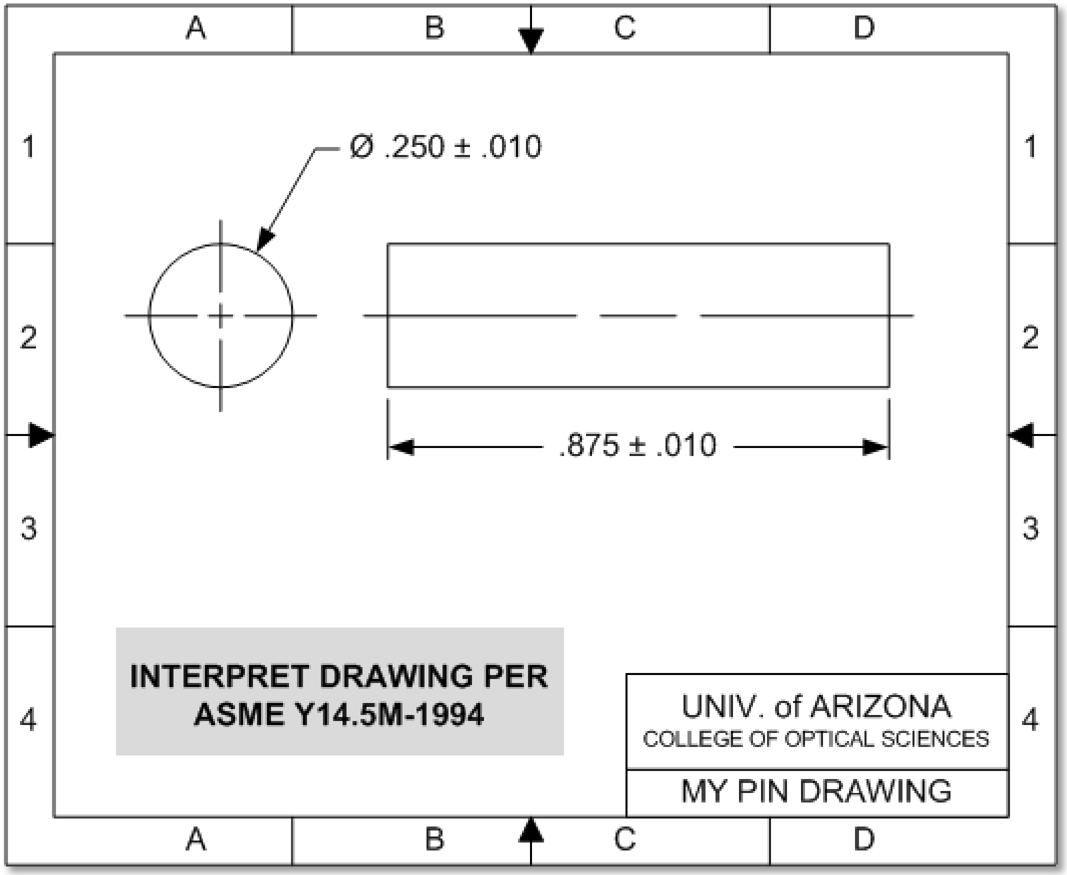
HIGH

VERY, VERY LOW

Success Scale

Another Option is to Use Geometric Dimensioning & Tolerancing (GD&T)

- Reference ASME Y14.5M-1994 on your drawing
- For the pin, that's all you need! Problem solved.



What is GD&T?

- "The purpose of GD&T is to describe the engineering intent of parts and assemblies" –ASME Y14.5M-1994
- GD&T is covered by several standards
  - **ASME Y14.5 - 2009** Dimensioning and Tolerancing
  - **ASME Y14.5M-1994** Dimensioning and Tolerancing
  - **ASME Y14.5.1M-1994** Mathematical Definition of Dimensioning and Tolerancing Principles
  - **ISO 286-1:1988** *ISO system of limits and fits — Part 1: Bases of tolerances, deviations and fits*
  - **ISO 286-2:1988** *ISO system of limits and fits — Part 2: Tables of standard tolerance grades and limit deviations for holes and shafts*
  - **ISO 1101:2005** *Geometrical Product Specifications (GPS) — Geometrical tolerancing — Tolerancing of form, orientation, location and run-out*
  - **ISO 5458:1998** *Geometric Product Specifications (GPS) — Geometrical tolerancing — Positional tolerancing*
  - **ISO 5459:1981** *Technical drawings — Geometrical tolerancing — Datums and datum-systems for geometrical tolerances*
- ASME Y14.5M-1994 will be discussed here
  - Still widely used in American Industry

ASME Y14.5M-1994

- Developed and published by the American Society of Mechanical Engineers
- Reaffirmed in 1999 and 2004
- Replaced by ASME Y14.5M-2009
- Committee members are selected from industry and volunteer their time (not paid)

Key Concepts of GD&T

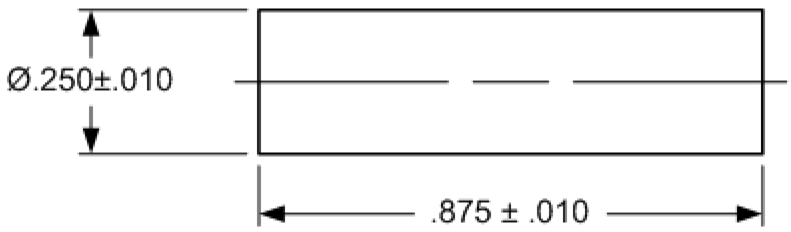
- Controls size, form, orientation, and location between features
- A feature can be a hole, surface, boss, radius, etc.
  - It can have either size or no size
  - Features of size have opposed elements
- Defines feature material conditions
  - Maximum material condition (MMC)
    - Variation of size where the feature contains the most material
  - Least material condition (LMC)
    - Variation of size where the feature contains the least material
- Tolerance allocation can be based on material condition
- Use of datums and basic dimensions to establish tolerance zones

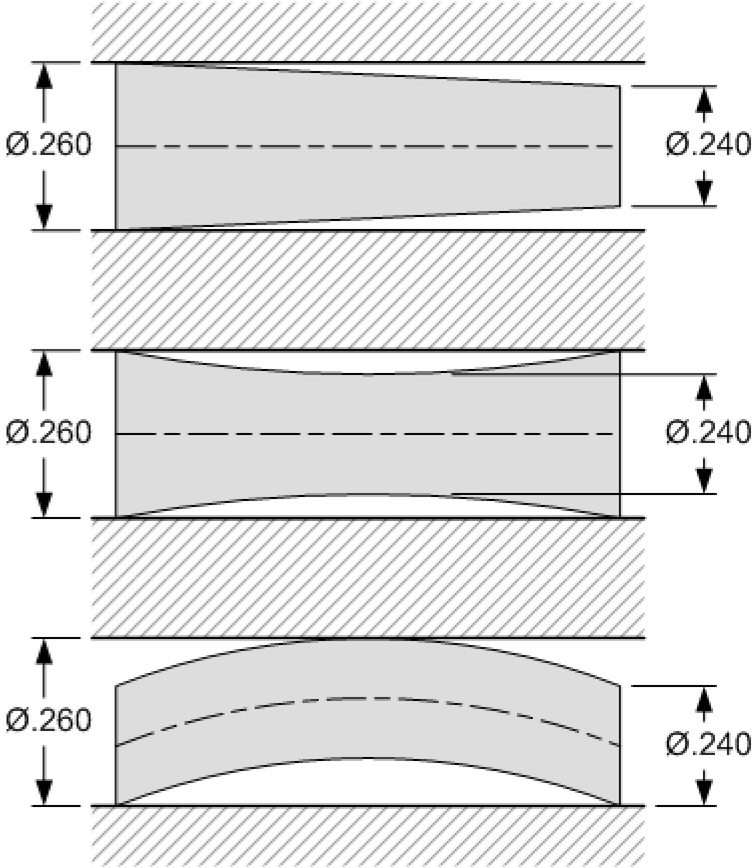
Rule #1

(aka Envelope Rule or Taylor Principal)

- **A)** The surface or surfaces of a feature shall not extend beyond a boundary (envelope) of perfect form at **MMC**
- **B)** Where the actual local size of a feature has departed from **MMC** toward **LMC**, a variation is form is allowed equal to the amount of such departure
- **C)** There is no requirement for a boundary of perfect form at **LMC**. At **LMC** the feature is permitted to vary from true form to the maximum variation allowed by the boundary of perfect form at **MMC**
- **Rule #1** can be negated by including the note PERFECT FOR AT **MMC** NOT REQUIRED

Rule #1 in Pictures

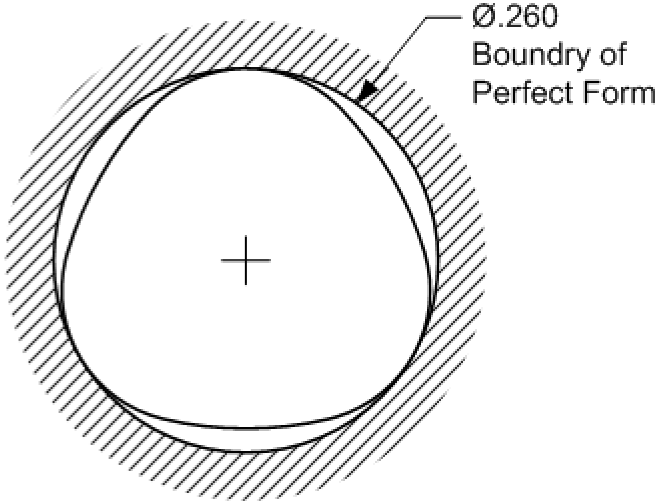


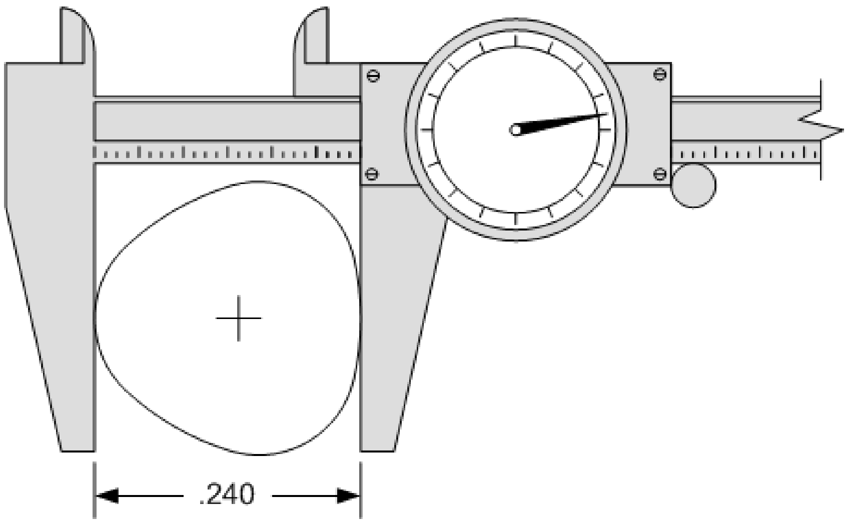


- Perfect form is required at **MMC**
- Variation in form is permitted as the feature departs from **MMC**
- Maximum allowable variation in form occurs at **LMC** not to exceed the boundary set by **MMC**

My Part Fits!

Thanks GD&T

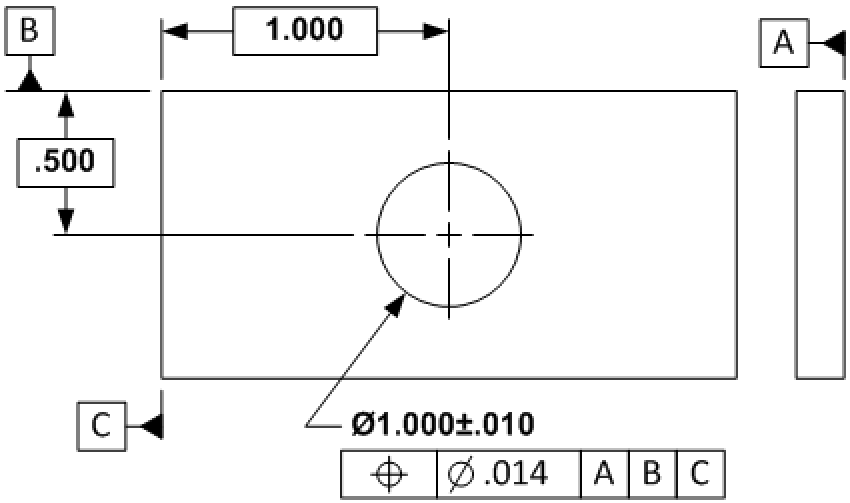


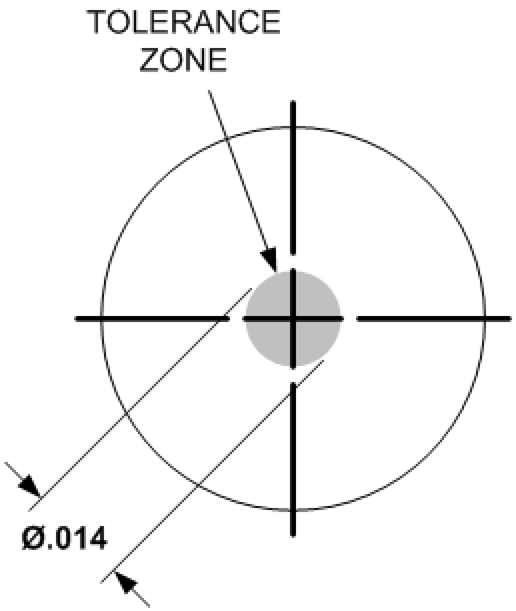


The pin can still be lobed, however now the variation in form cannot exceed the boundary of perfect form established at **MMC**

Cylindrical Tolerance Zones

Regardless of Feature Size (RFS)



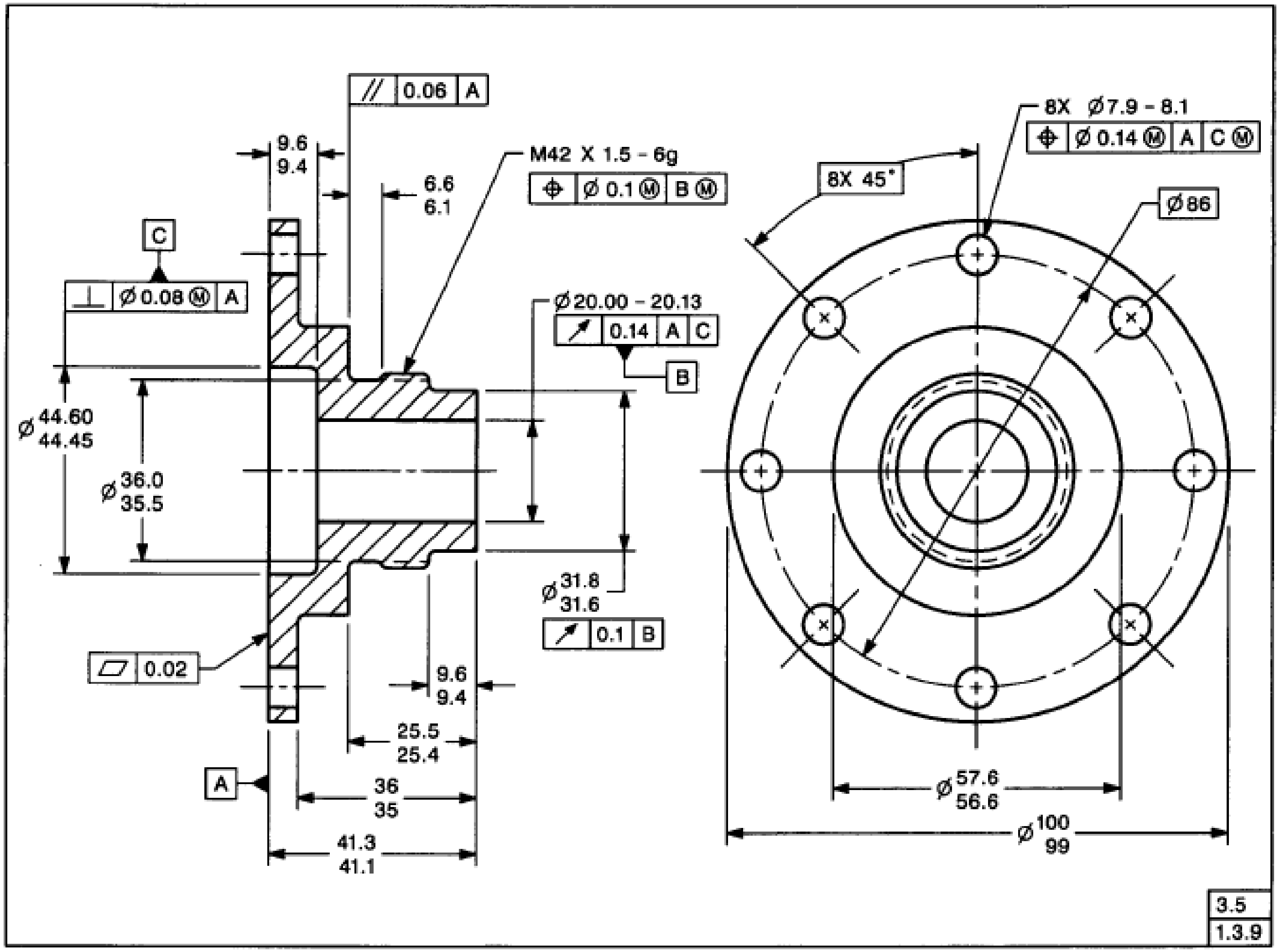


- In the example shown, the tolerance zone diameter remains constant regardless of the hole size
- Datums A, B, and C establish which part features and in what order they are used to locate the tolerance zone
- The basic dimensions provide the exact geometric position of the tolerance zone

Just the Tip of the Iceberg

- Tolerance zones can vary based on feature size
- Able to apply different constraints to location, orientation, and form all for the same feature
- Can add tolerance to complex surfaces
- Tolerance zones can be projected
- Interrelationship of patterned features can be controlled separately from the pattern location
- Strong application to optical assemblies due to axis controls

Questions?



// 0.06 A
9.6
9.4
6.6
6.1
M42 X 1.5 - 6g
⌖ Ø 0.1 Ⓜ B Ⓜ
C
⊥ Ø 0.08 Ⓜ A
Ø 20.00 - 20.13
↗ 0.14 A C
B
Ø 44.60
44.45
Ø 36.0
35.5
Ø 31.8
31.6
↗ 0.1 B
▱ 0.02
9.6
9.4
25.5
25.4
A
36
35
41.3
41.1
8X Ø7.9 - 8.1
⌖ Ø 0.14 Ⓜ A C Ⓜ
8X 45°
Ø86
Ø 57.6
56.6
Ø 100
99
3.5
1.3.9