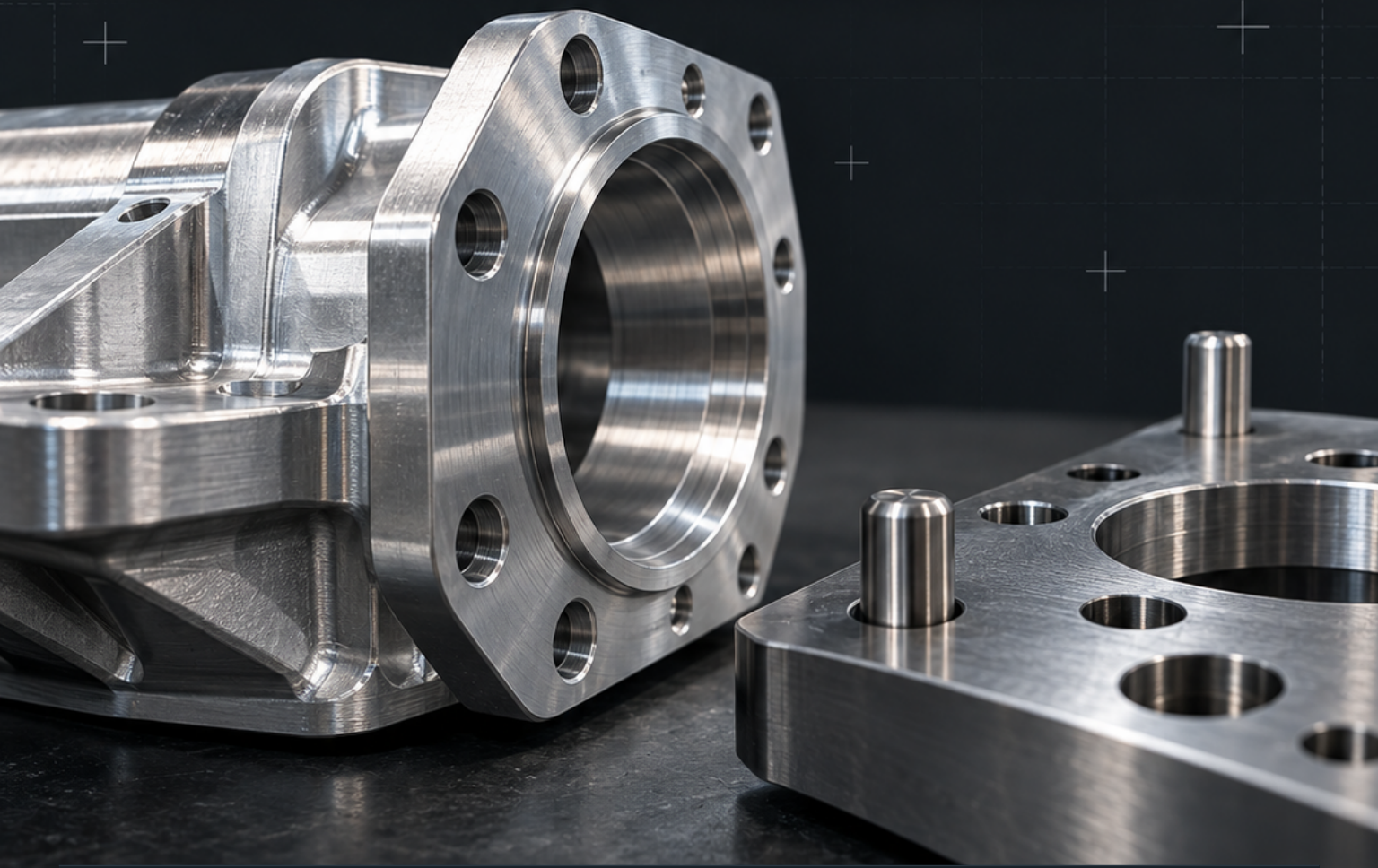
# THE SIZE IS RIGHT. THE PART STILL FAILS.

## GD&T FOR CNC MANUFACTURING

How geometry controls fit, function and inspection



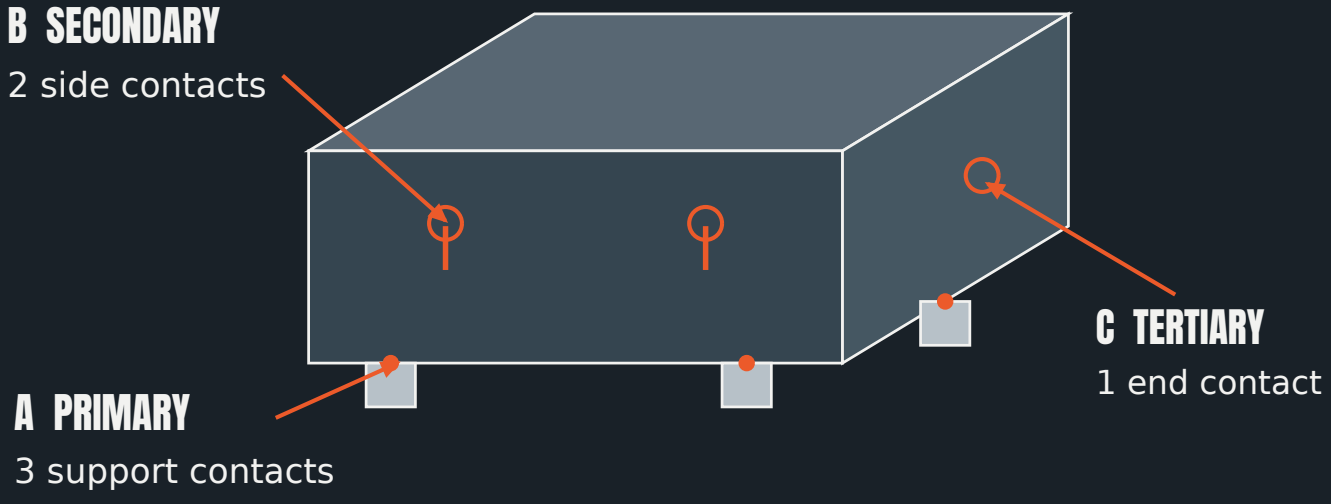
GD&T essentials - including legacy drawing controls.

SWIPE TO READ THE GEOMETRY

# THE REFERENCE COMES FIRST

A datum feature is physical. The datum is theoretically exact.

ILLUSTRATIVE PLANAR 3-2-1 LOCATION




Locators shown; clamps omitted. Not every datum scheme is 3-2-1.

## PRIMARY A

Establish the first reference.

## SECONDARY B

Constrain the next degrees of freedom.

## TERTIARY C

Complete the required location.

**Datum order changes how the part is evaluated.**

Read the drawing standard and the complete callout.

# POSITION

## WHERE THE FEATURE BELONGS

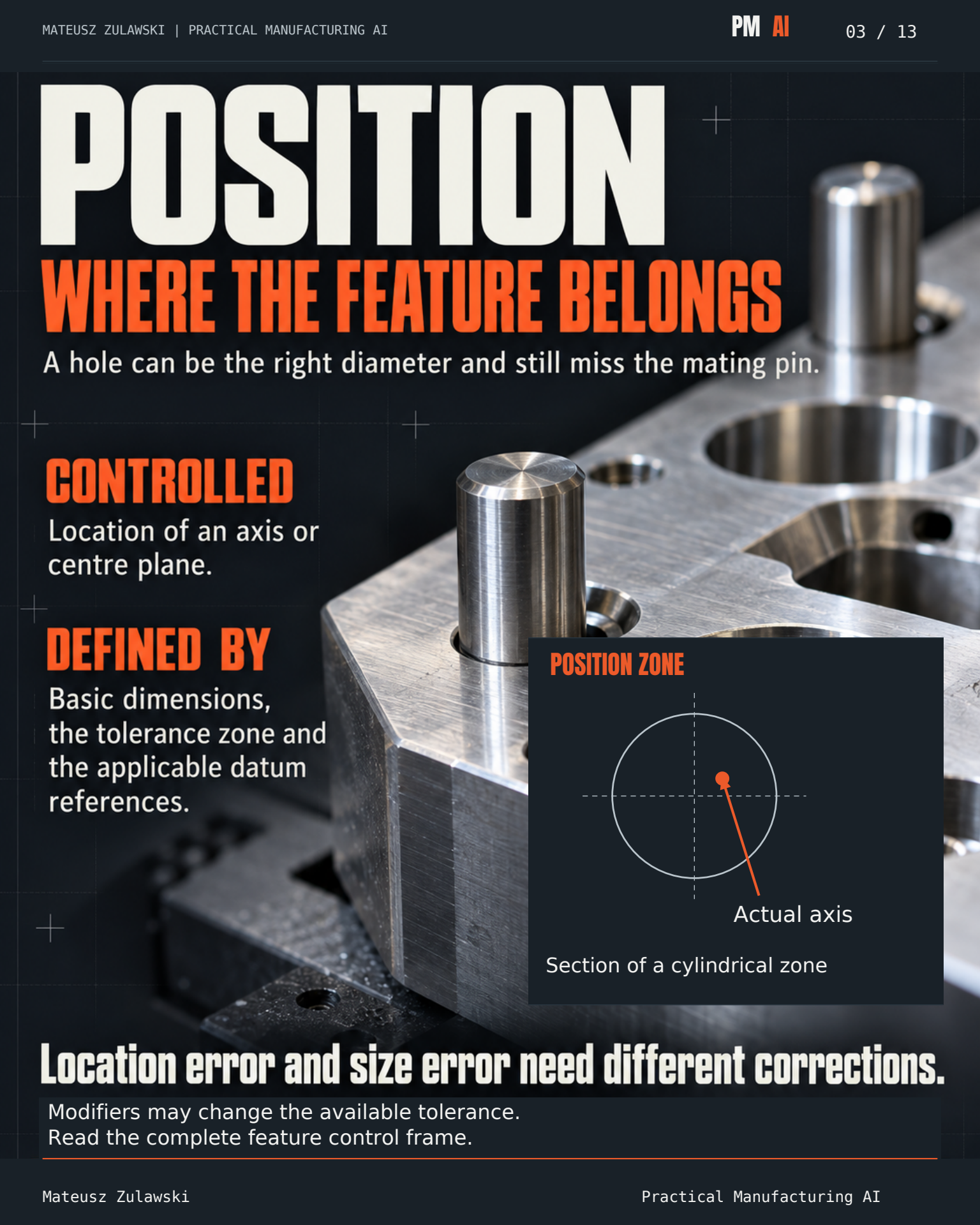
A hole can be the right diameter and still miss the mating pin.

### CONTROLLED

Location of an axis or centre plane.

### DEFINED BY

Basic dimensions, the tolerance zone and the applicable datum references.



### POSITION ZONE



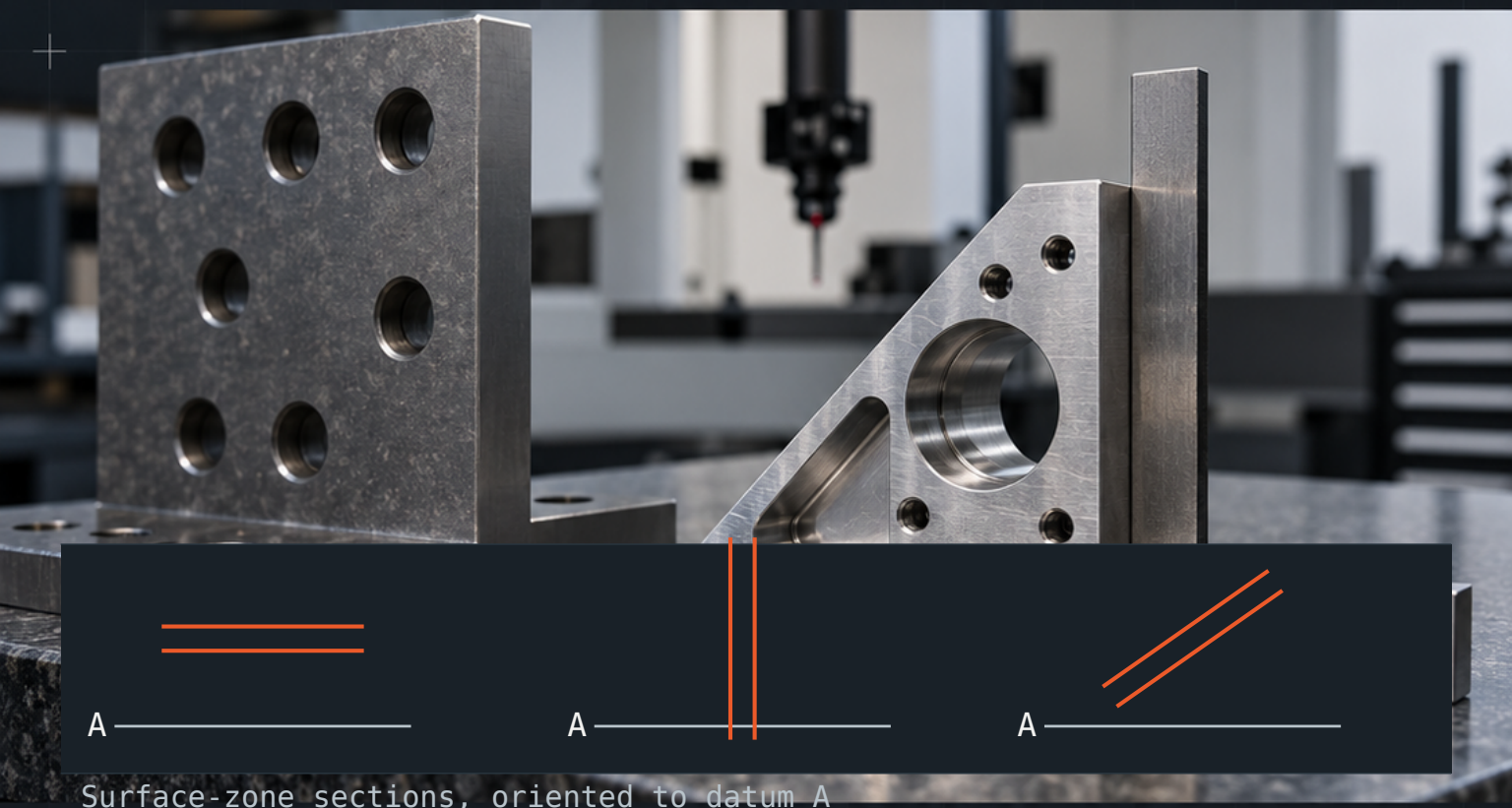

Section of a cylindrical zone

**Location error and size error need different corrections.**

Modifiers may change the available tolerance.
Read the complete feature control frame.

# ORIENTATION

## THE ANGLE NEEDS A REFERENCE




Surface-zone sections, oriented to datum A

### PARALLELISM

Direction parallel to the datum.

### PERPENDICULARITY

Direction at 90 degrees to the datum.

### ANGULARITY

Direction at the specified basic angle.

**These controls limit orientation within a zone. They do not by themselves fix the feature's location.**

Zone geometry depends on the controlled feature.

DRAWING ERROR / ILLUSTRATIVE EXAMPLE

# 0.05 IS NOT AN ANGLE.

Perpendicularity is specified as a length.

## INCORRECT CALLOUT

| ⊥ | 0.05° | A |
|---|---|---|

## CORRECT / DRAWING IN mm

| ⊥ | 0.05 | A |
|---|---|---|




Two parallel planes,
0.05 mm apart,
perpendicular to A.

Section shown; zone exaggerated.

## THE CHECK

Evaluate the whole specified surface in the zone.
An angle reading alone does not establish conformity.

**READ THE ZONE. THEN CHOOSE THE MEASUREMENT.**

# STRAIGHTNESS & FLATNESS

Line elements and whole surfaces answer different questions.



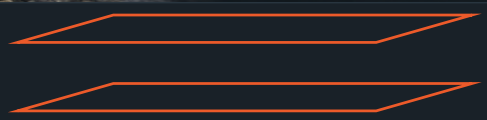
## STRAIGHTNESS

A surface line fits between two parallel lines.



## FLATNESS

The entire surface fits between two parallel planes.

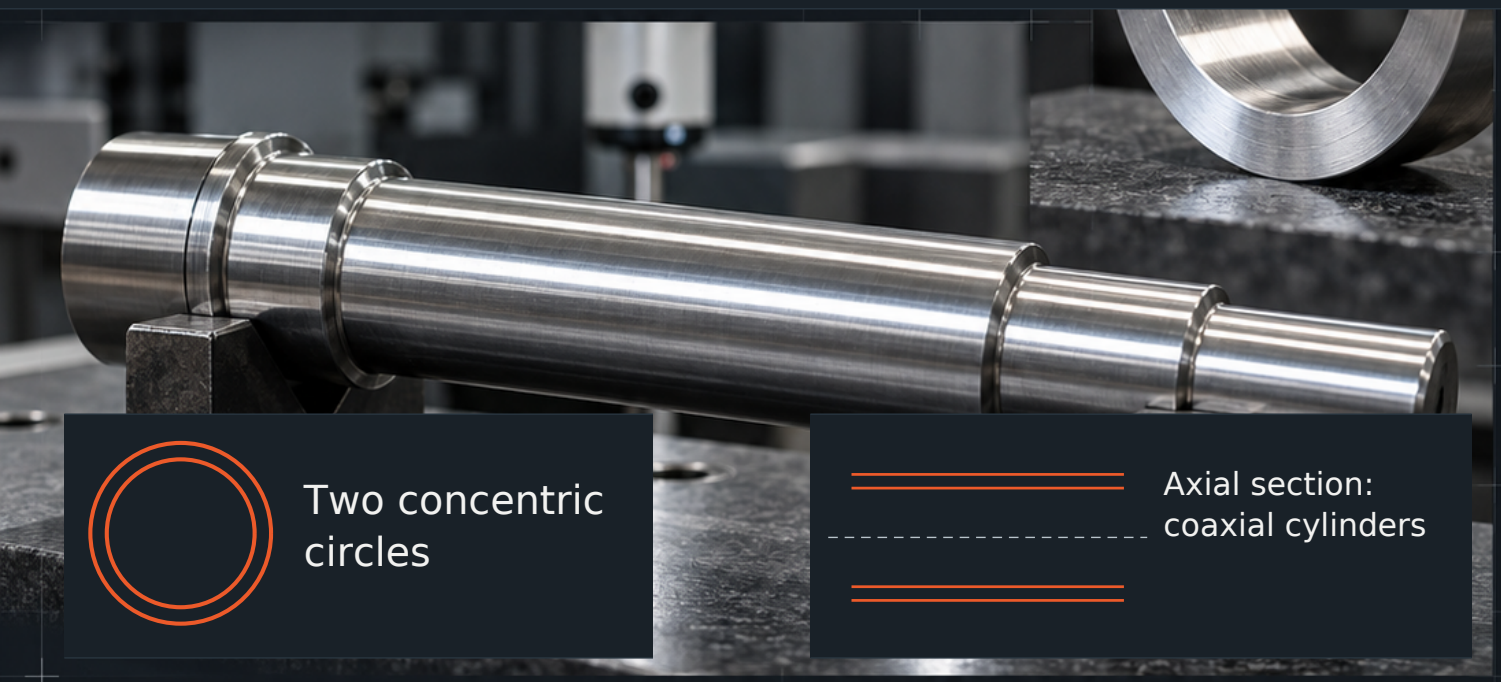


Two parallel planes

## Neither form control references a datum.

Surface examples. Axis straightness uses a different zone.

# ROUND IN ONE SECTION.
# CYLINDRICAL OVER THE WHOLE SURFACE?




## CIRCULARITY

Each circular section fits between two concentric circles.

## CYLINDRICITY

The whole cylindrical surface fits between two coaxial cylinders.

## A round section does not prove the whole cylinder's form.

No datum reference. Size limits remain a separate requirement.

# PROFILE
# THE DESIGNED CONTOUR

Curved geometry still needs a defined tolerance zone.



## PROFILE ZONE / SECTION



Illustrative equal bilateral zone.
The drawing defines its disposition.

## PROFILE OF A LINE

Controls the contour in specified cross-sections.

## PROFILE OF A SURFACE

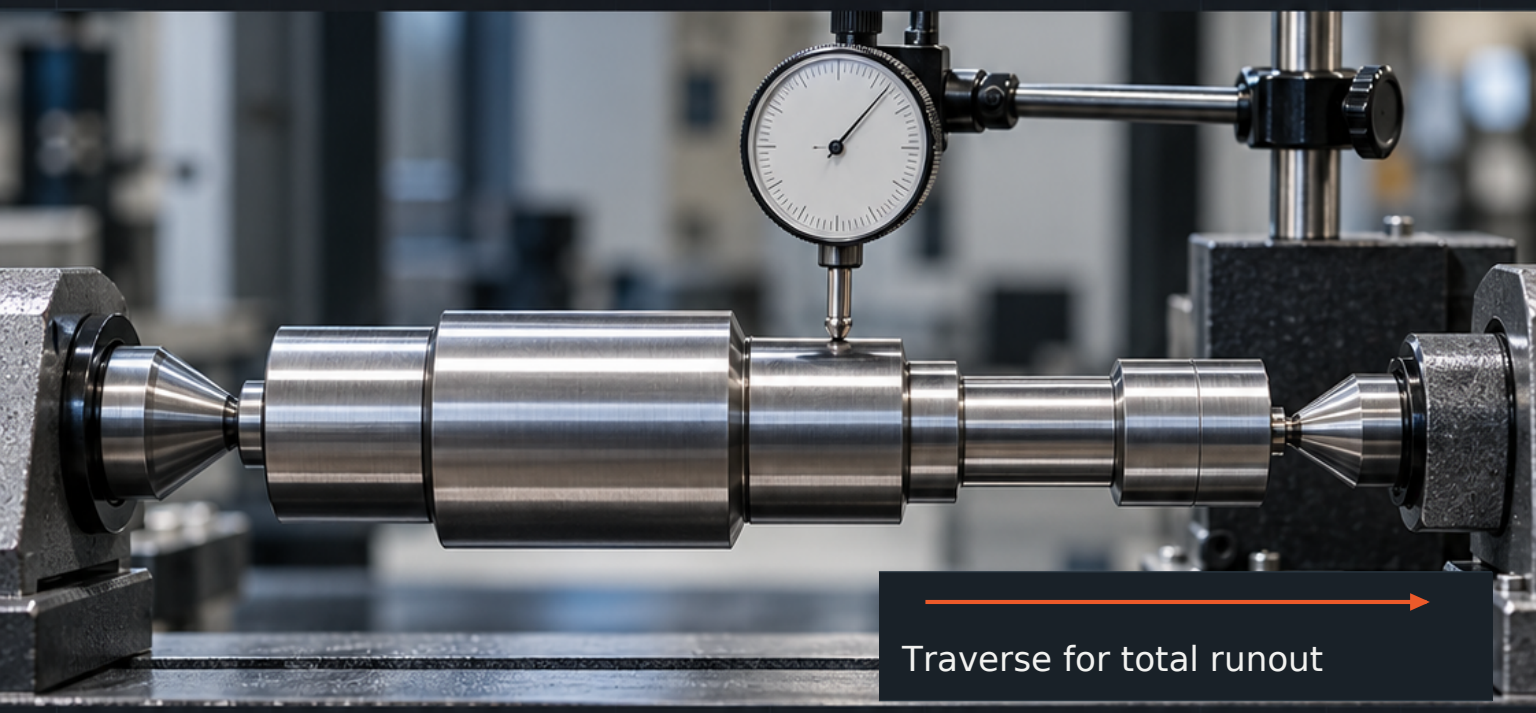
Controls the full specified surface.

**Datum references can anchor orientation and location.**

The drawing defines the nominal profile and zone arrangement.

# RUNOUT
## GEOMETRY DURING ROTATION

Both controls evaluate the surface relative to a datum axis.




### CIRCULAR RUNOUT

Evaluate each circular element during rotation.

### TOTAL RUNOUT

Evaluate the entire specified surface during rotation.

## One indicator trace does not explain every source of error.

Between-centres inspection must establish the specified datum axis. Radial example shown.

CONCENTRICITY / SYMMETRY

# OLDER DRAWINGS NEED THEIR OWN RULES

### LEGACY CONCENTRICITY

Median points relative to a datum axis.

### LEGACY SYMMETRY

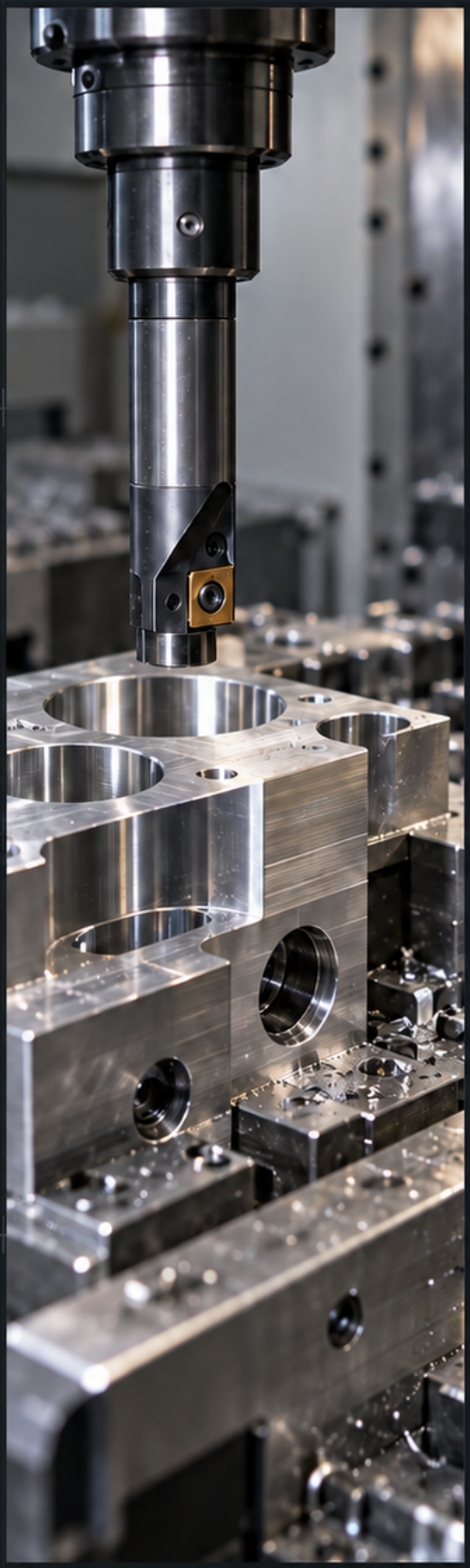
Median points relative to a datum centre plane.



**2018** | Removed from ASME Y14.5-2018.

## THE DRAWING EDITION STILL GOVERNS.

ISO GPS has its own rules.
Change legacy callouts only with engineering approval.



ILLUSTRATIVE CNC DIAGNOSTIC

# DIFFERENT FAILURES. DIFFERENT FIRST CHECKS.

## ALL HOLES SHIFTED

Check the datum, seating and coordinate system.

## ONE BORE UNDERSIZE

Check the tool, compensation and cutting behaviour.

## RUNOUT TOO HIGH

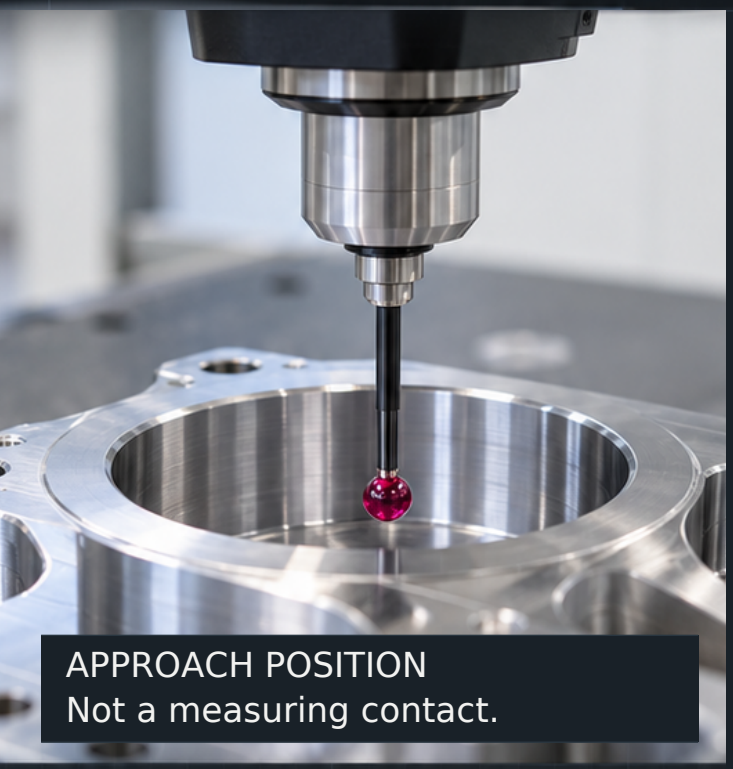
Check datum establishment, seating and surface geometry.

**The failed requirement tells you what to investigate.**

An offset can move geometry. It cannot remove every cause.

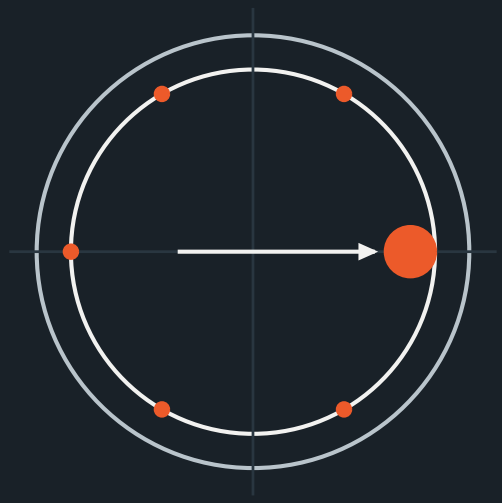
# THE MEASUREMENT MUST MATCH THE REQUIREMENT

A convenient reading can miss the controlled geometry.



APPROACH POSITION
Not a measuring contact.

## BORE CONTACT / TOP VIEW



Ball touches the wall

Sample around and at multiple heights.

## DEFINE THE REQUIREMENT

Feature + datum

Zone + sampling

## VALIDATE THE METHOD

Uncertainty + calibration

Contact + conditions

## MORE DIGITS DO NOT PROVE CONFORMITY.

The method must evaluate the specified characteristic.

# SAVE THE GEOMETRY MAP

**Read the requirement before choosing the measurement.**

## FORM

- Straightness
- Flatness
- Circularity
- Cylindricity

## ORIENTATION

- Parallelism
- Perpendicularity
- Angularity

## LOCATION

- Position

## PROFILE

- Profile of a line
- Profile of a surface

## RUNOUT

- Circular runout
- Total runout

Legacy ASME: concentricity / symmetry.

Check the drawing standard and edition.

## SAVE THE MAP.

**WHICH GD&T SYMBOL CAUSES THE MOST CONFUSION ON YOUR SHOP FLOOR? COMMENT BELOW.**

FOLLOW MATEUSZ ZULAWSKI | PRACTICAL MANUFACTURING AI