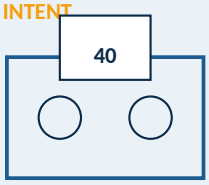
KINGSFORM METALWORKS

ENGINEERING TOOLKIT

GD&T QUICK GUIDE

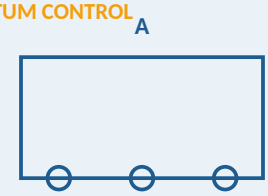
Practical interpretation of datums, feature control frames, geometric controls and inspection planning

DESIGN INTENT



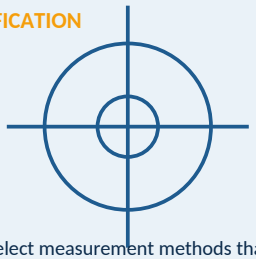
Replace ambiguous +/- coordinate tolerances with functional tolerance zones.

DATUM CONTROL



Establish a repeatable coordinate system that matches assembly and inspection.

VERIFICATION



Select measurement methods that reproduce the stated datum setup and tolerance zone.

DOCUMENT	VERSION	RELEASE DATE	LANGUAGE
KFR-017	1.0	2026-07-18	English
FORMAT	OWNER	CONTACT	WEBSITE
A4 PDF reference	KingsForm Engineering	info@kingsformmetalworks.com	kingsformmetalworks.com

01 SYSTEM, SCOPE AND DRAWING HIERARCHY

Choose ASME or ISO GPS before applying symbols

The symbol language is similar, but defaults, terminology and modifier rules are not fully interchangeable. State the governing standard and edition in the drawing package.

CURRENT CORE REFERENCES - STATUS CHECKED 18 JULY 2026

ASME Y14.5-2018 is listed by ASME as the current edition and reaffirmed in 2024. ISO geometrical specification is distributed across ISO 1101:2017, ISO 5459:2024 and related GPS standards.

Topic	ASME drawing system	ISO GPS drawing system
Core geometric tolerancing	ASME Y14.5-2018 (R2024)	ISO 1101:2017
Datums	Rules contained primarily in Y14.5	ISO 5459:2024
Material boundary rules	MMC/LMC and datum boundary rules in Y14.5	MMR/LMR/RPR in ISO 2692:2021
Pattern specifications	Position and pattern rules in Y14.5	ISO 5458:2018
Profile specifications	Profile rules in Y14.5	ISO 1660:2017 plus ISO 1101
Linear size rules	Y14.5 size rules and related Y14 standards	ISO 14405-1:2025 and ISO 8015:2011

DO NOT MIX DEFAULTS

A drawing can use company-specific rules, but the contract must state exactly which clauses, modifiers and defaults apply. Never assume an ASME interpretation for an ISO callout or vice versa.

1 **Function**

Identify fit, load path, sealing, motion, appearance and interchangeability.

2 **Datum scheme**

Select contact features that repeat the assembly coordinate system.

3 **Geometric control**

Choose the zone that directly limits the failure mode.

4 **Size control**

Specify mating size, limits and material boundary only where needed.

5 **Verification**

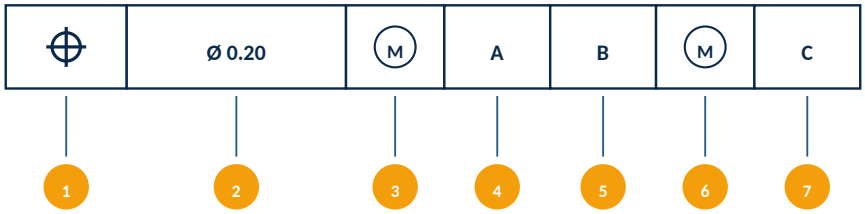
Define datum simulation, measurement method and acceptance rule.

02 FEATURE CONTROL FRAME ANATOMY

Read the frame from left to right

The frame states the geometric characteristic, tolerance-zone shape and value, material condition when applicable, and the ordered datum reference frame.

EXAMPLE FEATURE CONTROL FRAME



Characteristic	Zone + value	Feature modifier	Primary datum	Secondary datum	Datum boundary	Tertiary datum

Family	Controls	Datum normally required?	Typical zone
Form	Straightness, flatness, circularity, cylindricity	No	Two lines/planes, two circles or coaxial cylinders
Orientation	Parallelism, perpendicularity, angularity	Yes	Two lines/planes or a cylindrical zone at basic angle
Location	Position	Usually yes	Cylindrical or parallel-plane zone at basic location
Profile	Profile of a line / surface	Optional	Boundary around the true profile
Runout	Circular runout / total runout	Yes - datum axis	Indicator variation during rotation

Straightness	Flatness	Circularity	Cylindricity	Parallelism	Perpendicularity
Angularity	Position	Line profile	Surface profile	Circular runout	Total runout

03 DATUMS AND DATUM REFERENCE FRAMES

A datum is exact; the datum feature is manufactured

The part contacts a datum feature simulator. That simulator establishes the theoretically exact datum plane, axis or center plane used to orient and locate the tolerance zones.

DATUM FEATURE

The real surface or feature identified on the drawing.

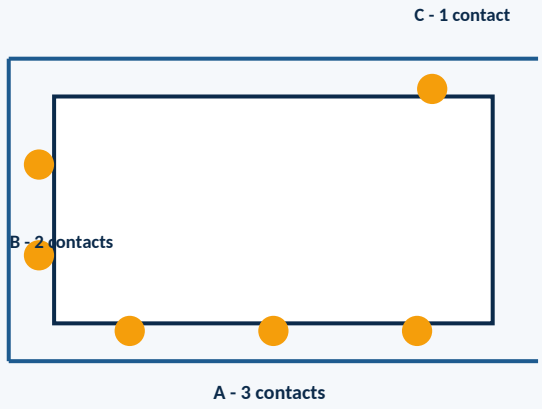
DATUM

The theoretically exact plane, axis, point or center plane derived from the feature.

DATUM SIMULATOR

The physical or mathematical surface used to establish the datum during assembly or inspection.

PLANAR 3-2-1 CONCEPT



Conceptual restraint of six degrees of freedom

Datum order	Function
Primary A	Stabilizes the part and removes three degrees of freedom.
Secondary B	Orients the part to A and removes two additional freedoms.
Tertiary C	Completes the reference frame by removing the final freedom.
Composite / common datum	Use only when the assembly simultaneously contacts multiple features as defined by the governing system.

FUNCTION FIRST

Do not select datums simply because a surface is large or easy to measure. Choose stable features that reproduce assembly contact, load transfer, alignment and service conditions. A cosmetic face is rarely the best primary datum.

04 FORM CONTROLS

Control the shape of a feature without locating it

Form controls normally do not reference datums. They limit how far the actual feature may depart from an ideal line, plane, circle or cylinder.

STRAIGHTNESS

Line element or, under specific size applications, a derived axis/median line.

No datum

FLATNESS

Entire surface must lie between two parallel planes.

No datum

CIRCULARITY

Each circular cross-section lies between two concentric circles.

No datum

CYLINDRICITY

Whole cylindrical surface lies between two coaxial cylinders.

No datum

Control	Protects against	Typical inspection	Important limitation
Straightness	Bow or local line waviness	Indicator, optical method, CMM	Does not orient or locate the feature.
Flatness	Rocking, poor sealing, local high spots	Surface plate and indicator, CMM	Does not control parallelism to another surface.
Circularity	Out-of-round cross-sections	Roundness tester, CMM section scan	Each section is independent; no axis location.
Cylindricity	Combined roundness, straightness and taper of a cylinder	Form tester or dense CMM scan	Does not locate the cylinder relative to datums.

FLATNESS IS NOT PARALLELISM

A perfectly flat surface may still be tilted relative to datum A.

▱	0.10

Use parallelism to A when orientation also matters.

05 ORIENTATION CONTROLS

Orient a surface or axis to a datum

Parallelism, perpendicularity and angularity establish a tolerance zone at a theoretically exact orientation. They do not define the basic location of the feature.

PARALLELISM

Zone is parallel to the referenced datum.

Datum required

PERPENDICULARITY

Zone is exactly 90 degrees to the datum.

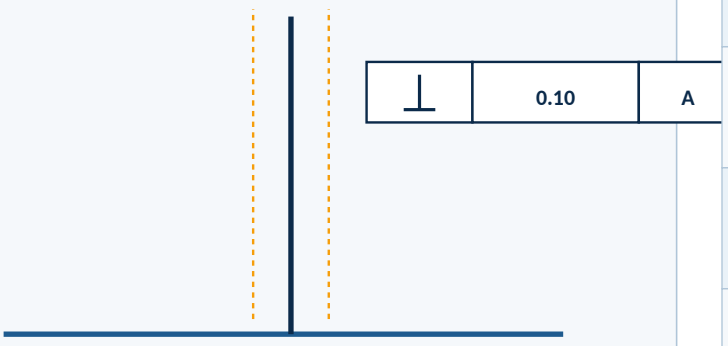
Datum required

ANGULARITY

Zone is at a specified basic angle other than 0 or 90 degrees.

Datum required

SURFACE PERPENDICULARITY EXAMPLE



DATUM A
Actual surface must remain between two planes 0.10 apart and perpendicular to A.

Question	Design answer
What is being controlled?	Surface orientation or the orientation of a derived axis / center plane.
What is the zone shape?	Two parallel planes for a surface; a cylindri zone when Ø is shown for an axis.
What is not controlled?	Exact feature location unless a location cor such as position or profile is added.
Can form exceed orientation?	No. The oriented feature must fit inside th orientation zone, so its form is limited by t zone.

BASIC ANGLES

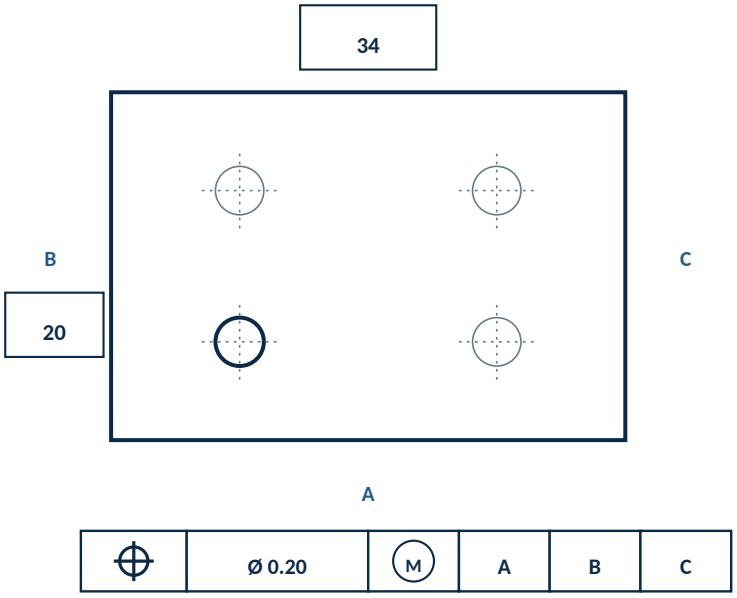
Angularity needs a theoretically exact/basic angle. A +/- angle dimension creates a wedge-shaped coordinate tolerance, not a constant-width angularity zone. Use the control that matches the functional boundary.

06 POSITION AND FEATURE PATTERNS

Control the axis or center plane at a basic location

Position is normally used for holes, pins, slots and patterns. Feature size and feature location are separate requirements and should be verified separately.

4-HOLE PATTERN



CMM TRUE-POSITION CALCULATION

For a cylindrical position zone, convert X/Y center deviation to a diametrical result:

Position = 2 x sqrt(DeltaX^2 + DeltaY^2)

Input	Value
DeltaX	+0.08 mm
DeltaY	-0.06 mm
Radial offset	0.10 mm
Diametrical position	0.20 mm

Result: passes a Ø0.20 RFS zone exactly at the limit, subject to the agreed decision rule and measurement uncertainty.

Design issue	Use position when...	Check separately
Bolt-hole pattern	Fastener clearance and interchangeability depend on axes relative to datums.	Hole size, edge distance, counterbore and surface condition.
Pin / boss	A mating feature must remain within a functional cylindrical boundary.	Pin size, form, perpendicularity if not inherent in the position requirement.
Slot	The derived center plane must be located and oriented.	Slot width, end radius, profile of the slot boundary.
Pattern to pattern	Multiple features must maintain mutual relationship and/or relationship to datums.	Whether composite or separate single-segment controls match assembly function.

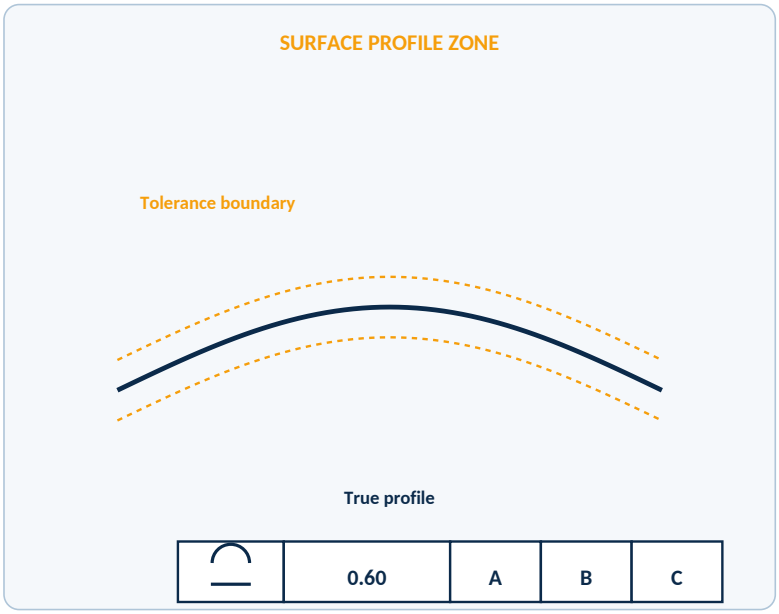
BASIC DIMENSIONS

Basic dimensions are theoretically exact. The geometric tolerance creates the permissible variation; do not add a +/- tolerance to the same location unless the governing standard and design intent explicitly require a separate control.

07 PROFILE OF A LINE AND PROFILE OF A SURFACE

Control complex contours with one functional boundary

Profile is especially useful for formed sheet, castings, forgings, tube shapes, aerodynamic surfaces and cosmetic boundaries where coordinate tolerances create uneven zones.



Callout	What it controls
Line profile	Each specified cross-section or line elemen interpretation depends on the indicated ex
Surface profile	The entire indicated surface in 3D.
No datum references	Form of the true profile only.
With A | B | C	Form, orientation and location relative to t reference frame.
Unequal / unilateral zone	Use the system-specific modifier or explicit graphical definition. Do not rely on informa

WHY PROFILE OFTEN REDUCES AMBIGUITY

- A constant-width boundary follows the true geometry, unlike independent X/Y coordinate limits that can create a square or wedge-shaped acceptance region.
- One callout can control blends, radii, freeform contours and adjacent surfaces when the drawing clearly defines the extent.
- Profile can incorporate orientation and location through datums, reducing stacks of separate dimensions and geometric controls.
- Inspection must use a sufficiently dense scan or defined section plan; a few point checks can miss local deviations.

SYSTEM DIFFERENCE

ASME and ISO use different notation and default rules for unequally disposed profile zones and specification modifiers. Confirm the governing edition before interpreting an offset zone.

08 CIRCULAR AND TOTAL RUNOUT

Control rotating surfaces relative to a datum axis

Runout is measured while the part rotates about a datum axis. It combines effects of surface form, orientation and coaxial location for the indicated surface.

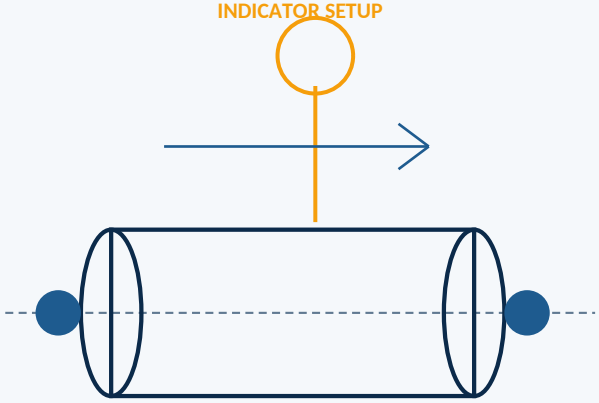
CIRCULAR RUNOUT

Indicator variation at each individual circular section during one revolution. Sections are evaluated independently.

TOTAL RUNOUT

Indicator variation over the entire surface while the part rotates and the indicator traverses the full length.

INDICATOR SETUP



Rotate about datum axis

Question	Engineering interpretation
Datum requirement	A stable datum axis must be established specified datum feature(s).
Circular runout	Useful for local radial or face variation a section at a time.
Total runout	Adds control along the complete surfac usually more restrictive.
Not the same as size	Diameter limits remain a separate requi
Not generic concentricity	Runout is a surface-based rotating meas not a center-point calculation.

FIXTURE EFFECT

A poor datum simulator, dirty centers, excessive clamping or bearing clearance can dominate the measured runout. The drawing, inspection plan and fixture must define the same axis-establishment concept.

09 MATERIAL CONDITION AND BOUNDARY MODIFIERS

Use modifiers only for features of size and functional boundaries

The ASME worked example below illustrates bonus tolerance and virtual condition. ISO maximum material requirement is related but uses its own terminology and rule structure.

Term	Internal feature (hole)	External feature (pin)	Design use
Maximum material condition / MMC	Smallest permitted hole	Largest permitted pin	Worst-case material boundary for assembly
Least material condition / LMC	Largest permitted hole	Smallest permitted pin	Protect minimum wall or edge thickness
RFS / default size independence	Geometric tolerance does not change with actual feature size	Same concept for pin	Use when function is independent of mating clearance
Datum boundary modifier	May permit datum feature shift/mobility	May permit datum feature shift/mobility	Reflects assembly clearance at datum features

ASME EXAMPLE - HOLE POSITION AT MMC

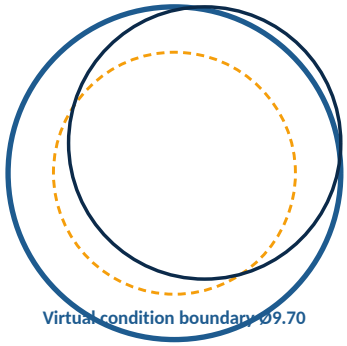
CALLOUT

Hole size: Ø10.0 +/-0.1

⌖	Ø 0.20	Ⓜ	A	B	C

Step	Result
Hole MMC	Ø9.90
Actual hole	Ø10.05
Bonus tolerance	10.05 - 9.90 = 0.15
Total position tolerance	0.20 + 0.15 = Ø0.35

FUNCTIONAL BOUNDARY



Actual hole may move inside the allowed position zone as its size departs from MMC.

INTERNAL FEATURE

VC = MMC size - geometric tolerance at MMC

EXTERNAL FEATURE

VC = MMC size + geometric tolerance at MMC

DO NOT APPLY MMC TO A SURFACE

MMC/LMC modifiers apply to regular features of size, such as a cylinder or two opposed parallel planes. They are not universal modifiers for flatness of a single surface or profile of an arbitrary contour.

10 INSPECTION PLANNING AND COMMON ERRORS

Verify the tolerance zone, not just convenient points

The measurement setup must reproduce the datum reference frame, feature association and decision rule stated by the governing drawing system.

Control	Typical method	Critical setup issue
Flatness	Surface plate and indicator, CMM or form tester	Support condition, sampling density and unrestrained vs restrained state
Perpendicularity / parallelism	CMM, surface plate with indicator, functional fixture	Correct datum simulation and sufficient sweep length
Position	CMM, optical system, fixed/functional gauge	Feature axis association, datum order and material boundary rules
Profile	CMM, scanner, contour system or checking fixture	Coverage of the complete indicated surface and datum alignment
Runout	Indicator while rotating on datum simulator	Axis establishment, bearing/center quality and indicator direction

TEN HIGH-RISK DRAWING ERRORS

1. No governing ASME or ISO edition is stated.
2. Datums are selected for inspection convenience rather than assembly function.
3. Position is used to control a surface boundary instead of an axis or center plane.
4. Flatness is expected to make a surface parallel to another feature.
5. Profile extent is unclear, so supplier and customer inspect different areas.
6. MMC/LMC is applied to a feature that is not a feature of size.
7. A basic dimension is also given a conflicting +/- tolerance.
8. Datum order A | B | C is changed in the inspection setup.
9. Coordinate point checks are used even though the tolerance is a continuous surface zone.
10. Pass/fail decisions ignore measurement uncertainty near the specification limit.

ACCEPTANCE RULE

For ISO GPS verification, ISO 14253-1:2017 addresses conformity decisions near specification limits while accounting for measurement uncertainty. Customer and supplier should agree the decision rule before production.

11 RELEASE CHECKLIST AND OFFICIAL REFERENCES

Twenty-four checks before a GD&T drawing is issued

Use this page for design review, supplier handoff, first-article planning and inspection-program approval.

- ☐ 01 Governing ASME or ISO system and edition stated
- ☐ 02 Drawing defaults and company supplements controlled
- ☐ 03 Functional requirements identified
- ☐ 04 Datum features reproduce assembly contact
- ☐ 05 Primary datum provides stable support
- ☐ 06 Secondary and tertiary datums remove remaining freedoms
- ☐ 07 Datum order is consistent across related callouts
- ☐ 08 Feature control frame characteristic matches failure mode
- ☐ 09 Tolerance-zone shape is understood
- ☐ 10 Basic dimensions define exact location/orientation
- ☐ 11 Feature size is controlled separately where required
- ☐ 12 Form controls do not incorrectly reference datums
- ☐ 13 Orientation control does not substitute for location
- ☐ 14 Position applies to axis or center plane as intended
- ☐ 15 Profile extent is clearly indicated
- ☐ 16 Profile zone disposition is unambiguous
- ☐ 17 Runout datum axis can be physically simulated
- ☐ 18 MMC/LMC only applied to features of size
- ☐ 19 Bonus tolerance and virtual condition checked
- ☐ 20 Datum boundary modifier effect evaluated
- ☐ 21 Tolerance stack and mating clearance verified
- ☐ 22 Inspection method reproduces datum setup
- ☐ 23 Measurement uncertainty and decision rule agreed
- ☐ 24 FAI/report format identifies every controlled characteristic

OFFICIAL REFERENCES - STATUS CHECKED 18 JULY 2026

ASME Y14.5-2018 (R2024) - Dimensioning and Tolerancing. https://www.asme.org/codes-standards/find-codes-standards/y14-5-dimensioning-tolerancing

ISO 1101:2017 - Geometrical tolerancing, symbol language and interpretation. https://www.iso.org/standard/66777.html

ISO 5459:2024 - Datums and datum systems. https://www.iso.org/standard/87855.html

ISO 2692:2021 - Maximum material, least material and reciprocity requirements. https://www.iso.org/standard/74592.html

ISO 5458:2018 - Pattern and combined geometrical specification. https://www.iso.org/standard/65559.html

ISO 1660:2017 - Profile tolerancing. https://www.iso.org/standard/63218.html

ISO 8015:2011 - GPS fundamentals, concepts, principles and rules. https://www.iso.org/standard/55979.html

ISO 14405-1:2025 - Dimensional tolerancing of linear sizes. https://www.iso.org/standard/14405-1

ISO 14253-1:2017 - Decision rules for conformity and nonconformity. https://www.iso.org/standard/70137.html