
ISO 1101.pdf

# GEOMETRIC DIMENSIONING

## FACTS TO REMEMBER

Ⓜ - MMC - MAXIMUM MATERIAL CONDITION: THAT CONDITION WHERE A FEATURE OF SIZE CONTAINS THE MAXIMUM AMOUNT OF MATERIAL WITHIN THE STATED LIMITS OF SIZE. EXAMPLE: MINIMUM HOLE SIZE AND MAXIMUM SHAFT SIZE.

Ⓛ - LMC - LEAST MATERIAL CONDITION: THAT CONDITION WHERE A FEATURE OF SIZE CONTAINS THE LEAST AMOUNT OF MATERIAL WITHIN THE STATED LIMITS OF SIZE. EXAMPLE: MAXIMUM HOLE SIZE AND MINIMUM SHAFT SIZE.

RFS - REGARDLESS OF FEATURE SIZE: THIS IS THE DEFAULT CONDITION FOR ALL GEOMETRIC TOLERANCES. NO BONUS TOLERANCES ARE ALLOWED. FUNCTIONAL GAGES MAY NOT BE USED.

Ⓟ - PROJECTED TOLERANCE ZONE: WHEN THE SYMBOL IS SHOWN, IT MEANS THE STATED TOLERANCE ZONE IS EXTENDED BEYOND THE SURFACE OF THE PART, NOT WITHIN THE PART.

(ST) - STATISTICAL TOLERANCE: A TOLERANCE FOR A PART OF AN ASSEMBLY BASED ON THE RESULTS FROM A STATISTICAL CALCULATION. THE INDICES SHOULD BE NOTED.

Ⓕ - FREE STATE: THIS SYMBOL INDICATES THE PARTS MUST NOT BE RESTRICTED DURING INSPECTION.

A DATUM SYMBOL: THIS SYMBOL IS ATTACHED TO A PLANE OR SIZE FEATURE THAT MUST BE CONTACTED FOR MACHINING, ASSEMBLY, OR INSPECTION.

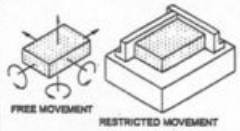
XXX BASIC DIMENSION: THESE DIMENSIONS HAVE NO TOLERANCE. THEY SIMPLY LOCATE A TOLERANCE ZONE.

Ø - DIAMETER SYMBOL: THIS SYMBOL REPLACES THE WORD "DIAMETER". IT SHOULD BE USED ANYWHERE THERE IS A DIAMETER ON THE DRAWING, AND WHEN A TOLERANCE ZONE IS CYLINDRICAL.

TOLERANCE ZONES: ALL TOLERANCE ZONES SHOWN IN THE FEATURE CONTROL FRAME ARE TOTAL. EXAMPLE: FLATNESS WITHIN .004 MEANS THAT TWO PARALLEL PLANES NO MORE THAT .004 APART DEFINE THE TOLERANCE ZONE.

A1 DATUM TARGETS: USED TO LOCATE SPECIFIC POINTS, LINES, OR AREAS ON PARTS USED FOR SUPPORT FOR PROCESSING, ASSEMBLY, AND INSPECTION. COMMONLY USED ON CASTINGS, FORGINGS, SHEET METAL PARTS, WELDMENTS AND OTHER MOLDED OR FLEXIBLE PARTS.

DATUM REFERENCE FRAME (THREE PLANE CONCEPT): THE CONCEPT OF USING A MUTUALLY PERPENDICULAR DATUM REFERENCE FRAME TO CONTROL ITS FREE MOVEMENT IN SPACE (DEGREES OF FREEDOM). SEE DATUM SYMBOL AND DATUM TARGETS.




LIMITS OF SIZE RULE: WHERE ONLY A SIZE DIMENSION IS GIVEN, a) THE SIZE DIMENSIONS AT ANY CROSS SECTION MUST BE WITHIN THE SIZE TOLERANCE, b) THE SURFACE(S) SHALL NOT EXTEND BEYOND THE PERFECT FORM DEFINED BY THE MMC SIZE, c) THE FORM MAY VARY WITHIN AN ENVELOPE BETWEEN THE MMC AND LMC.

GEOMETRIC TOLERANCE RULE: GEOMETRIC TOLERANCES ARE APPLIED RFS. IF MMC OR LMC IS REQUIRED, IT MUST BE PLACED IN REQUIRED, IT MUST BE PLACED IN THE FEATURE CONTROL FRAME. SEE MMC, LMC OR RFS COLUMN.

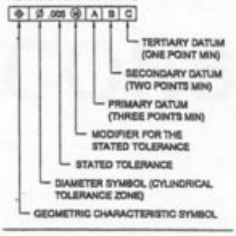
PITCH DIAMETER RULE: TOLERANCES THAT APPLY TO SCREW THREADS APPLY TO THE AXIS OF THE THREAD DERIVED FROM THE PITCH CYLINDER. IF ANOTHER PART OF THREAD IS TO BE USED TO DERIVE THE AXIS, IT MUST BE STATED BENEATH THE FEATURE CONTROL FRAME. ANY OTHER PART THAT HAS A PITCH DIAMETER MUST NOTE THE DATUM FEATURE.

## SYMBOLS, RULES, AND GUIDELINES

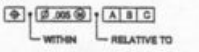
| TYPE | SYMBOL | AS SHOWN ON DRAWING | TOLERANCE ZONE | MMC LMC OR RFS | DATUM USED | FUNC GAGE USED | TOLER ZONE TYPE * |
|---|---|---|---|---|---|---|---|
| FORM | STRAIGHTNESS | Ø .500±.005 / .004 | TWO PARALLEL LINES .004 APART | Ⓜ CAN APPLY TO A FEATURE OF SIZE | NO | YES IF Ⓜ IS STATED | (A) (H) (J) |
| FORM | FLATNESS | □ .500±.005 / .004 | TWO PARALLEL PLANES .004 APART | DOES NOT APPLY | NO | NO | (D) |
| FORM | CIRCULARITY | Ø .500±.005 / .004 | TWO CONCENTRIC CIRCLES .004 APART | DOES NOT APPLY | NO | NO | (B) |
| FORM | CYLINDRICITY | Ø .500±.005 / .004 | TWO CONCENTRIC CYLINDERS .004 APART | DOES NOT APPLY | NO | NO | (E) |
| ORIENTATION | PARALLELISM | // .004 A | TWO PARALLEL PLANES .004 APART | Ⓜ CAN APPLY TO A FEATURE OF SIZE | YES | YES IF Ⓜ IS STATED | (A) (D) (G) (H) (J) |
| ORIENTATION | PERPENDICULARITY | ⊥ .004 A | TWO PARALLEL PLANES .004 APART, 90° | Ⓜ CAN APPLY TO A FEATURE OF SIZE | YES | YES IF Ⓜ IS STATED | (A) (D) (G) (H) (J) |
| ORIENTATION | ANGULARITY | ∠ .004 A, 15° | TWO PARALLEL PLANES .004 APART, 15° | Ⓜ CAN APPLY TO A FEATURE OF SIZE | YES | YES IF Ⓜ IS STATED | (A) (D) (G) (H) (J) |
| PROFILE | LINE PROFILE | ⌒ .004 | TWO LINES .004 APART ALONG TRUE PROFILE | DOES NOT APPLY | MAY BE USED OR MAY NOT | NO | (A) (B) (C) |
| PROFILE | SURFACE PROFILE | ⌓ .004 A | TWO PLANES .004 APART ALONG TRUE PROFILE A | DOES NOT APPLY | MAY BE USED OR MAY NOT | NO | (D) (E) (F) |
| RUNOUT | CIRCULAR RUNOUT | ↗ .004 A, XX | CONCENTRIC CIRCLES .004 APART, XX | RFS ALWAYS | YES | NO | (B) |
| RUNOUT | TOTAL RUNOUT | ⌰ .004 A, XX | TWO CONCENTRIC CYLINDERS .004 APART, XX | RFS ALWAYS | YES | NO | (E) |
| LOCATION | POSITION | Ø .250±.004; Ø.014 Ⓜ A B C; .38; .62 | Ø.022 ZONE AT LMC; Ø.014 ZONE AT MMC; .62; .38; TRUE CENTER | Ⓜ OR Ⓛ CAN APPLY TO A FEATURE OF SIZE | YES | YES IF Ⓜ IS STATED | (G) (H) (J) |
| LOCATION | CONCENTRICITY | ◎ Ø .004 A, XX | Ø .004 AROUND DATUM AXIS, XX | RFS ALWAYS | YES | NO | (J) |
| LOCATION | SYMMETRY | = A, XX; .004 A | XX; .004 EQUALLY DISPOSED FROM CENTER PLANE | RFS ALWAYS | YES | NO | (H) |

## FACTS TO REMEMBER

FEATURE CONTROL FRAME:




BASIC SENTENCE STRUCTURE: WHEN USING THE ENGLISH LANGUAGE TO SAY WHAT IS IN THE FEATURE CONTROL FRAME, YOU MAY USE THE FOLLOWING CONNECTING WORDS:




EXAMPLE: THE POSITION OF THE FEATURE AXIS MUST BE *WITHIN* A .005 TOLERANCE ZONE AT MMC *RELATIVE TO* DATUM FEATURES A, B, AND C.

BONUS TOLERANCE: WHEN MMC IS SHOWN MODIFYING A GEOMETRIC TOLERANCE, THE STATED TOLERANCE APPLIES ONLY WHEN THE FEATURE BEING CONTROLLED IS AT MMC. THE BONUS IS THE DIFFERENCE BETWEEN THE ACTUAL SIZE AND THE MMC SIZE AND MAY BE ADDED DIRECTLY TO THE ORIGINAL GEOMETRIC TOLERANCE

HOLE EXAMPLE:

Ø .375-.380
Ø .007 Ⓜ A

| | |
|---|---|
| ACTUAL | = .378 |
| − MMC | − .375 |
| BONUS | = .003 |

AT MMC, THE HOLE MUST BE POSITIONED WITHIN A CYLINDRICAL TOLERANCE ZONE OF .007 DIAMETER. AS THE EXAMPLE SHOWS, THE HOLE HAS DEPARTED FROM MMC BY .003. THE .003 BONUS TOLERANCE MAY NOW BE ADDED TO THE ORIGINAL .007 ZONE TO GET A TOTAL OF .010 TOLERANCE.

FUNCTIONAL GAGES: DEVICES THAT MEASURE THE COLLECTIVE EFFECTS OF SIZE AND GEOMETRIC TOLERANCES AT THE SAME TIME. IT REPRESENTS A SIMULATED MATING PART.

BONUS TOLERANCES AND FUNCTIONAL GAGES: DIRECTLY APPLICABLE TO ANY GEOMETRIC CHARACTERISTIC THAT IS MODIFIED BY Ⓜ.

SHIFT: AS A DATUM FEATURE OF SIZE, THAT IS GEOMETRICALLY CONTROLLED, DEPARTS FROM MMC, ADDITIONAL TOLERANCE MAY BE CONSIDERED FOR THE CONTROLLED FEATURES. THIS ADDITIONAL TOLERANCE DOES NOT ADD DIRECTLY TO THE ORIGINAL GEOMETRIC TOLERANCE BUT MUST BE APPLIED TO THE PATTERN OF FEATURES AS A GROUP. IT IS CONSIDERED A TOLERANCE THAT ALLOWS THE CONTROLLED FEATURES TO SHIFT AS A GROUP.

VIRTUAL CONDITION: THE COLLECTIVE EFFECT OF SIZE & FORM, OR SIZE & LOCATION THAT MUST BE CONSIDERED IN DETERMINING THE FIT OR CLEARANCE BETWEEN MATING PARTS OR FEATURES.

TO CALCULATE VIRTUAL CONDITION:

EXTERNAL FEATURES:
MMC SIZE + TOLERANCE OF FORM, ORIENTATION, OR LOCATION

INTERNAL FEATURES:
MMC SIZE - TOLERANCE OF FORM, ORIENTATION, OR LOCATION

A VIRTUAL CONDITION WILL EXIST ONLY FOR TOLERANCES THAT CONTROL SIZE FEATURES.

* THIS COLUMN REFERENCES POSSIBLE TOLERANCE ZONES THAT MAY BE USED WITH THE VARIOUS CONTROLS. THE DIFFERENT TOLERANCE ZONES ARE SHOWN ALONG THE BOTTOM OF THIS CHART. DASHED CIRCLES INDICATE THE ZONE IN THE ILLUSTRATION.

THIS CHART IS PROVIDED FOR INFORMATIONAL PURPOSES ONLY. THE INFORMATION CONFORMS TO ASME Y14.5M - 1994


multimac
6000 RADFORD ROAD, ATHENS OH 45701


ISO 1101.pdf

doctor.com/pdf-files/opp1101.pdf. Hopkin, P. (2012). ... Geneva: ISO. Retrieved June 22, 2015, from http://www.ios.org/iso/04_concept_and-use-of-the-process- ...

1dff872cbc