

# GEOMETRIC DIMENSIONING

FACTS TO REMEMBER		SYMBOLS, RULES, AND GUIDELINES				FACTS TO REMEMBER								
	TYPE	SYMBOL	AS SHOWN OR DRAWING	TOLERANCE ZONE	MMC/LMC OR RFS	FUNCTION USED	TOLER ZONE TYPE							
<p><b>MAX</b> - MAXIMUM MATERIAL CONDITION THAT CONDITION WHERE A FEATURE OF SIZE CONTAINS THE MAXIMUM ALLOWED SIZE. EXAMPLE: MAXIMUM HOLE SIZE AND MAXIMUM SHIRT SIZE.</p> <p><b>LAC</b> - LEAST MATERIAL CONDITION THAT CONDITION WHERE A FEATURE OF SIZE CONTAINS THE LEAST AMOUNT OF MATERIAL. WITHIN THE STATED LIMITS OF SIZE. EXAMPLE: MINIMUM HOLE SIZE AND MINIMUM SHIRT SIZE.</p> <p><b>RFS</b> - REGARDLESS OF FEATURE SIZE THIS IS THE DEFAULT CONDITION FOR ALL GEOMETRIC TOLERANCES. NO BONUS TOLERANCES ARE ALLOWED. FUNCTIONAL GAGES MAY NOT BE USED.</p> <p><b>PROJECTED TOLERANCE ZONE</b> - WHEN THE SYMBOL IS SHOWN, IT MEANS THE STATED TOLERANCE ZONE EXTENDS BEYOND THE SURFACE OF THE PART, NOT WITHIN THE PART.</p> <p><b>STATISTICAL TOLERANCE</b> - A TOLERANCE FOR PARTS FROM AN ASSEMBLY BASED ON THE RESULTS FROM A STATISTICAL ANALYSIS OF THE RESULT TO LARGER TOLERANCES.</p> <p><b>FREE STATE</b> - THIS SYMBOL INDICATES THE PARTS MUST NOT BE RESTRICTED DURING INSPECTION.</p> <p><b>DATUM SYMBOL</b> - THIS SYMBOL IS ATTACHED TO A FEATURE THAT MUST BE CONTACTED FOR MACHINING AND INSPECTION.</p> <p><b>MFC</b> - BASIC DIMENSION: THESE DIMENSIONS HAVE NO TOLERANCE. THEY ONLY LOCATE A TOLERANCE ZONE.</p> <p><b>DIAMETER SYMBOL</b> - THIS SYMBOL REPLACES THE WORD "DIAMETER" - IT SHOULD BE USED ANYWHERE THERE IS A DIAMETER ON THE DRAWING, AND WHEN A TOLERANCE ZONE IS TO BE INDICATED.</p> <p><b>TOLERANCE ZONES</b> - ALL TOLERANCE ZONES SHOWN IN THE FEATURE CONTROL FRAME ARE TOTAL. EXAMPLE: FLATNESS WITHIN OR MEANS THAT TWO PARALLEL PLANES NO MORE THAN .004 IN. DEFINE THE TOLERANCE ZONE.</p> <p><b>DATUM TARGETS</b> - USED TO LOCATE SPECIFIC POINTS, LINES, OR AREAS ON PARTS USED FOR SUPPORT AND MACHINING AND INSPECTION. COMMERCIALY USED ON BRIG-PARTS LIKE CASTINGS AND GEOMETRIC AND NON-GEOMETRIC PARTS MADE FROM PLASTIC, RUBBER, OR SHEET METAL.</p> <p><b>DATUM REFERENCE FRAME</b> - (THREE PLANE CONCEPT) - THE CONCEPT OF USING A PART TO CONTROL ITS FREE MOVEMENT IN SPACE DETERMINES THE REFERENCE FRAME SYMBOL AND DATUM TARGETS.</p> <p><b>CHARACTERISTICS OF FREE MOVEMENT</b></p> <p><b>RESTRICTED MOVEMENT</b></p> <p>LIMITS OF SIZE RULE: WHERE ONLY A SIZE DIMENSION IS GIVEN, AS THE SIZE DIMENSIONS AT ANY CROSS SECTION MUST BE WITHIN THE SIZE TOLERANCE, THE SURFACES SHALL NOT EXTEND BEYOND THE TOLERANCE FORM DEFINED BY THE MMC SIZE. (IF THE FORM MAY VARY WITHIN AN ENVELOPE BETWEEN THE MMC AND LMC.)</p> <p><b>GEOMETRIC TOLERANCE RULE</b> - GEOMETRIC TOLERANCES ARE UNDERSTOOD TO BE APPLIED TO A SURFACE UNLESS REQUIRED, IT MUST BE PLACED IN THE FEATURE CONTROL FRAME. (SEE MMC, LMC, OR RFS COLUMN.)</p> <p><b>PITCH DIAMETER RULE</b> - TOLERANCES THAT APPLY TO TOOTH THREADS APPLY TO THE ARCS OF THE THREAD DERIVED FROM THE PITCH CIRCLES IF ANOTHER PART OF THE THREAD IS TO BE USED TO DEFINE THE PITCH CIRCLES. (SEE DATUM SYMBOL.)</p> <p><b>SYMMETRY</b> - THIS SYMBOL INDICATES THE FEATURE CONTROL FRAME ANY OTHER FEATURES THAT HAVE A PITCH CHARACTER MUST HAVE THE DATUM FEATURE STATED.</p>	<p><b>STRAIGHTNESS</b></p>	<p><b>FLATNESS</b></p>	<p><b>CIRCULARITY</b></p>	<p><b>CYLINDRICITY</b></p>	<p><b>PARALLELISM</b></p>	<p><b>PERPENDICULARITY</b></p>	<p><b>ANGULARITY</b></p>	<p><b>PROFILE OF A LINE</b></p>	<p><b>PROFILE OF A SURFACE</b></p>	<p><b>CIRCULAR RUNOUT</b></p>	<p><b>TOTAL RUNOUT</b></p>	<p><b>POSSIBILITY</b></p>	<p><b>CONCENTRICITY</b></p>	<p><b>SYMMETRY</b></p>
	<p><b>FORM</b></p>		<p><b>ORIENTATION</b></p>		<p><b>PROFILE</b></p>		<p><b>RUNOUT</b></p>		<p><b>LOCATION</b></p>					
	<p><b>FEATURE CONTROL FRAME</b></p> <p>BASIC SENTENCE STRUCTURE: WHEN USING THE ENGLISH LANGUAGE TO SAY WHAT THE FEATURE CONTROL FRAME YOU MAY USE THE FOLLOWING CONNECTING WORDS:</p> <p>RELATIVE TO WITHIN</p> <p>EXAMPLE: THE POSITION OF THE FEATURE SIZE MUST BE WITHIN A .005 TOLERANCE ZONE AT MMC RELATIVE TO DATUM FEATURES A, B, AND C.</p> <p>BONUS TOLERANCE: WHEN MMC IS SHOWN ADOPTING A PARTICULAR TOLERANCE TO THE TOLERANCE ZONE APPLIES ONLY WHEN THE FEATURE BEING CONTROLLED AT MMC. THE BONUS IS THE DIFFERENCE BETWEEN THE ACTUAL SIZE AND THE MMC SIZE AND MAY BE ADDED DIRECTLY TO THE ORIGINAL TOLERANCE.</p> <p>EXAMPLE: <math>\text{M} \begin{matrix} .005 \\ .004 \end{matrix} \text{B} \begin{matrix} .005 \\ .004 \end{matrix} \text{C}</math></p> <p>AT MMC THE HOLE MUST BE POSITIONED WITHIN A .004 ORIGINAL TOLERANCE ZONE OF SIZE (DIAMETER) AS THE EXAMPLE SHOWS. THE HOLE HAS DEFINED BONUS TOLERANCE. THE .003 BONUS TOLERANCE MAY NOW BE ADDED TO THE ORIGINAL .005 ZONE FOR A TOTAL OF .008 TOLERANCE.</p> <p>FUNCTIONAL GAGES: DEVICES THAT MEASURE THE COLLECTIVE EFFECTS OF SIZE AND GEOMETRIC TOLERANCES AT THE SAME TIME. IT REPRESENTS A SIMPLIFIED MACHINING CONDITION.</p> <p>BONUS TOLERANCES AND FUNCTIONAL GAGES: DIRECTLY APPLICABLE TO ANY GEOMETRIC CHARACTERISTIC THAT IS MODELED BY <math>\text{M}</math>.</p> <p>SHIFT AS A DATUM FEATURE OF SIZE: THIS IS GEOMETRICALLY CONTROLLED. DERIVED FROM BONUS. ADDITIONAL TOLERANCE MUST BE CONSIDERED FOR THE CONTROLLED FEATURES. THIS ADDITIONAL 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.</p> <p>VERTICAL CONDITION: THE COLLECTIVE EFFECT OF SIZE AND GEOMETRIC TOLERANCES THAT MUST BE CONSIDERED IN DETERMINING THE FIT OR CLEARANCE BETWEEN MATING PARTS OR FEATURES.</p> <p>TO CALCULATE VERTICAL CONDITIONS:</p> <p>EXTERNAL FEATURES: MMC SIZE - TOLERANCE OF FORM, ORIENTATION, OR LOCATION.</p> <p>INTERNAL FEATURES: MMC SIZE - TOLERANCE OF FORM, ORIENTATION, OR LOCATION.</p> <p>A VERTICAL CONDITION WILL FIRST ONLY FOR TOLERANCES THAT CONTROL SIZE FEATURES.</p> <p>* THIS COLUMN INDICATES POSSIBLE TOLERANCE ZONES THAT MAY BE USED WITH THE VARIOUS CONDITIONS. THE DIFFERENT TOLERANCE ZONES ARE SHOWN ALONG THE BOTTOM OF THIS CHART.</p> <p>THIS CHART IS PROVIDED FOR INFORMATIONAL PURPOSES ONLY. THE INFORMATION IS BASED UPON ASME Y14.5M-2014.</p>		<p><b>FORM</b></p> <p>MMC/LMC OR RFS</p> <p>FUNCTION USED</p> <p>TOLER ZONE TYPE</p> <p>RELATIVE TO WITHIN</p> <p>EXAMPLE: THE POSITION OF THE FEATURE SIZE MUST BE WITHIN A .005 TOLERANCE ZONE AT MMC RELATIVE TO DATUM FEATURES A, B, AND C.</p> <p>BONUS TOLERANCE: WHEN MMC IS SHOWN ADOPTING A PARTICULAR TOLERANCE TO THE TOLERANCE ZONE APPLIES ONLY WHEN THE FEATURE BEING CONTROLLED AT MMC. THE BONUS IS THE DIFFERENCE BETWEEN THE ACTUAL SIZE AND THE MMC SIZE AND MAY BE ADDED DIRECTLY TO THE ORIGINAL TOLERANCE.</p> <p>EXAMPLE: <math>\text{M} \begin{matrix} .005 \\ .004 \end{matrix} \text{B} \begin{matrix} .005 \\ .004 \end{matrix} \text{C}</math></p> <p>AT MMC THE HOLE MUST BE POSITIONED WITHIN A .004 ORIGINAL TOLERANCE ZONE OF SIZE (DIAMETER) AS THE EXAMPLE SHOWS. THE HOLE HAS DEFINED BONUS TOLERANCE. THE .003 BONUS TOLERANCE MAY NOW BE ADDED TO THE ORIGINAL .005 ZONE FOR A TOTAL OF .008 TOLERANCE.</p> <p>FUNCTIONAL GAGES: DEVICES THAT MEASURE THE COLLECTIVE EFFECTS OF SIZE AND GEOMETRIC TOLERANCES AT THE SAME TIME. IT REPRESENTS A SIMPLIFIED MACHINING CONDITION.</p> <p>BONUS TOLERANCES AND FUNCTIONAL GAGES: DIRECTLY APPLICABLE TO ANY GEOMETRIC CHARACTERISTIC THAT IS MODELED BY <math>\text{M}</math>.</p> <p>SHIFT AS A DATUM FEATURE OF SIZE: THIS IS GEOMETRICALLY CONTROLLED. DERIVED FROM BONUS. ADDITIONAL TOLERANCE MUST BE CONSIDERED FOR THE CONTROLLED FEATURES. THIS ADDITIONAL 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.</p> <p>VERTICAL CONDITION: THE COLLECTIVE EFFECT OF SIZE AND GEOMETRIC TOLERANCES THAT MUST BE CONSIDERED IN DETERMINING THE FIT OR CLEARANCE BETWEEN MATING PARTS OR FEATURES.</p> <p>TO CALCULATE VERTICAL CONDITIONS:</p> <p>EXTERNAL FEATURES: MMC SIZE - TOLERANCE OF FORM, ORIENTATION, OR LOCATION.</p> <p>INTERNAL FEATURES: MMC SIZE - TOLERANCE OF FORM, ORIENTATION, OR LOCATION.</p> <p>A VERTICAL CONDITION WILL FIRST ONLY FOR TOLERANCES THAT CONTROL SIZE FEATURES.</p> <p>* THIS COLUMN INDICATES POSSIBLE TOLERANCE ZONES THAT MAY BE USED WITH THE VARIOUS CONDITIONS. THE DIFFERENT TOLERANCE ZONES ARE SHOWN ALONG THE BOTTOM OF THIS CHART.</p> <p>THIS CHART IS PROVIDED FOR INFORMATIONAL PURPOSES ONLY. THE INFORMATION IS BASED UPON ASME Y14.5M-2014.</p>											



GENIUM PUBLISHING COMPANY  
ONE GENIUM PLAZA  
SCENIC RIDGE, NY 12584  
(518) 577-8824