# GD&T APPLICATIONS 312

## Class Vocabulary

| Term | Definition |
|---|---|
| 16 UN | 16 Unified National Fine. A standard diameter-pitch combination with fine-pitch threads and 16 threads per inch. The 16 UN series is commonly used for fine-threaded bolts with diameters over 1 3/4 inch. |
| 3D | Three-dimensional. Having a length, width, and depth. Nearly all physical objects are 3D. |
| Actual Mating Envelope | AME. A geometrically perfect shape that is an imaginary shape that best fits around a feature. Actual mating envelope can be established by inspecting a feature's height with a functional gage, as well as inspecting the feature's cross-sectional dimensions. |
| American Society Of Mechanical Engineers | ASME. An organization that publishes technical materials and sets industrial and manufacturing standards. The American Society of Mechanical Engineers provides written standardization for GD&T in ASME Y14.5-2018. |
| Angularity | A three-dimensional tolerance that describes the allowable variability in the angular relationship between a surface and a datum. Angularity is a type of orientation tolerance. |
| ASME Y14.5-2018 | The standard for Geometric Dimensioning and Tolerancing published by the American Society of Mechanical Engineers (ASME). The 2018 revision is the most recent version of the standard. |
| Axis | A theoretical straight line that lies in the center of an object. When a cylindrical surface forms a feature of size in GD&T, it establishes the axis of that cylinder. |
| Basic Dimension | A dimension that is theoretically perfect. A basic dimension has no direct tolerance and is denoted on a GD&T blueprint as a number enclosed in a rectangular box. |
| Bonus Tolerance | Additional tolerance that applies to a feature as its size shifts from a stated material condition. Bonus tolerance is permissible as a hole deviates from MMC to LMC. |
| Calipers | A measuring instrument with a pair of jaws on one end and a long beam containing a marked scale of unit divisions. Caliper jaws can measure both internal and external features. |
| Centerpoint | A single imaginary point located in the center and an equal distance from the exterior of a circular feature. When a spherical surface forms a feature of size, it establishes the centerpoint of that sphere. |
| Chuck | A device that holds a workpiece in place as it rotates on a lathe or other machine tool. Chucks commonly have three or four jaws that can be adjusted to fit various workpieces. |
| Circular Runout | A two-dimensional geometric tolerance that controls the form, orientation, and location of multiple cross sections of a cylindrical part as it rotates. Circular runout is one of two types of runout tolerances. |
| Circularity | A two-dimensional tolerance that describes the allowable variability in the shape and appearance of a circle. Circularity, also known as roundness, is a type of form tolerance. |
| Clearance | An amount of space or distance between two objects or areas. Clearance helps ensure two parts or components can move freely or ensures there is not contact between two surfaces. |
| CMM | Coordinate measuring machine. A sophisticated measuring instrument with a flat polished table and a suspended probe that measures parts in three-dimensional space. CMMs can measure using either contact or noncontact methods. |
| Coaxial | Two round or square features or patterns of features, like holes, that share the same center line or axis on the part. A pattern of identical holes positioned the same distance from an axis and spread equally around it are considered coaxial to the axis. |
| Collet | A slotted device that holds a workpiece in place as it rotates. A collet has a hole through which the workpiece passes and is designed to hold specific dimensions. |
| Computer-Aided Design | CAD. A computer software program that aids in the automated design and technical precision drawing of a part, product, process, or building. A Computer-aided design model can be used like a part drawing. |

| | |
|---|---|
| **Coordinate Measuring Machine** | CMM. A sophisticated measuring instrument with a flat polished table and a suspended probe that measures parts in three-dimensional space. Coordinate measuring machines can measure using either contact or noncontact methods. |
| **Counterbore** | A flat-bottomed recess machined into the opening of an existing hole. A counterbore provides space for a flat-faced component or fastener to rest inside a portion of an existing hole. |
| **Cylindricity** | A three-dimensional form tolerance that describes the allowable variability in the shape and appearance of a cylinder. Cylindricity defines how close a cylindrical feature must be to be perfectly cylindrical relative to a datum axis. |
| **Datum Feature Simulators** | A gaging surface, machine component, or other device that establishes a datum plane in the datum reference frame. Datum feature simulators adhere to tolerances and act as a reasonable substitute for the datum reference frame. |
| **Datum Features** | A physical feature that acts as an acceptable substitute for a datum. Datum features relate the various features of the part to each other. |
| **Datum Reference Frame** | DRF. Three theoretical planes perpendicular to one another that are mapped onto the part. The datum reference frame provides an anchor for relating part features to each other. |
| **Datums** | A point of reference for machine tools, programs, and fixtures from which measurements are taken. A datum can be a hole, line, or any three-dimensional shape. |
| **Depth Micrometer** | A measuring instrument that aligns with the top of a recess and measures a hole with a shaft and measuring rod that are lowered into the hole. Depth micrometers are used to measure the depth of a hole, as opposed to a diameter like other micrometers. |
| **Dial Indicator** | A measuring instrument with a contact point attached to a spindle and gears that move a pointer on the dial. Dial indicators have graduations that show different measurement values. |
| **Drawing Block** | The area of a blueprint containing information such as company name, part name, part number, designer, scale, material, and tolerances for dimensions without specified size limits. Drawing blocks, or title blocks, are unique to each manufacturer. |
| **FCF** | Feature control frame. A series of compartments containing symbols and values that describe the geometric tolerance of a feature. The order and purpose of FCF compartments follow a consistent standard. |
| **Feature** | A defining physical characteristic. Features include shapes, lines, and other elements machined into a workpiece. |
| **Feature Control Frame** | FCF. A series of compartments containing symbols and values that describe the geometric tolerance of a feature. The order and purpose of feature control frame compartments follow a consistent standard. |
| **Feature Of Size** | FOS. A cylindrical surface, spherical surface, or two opposed parallel elements or surfaces that can be associated with a size dimension. A feature of size establishes an axis, median plane, or centerpoint. |
| **Fixed CMMs** | A stationary coordinate measuring machine. Fixed CMMs have a large base with a suspended probe and are operated in a controlled environment. |
| **Flange** | A ring or collar surrounding a cylindrical component such as a shaft or toolholder. A flange is mounted to other components to provide support and rigidity. |
| **Flatness** | A three-dimensional tolerance that describes the allowable variability in the shape and appearance of a surface that lies in a plane. Flatness is a type of form tolerance. |
| **Form Tolerances** | A geometric tolerance that limits the amount of acceptable error in the shape of a feature. The form tolerances include straightness, flatness, circularity, and cylindricity. |
| **FOS** | Feature of size. A cylindrical surface, spherical surface, or two opposed parallel elements or surfaces that can be associated with a size dimension. An FOS establishes an axis, median plane, or centerpoint. |
| **Functional Gages** | A gage representing a worst-case mating part that provides a simple pass/fail assessment of the inspected part. Functional gages often can quickly inspect several features at once. |
| **GD&T** | Geometric dimensioning and tolerancing. An international standard for communicating instructions about the design and manufacturing of parts. GD&T uses universal symbols and emphasizes the function of the part. |
| **Geometric** | The linear and curved shapes that characterize an object. Geometric features of a part determine its assembly and functionality. |
| **Geometric Control Symbols** | A set of 12 symbols used to describe tolerances for geometric characteristics of part features. Geometric control symbols are categorized in five groups by tolerance type: form, profile, orientation, location, and runout. |

| | |
|---|---|
| **Geometric Dimensioning And Tolerancing** | GD&T. An international standard for communicating instructions about the design and manufacturing of parts. Geometric Dimensioning and Tolerancing uses universal symbols and emphasizes the function of the part. |
| **Go/No-Go Gage** | An instrument with an established standard size that determines whether a part feature passes or fails inspection. Go/no-go gages do not determine the degree of variation. |
| **ID** | Inside diameter. The diameter measurement of a hole's interior surface. An inside diameter measurement extends the entire length of the hole |
| **Inside Diameter** | ID. The diameter measurement of a hole's interior surface. An inside diameter measurement extends the entire length of the hole. |
| **Inside Diameter Grinder** | A machine that uses an abrasive to machine an internal workpiece surface and achieve highly accurate measurements. Inside diameter grinders commonly use abrasive grains bonded into a wheel to shape internal features, such as the surface of a hole. |
| **Least Material Condition** | LMC. The point at which a feature contains the least amount of material within its acceptable size limit. Least material condition is the largest acceptable hole and the smallest acceptable shaft, for example. |
| **Location Tolerances** | A geometric tolerance that limits the location or placement of features relative to a datum or other features. Location tolerances include position, concentricity, and symmetry. |
| **Major Diameter** | The largest diameter feature on a thread. Major diameter is the diameter from crest to crest of an external thread or root to root of an internal thread. |
| **Material Condition Modifiers** | Defines the tolerance of a feature in relation to its acceptable size limits. There are three material condition modifiers used in GD&T: maximum material condition, least material condition, and regardless of feature size. |
| **Maximum Material Condition** | MMC. The point at which a feature contains the greatest amount of material within its acceptable size limit. Maximum material condition is the smallest acceptable hole and the largest acceptable shaft, for example. |
| **Median Plane** | An imaginary, perfectly flat plane positioned in the middle between two opposing flat surfaces. Median planes are established by surfaces that form a feature of size in GD&T, establishing the median plane between them. |
| **Minor Diameter** | The smallest diameter on a thread. Minor diameter is the diameter from root to root of an external thread or crest to crest of an internal thread. |
| **MMC** | Maximum material condition. The point at which a feature contains the greatest amount of material within its acceptable size limit. The smallest acceptable hole and the largest acceptable shaft are examples of MMC. |
| **Normal** | A line that is perpendicular to the tangent plane of a surface at the point of contact with that surface. Normal measurements to the true profile are required to inspect profile tolerances. |
| **OD** | Outside diameter. The diameter measurement of a cylindrical feature's exterior surface. An OD measurement extends the entire length of the surface. |
| **Orientation Tolerances** | A geometric tolerance that limits the direction, or orientation, of a feature in relation to other features. Orientation tolerances include angularity, perpendicularity, and parallelism. |
| **Outside Diameter** | OD. The diameter measurement of a cylindrical feature's exterior surface. An outside diameter measurement extends the entire length of the surface. |
| **Parallelism** | A three-dimensional tolerance that describes the equal distance between pairs of points, lines, or planes. Parallelism is a type of orientation tolerance. |
| **Perpendicular** | Two lines, planes, features, or objects that exist at a right angle to one another. Perpendicular lines intersect one another at a 90° angle. |
| **Perpendicularity** | A three-dimensional tolerance that describes the allowable variation of a feature from exactly 90° relative to a surface, plane, or axis. Perpendicularity is a type of orientation tolerance. |
| **Pin** | A narrow, non-threaded, cylindrical component. Pins are used for fastening. |
| **Pitch Diameter** | An identifying dimension of threaded objects or components. Pitch diameter is a theoretical dimension halfway between major and minor diameter. |
| **Pitch Diameter Rule** | A GD&T rule that states tolerances and datum references apply to the pitch diameter of threaded components to ensure assembly with other threaded components. The pitch diameter rule does not apply in cases where major diameter or minor diameter is noted beneath the feature control frame. |
| **Portable CMMs** | A coordinate measuring machine that can be easily carried or moved. Portable CMMs can be used on the factory floor. |

| | |
|---|---|
| **Position** | A three-dimensional tolerance that controls how much the location of a feature can deviate from its true position. Position is a type of location tolerance. |
| **Primary Datum** | The datum feature that first situates the part within the datum reference frame. The primary datum is the first feature to contact a fixture or surface during assembly. |
| **Primary Datum Reference** | The main datum attributed to the tolerance, or control. The primary datum reference, represented by a capital letter associated with a datum feature, appears in the third compartment of a feature control frame. |
| **Print** | A document containing all the instructions necessary to manufacture a part. A print, or blueprint, includes a part drawing, dimensions, and notes. |
| **Profile Of A Line** | A two-dimensional tolerance that describes the allowable variability in the contour of the edge seen in the section view. Profile of a line is one of two types of profile tolerances. |
| **Profile Of A Surface** | A three-dimensional profile tolerance that describes the allowable variation in the contour of a surface. Profile of a surface tolerance limits how much a feature surface deviates from its appearance in the part drawing. |
| **Profile Tolerances** | A geometric tolerance that controls the size, location, orientation, and form of a feature. Profile tolerances include profile of a line and profile of a surface. |
| **Radial Distance** | A measurement perpendicular to an axis. The radial distance between two cylinders is the shape of the tolerance zone for a total runout control. |
| **Regardless Of Feature Size** | RFS. A modifier indicating that the stated tolerance for a feature applies regardless of its actual size within an acceptable size limit. Regardless of feature size does not permit bonus tolerance. |
| **RFS** | Regardless of feature size. A modifier indicating that the stated tolerance for a feature applies regardless of its actual size within an acceptable size limit. RFS does not permit bonus tolerance. |
| **Rule #1** | A rule stating that, when a tolerance for a feature of size is specified, the surfaces of that feature cannot extend beyond its boundary of perfect form at its maximum material condition. Rule #1, or the envelope princple, also states that as the material of the feature of size moves from MMC to LMC, additional variation is allowed. |
| **Rule #2** | The GD&T rule stating that, for all applicable geometric tolerances, the regardless of feature size modifier applies to the individual tolerance, datum reference, or both where no modifier symbol is specified. Rule #2 addresses how features and datums relate to one another. |
| **Runout Tolerances** | A geometric tolerance that simultaneously limits the form, location, and orientation of cylindrical parts. Runout tolerances include circular runout and total runout. |
| **Secondary Datum** | The datum feature that situates the part within the datum reference frame after the primary datum. The secondary datum is the second feature to contact a fixture or surface during assembly. |
| **Size Dimension** | A linear measurement across two opposing points, lines, planes, or surfaces. A size dimension is a required characteristic of a feature of size. |
| **Size Limits** | Exact boundaries that represent the maximum and minimum allowable size of a feature. Size limits set a range of acceptable size dimensions for a part between maximum material condition and least material condition. |
| **Straightness** | A two-dimensional tolerance that describes allowable variation in the shape and appearance of a line from perfectly straight. Straightness is a type of form tolerance. |
| **Telescope Gage** | A measuring instrument with a spring-loaded device used to check the inner diameter of a part. Telescope gages are used with a micrometer to get the proper value. |
| **Tertiary Datum** | The datum feature that situates the part within the datum reference frame after the secondary datum. The tertiary datum plane must be perpendicular to both the primary and secondary planes and is usually the smallest surface of the workpiece. |
| **Three-Dimensional** | 3D. Having length, depth, and width. Nearly all physical objects are three-dimensional. |
| **Through Hole** | A hole that begins on one side of an object and exits on another side, passing completely through the object.Through holes have no bottom because they pass through both sides of a workpiece. |
| **Title Block** | The area of a blueprint containing information such as company name, part name, part number, designer, scale, material, and tolerances for dimensions without specified size limits. Title blocks, or drawing blocks, are unique to each manufacturer. |

| | |
|---|---|
| **Tolerance Zone** | An imaginary zone in which a part feature must be completely contained for the part to pass inspection. The tolerance zone contains the dimensions between the maximum and minimum limits of a feature's location. |
| **Tolerances** | The amount of permitted variation from a specified dimension of a part. A part within tolerances will meet specifications and pass inspection. |
| **Total Runout** | A three-dimensional geometric tolerance that controls the form, orientation, and location of the entire length of a cylindrical part as it rotates. Total runout is a related tolerance. |
| **Total Runout** | A three-dimensional tolerance that controls the form, orientation, and location of the entire length of a cylindrical part as it rotates. Total runout is one of two types of runout tolerances. |
| **True Position** | The imaginary perfect position of a feature of size as defined by coordinates and dimensions of a part drawing. True position serves as the reference location from which a feature may deviate with a position tolerance. |
| **True Position** | The theoretical perfect position of a feature of size as defined by coordinates and dimensions of a part drawing. True position serves as the reference location from which a feature may deviate with a position tolerance. |
| **True Profile** | The perfect, imaginary profile described by the design specifications. The profile tolerances compare the actual profile of a feature to the true profile. |
| **Two-Dimensional** | 2D. Having a length and width, but not depth. Flat shapes are two-dimensional. |
| **V-Blocks** | A workholding component that is either magnetic or laminated. V-blocks are designed for grinding angles and holding round, square, or rectangular workpieces. |