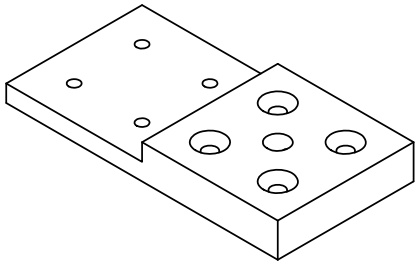
EML2322L – MAE Design and Manufacturing Laboratory

GD&T Shaft Clamp Bracket

Geometric Dimensioning and Tolerancing (GD&T) is a dimensioning standard with (purportedly) no ambiguity: every drawing can yield only a single part. The converse however, is generally not true: for a given desired part there are usually multiple ways to dimension and tolerance the drawing. As engineers, regardless of whether we personally create part drawings or sign off on drawings, we need to be capable of interpreting a GD&T drawing.

A GD&T drawing requires notably more effort to create compared to a traditionally dimensioned and toleranced drawing (TD&T). Therefore, understanding when GD&T or TD&T is appropriate may save significant time and money. TD&T is better suited for inter-office documents where the drawing's purpose is to communicate ideas (not exact part specifications) or for parts that will be manufactured in-house. Often TD&T is used for prototype or low quantity parts (again, in-house). GD&T is well suited for parts that will be mass produced, parts which have important geometric tolerance requirements, parts which will be manufactured by another facility (whether or not it is owned by the same company), or parts which have features that cannot be clearly communicated via TD&T.

The following pages present and explain the shaft clamp bracket using GD&T. Note the tolerances and specifications for both the GD&T and TD&T drawings are closely matched.



A **NOTES:**
MAT'L: 6061-T6 ALUMINUM
QTY: 1 (NOT NORMALLY SHOWN ON DETAIL DRAWING)

SolidWorks Student License
Academic Use Only

SPECIFICATIONS UNLESS NOTED	TOLERANCE UNLESS NOTED			TITLE: SHAFT CLAMP BRACKET
DRAWINGS ARE CREATED ACCORDING TO ASME Y14.5-2009 D	OPERATION	PLACES IN DIMENSION .0	.00	.000	DRAWN J. DERSCH
DIMS IN INCHES BREAK ALL EDGES HOLE-DEPTH MEASURED BY FULL DIAMETER C	MACHINING	±.050	±.020	±.005	DESIGNED J. KOCH
	CUT OFF (SAW, BURN, SHEAR)	±.1	±.060		SIZE A / DWG. NO. EML2322L-010 / REV B
	WELDING	±.1	±.060		
E THIRD ANGLE PROJECTION	ANGULAR DIMS	±5	±2	±.5	SCALE: 1:1 / SHEET 1 OF 1

4 3 2 1

DRAWING NOTES:

A) ***Material*** and ***quantity*** are specified the same way on both the TD&T and GD&T drawings. Note quantity is generally ONLY specified in the BOM and not on detail drawings. In EML2322L, quantity was included on detail drawings to simplify manufacturing.

B) ***Units*** and ***debur notes*** are provided on both the TD&T and GD&T drawings.

C) How **hole-depths** are measured should be specified; two common methods are to measure the depth with respect to the full diameter or the tapered point.

D) The **GD&T drawing standard** according to which the drawing was created should always be provided to ensure a single interpretation of the drawing.

E) ***Drawing projection*** informs the reader of the drawing how views are projected. Third angle projection is standard for much of North America.

Surface Finish

F) These surfaces have a flatness specification of .003, which requires a finished surface. (The surface finish specification on the TD&T drawing is intended to emulate the flatness specification provided here on the GD&T drawing.)
   * The symbol (A ←→ B) indicates the associated feature control frames (FCF) apply from A (upper left corner of part) to B (upper right corner of part). The A and B here are references and have no association with Datums A or B.

G) This surface has a perpendicularity specification of .003. As a result, the surface must fall within an envelope that is .003" thick and perpendicular to Datum A. This tight perpendicularity requirement matches the intent on the TD&T drawing. (The TD&T drawing assumes perpendicular faces and requires a surface finish.)

DIMENSIONS & TOLERANCES:

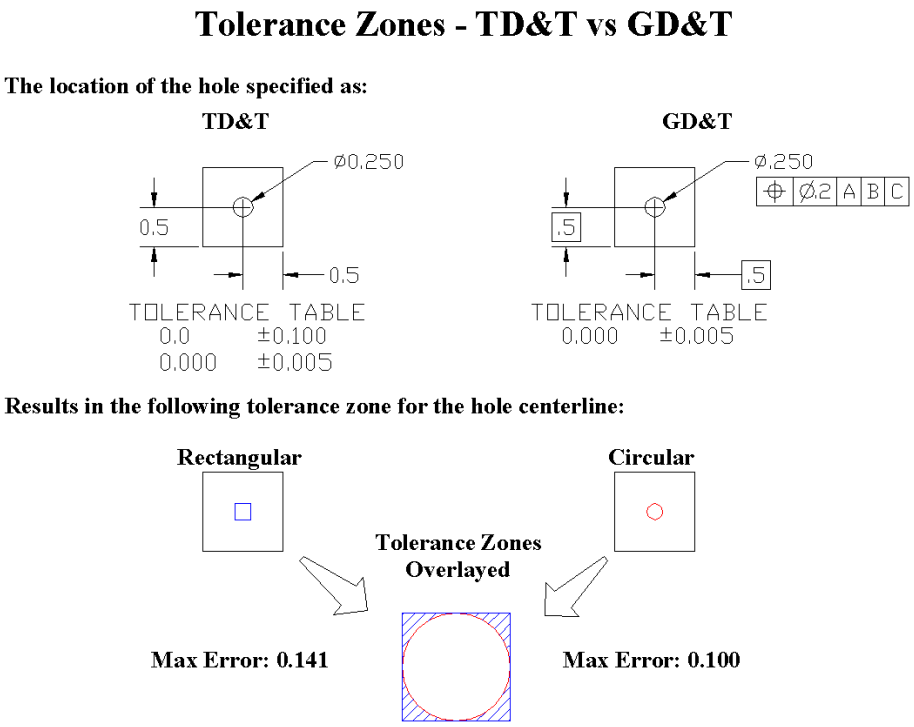
General Note: Basic dimensions (dimensions with a box around them) are used to locate most features on this part. Basic dimensions indicate the "perfect" dimension and have no associated tolerance yet every feature REQUIRES a tolerance. Therefore, if a feature is located by a basic dimension, a FCF will add additional tolerance information.

H) This dimension is non-basic and therefore has a tolerance. In this case, the width of the part is 2.00±.020". This means the located surface must fall within a ±.020" window centered exactly at 2.00" and parallel to Datum A as in the TD&T drawing.

I) The remaining surfaces that must be located are located by this FCF. It specifies the surfaces (from A to B – see *) fall in a .040" window centered about the perfect surface location. A .040" window is equivalent to ±.020".
   * This symbol (A ←→ B) indicates that the associated feature control frames (FCF) apply from A (upper left corner of part) to B (upper right corner of part). The A and B here are references and have no association with Datums A or B.

HOLES:

General Note: In this drawing basic dimensions are exclusively used to locate hole positions, which allows hole tolerance zones to be defined by FCFs and to be circular rather than rectangular (illustrated below).



J) Hole callouts in the GD&T drawing are identical to those in the TD&T drawing. (Note that the TD&T's 10-24 UNC holes are further defined here as 10-24 UNC-2B. This additional information is merely indicating the thread class and should be provided regardless of TD&T or GD&T.)

K) This position requirement states the centerline of the hole must completely fall within a .04" diameter cylinder located as defined by the basic dimensions. Note a .04" diameter tolerance zone is *roughly* equivalent to a ±.02" tolerance on location provided in a TD&T drawing (see above illustration).

L) This position requirement states the centerline of the hole must completely fall within a .01" diameter cylinder located as defined by the basic dimensions. Note a .01" diameter tolerance zone is *roughly* equivalent to a ±.005" tolerance on location provided in a TD&T drawing (see above illustration).

* The Ⓥ modifies the tolerances in a special way: the tolerance requirement is loosened without reducing the quality of this hole's function. The result is that regardless of where the hole's position and size falls (within tolerance), there is guaranteed to be a minimum amount of clearance. This condition is often used when mating clearance holes with existing bolt patterns to ensure clearance. See discussion below for more details.