SPECIFICATION OF ENGINEERING DRAWING AND RELATED DOCUMENTATION PRACTICES

APPROVAL *	DATE MO/DAY/YR
Written by (Mechanical Engineer) **Gordon W. Cumming**	**6/5/14**
Approved by (Mgr. Mech.Design) **Christian L. Lott**	**6/9/14**
Approved by (Mgr. Manufacturing) **Brian L. Groves**	**6/10/14**
Approved by (Quality) **Russell D. Lund**	**6/16/14**
Approved by (Supply Chain Eng.) **Craig S. Archuleta**	**6/10/14**
Approved by (Dir. Hardware Eng.) **Douglas R. Mangum**	**6/12/14**
Approved by (Dir. Manufacturing) **Roger T. Su**	**6/9/14**

* Official approvals are contained in Teamcenter

L-3 Communications
Communication Systems-West
640 North 2200 West
P.O. Box 16850
Salt Lake City, UT 84116

DOCUMENT NUMBER:

1000383796

CAGE CODE:	REV.:
06401	**C**

This page intentionally left blank

Table of Contents

1. Purpose

The purpose of this document is to define and communicate L-3 CSW engineering drawing and related documentation practices. These practices are based on accepted ASME and other standards. Should there be any conflict between the text of this document and other cited standards, the text of this document shall take precedence. Included items define L-3 CSW practices where cited standards provide discretion, lack clarity or L-3 CSW has a compelling need to deviate. This Standard also establishes some default tolerances that are applicable where explicit tolerances are not given.

2. Scope

This Standard establishes uniform practices used at L-3 CSW for stating and interpreting product definition data for use on drawings, master data sets and in related documents. It is applicable to all CSW hardware parts when referenced in detail or assembly drawing notes, 3D CAD models or relevant purchase orders. COTS items, MIL-SPEC items, Electrical Diagrams (Schematic/Circuit), or Printed Wiring Board and Circuit Card Assembly drawings are typically excluded. Control drawings, typically Procurement Control or Source Control Drawings, may also reference this Standard.

3. References

The following documents form a part of this Standard to the extent specified herein. A more recent revision may be used provided there is no conflict with the text of this Standard. In the event of a conflict between the text of this Standard and the references cited herein, the text of this Standard shall take precedence.

3.1. Cited Standards

- IS-006, Drawing Standards (L-3 CSW Internal Specification for drawing referenced standards)
- ASME B46.1-2009, Surface Texture (Surface Roughness, Waviness, and Lay)
- ASME Y14.41-2012, Digital Product Definition Data Practices
- ASME Y14.43-2011, Dimensioning and Tolerancing Principles for Gages and Fixtures
- ASME Y14.5-2009, Dimensioning and Tolerancing
- ASME Y14.5.1M-1994 (R2012), Mathematical Definition of Dimensioning and Tolerancing Principles
- AWS D17.1-2010, Specification for Fusion Welding for Aerospace Applications
- IEEE/ASTM SI 10-2010, American National Standard for Metric Practice

4. Definitions

4.1. Derivative Data Set

derivative data set: The media created any time data is extracted from a master data set for machine programming, visual aids, inspection aides, First Article Inspection (FAIs), tool fabrication/measurement, or the like. Derivative data is taken from its native environment and used in another CAD/CAM, CMM, or manufacturing/inspection system. Derivative data set geometry is modified from the master data set and translated from

one system to another using a neutral format translator or manually into some 2D systems.

4.2. Master Data Set

master data set: This is the engineering definition typically provided in a 3D representation of the product and viewable on a Computer Aided Design (CAD) system. Models created from the CAD system and provided by L-3 CSW in any of the formats listed in Model Data Standards are considered part of the master data set.

4.3. Product Definition Data Set

product definition data set: a collection of one or more data file(s) that discloses, directly or by reference, by means of graphic or textual presentations, or combinations of both, the physical or functional requirements of an item. Figure 4-1 of ASME Y14.41-2012 gives a good visual representation of a Product Definition Data Set.

5. General Rules

When interpreting L-3 CSW drawings, CAD models and corresponding data sets, the following rules shall apply.

5.1. Unless otherwise specified neither the drawing nor the CAD model represent a complete product definition by themselves. When both primary documents are required for complete product definition, the requirements of ASME Y14.41-2012 paragraph 5.2.2 will apply. Either the drawing or the model may refer to other auxiliary documents. The drawing, 3D CAD model and referenced auxiliary documents comprise a product definition data set that provides full product definition.

5.1.1. A note similar to the following should be used on all drawings to represent this relationship:

THIS DRAWING SHALL BE USED WITH MODEL DATA AVAILABLE UNDER THIS SAME PART NUMBER FOR COMPLETE PRODUCT DEFINITION.

5.2. Parts are modeled at their mean condition unless otherwise specified. The mean condition represents the ideal form desired for the part. If a part is modeled at another condition such as minimum or maximum, a data set note shall state which condition the geometry is modeled at.

5.2.1. Features are fully modeled to show their net shape mean condition except for threaded features which are shown symbolically with a 2D LMC surface for the threads and a 3D MMC surface for the pilot hole. Any other features represented as something other than their net shape shall be noted in the data set.

5.3. If there is any contradictory information between the master data set and any existing drawing, the drawing information shall take precedence.

5.4. Feature of Size definition shall be per ASME Y14.5-2009.

5.5. Product dimensions apply prior to finish unless otherwise specified.

5.6. General default tolerances

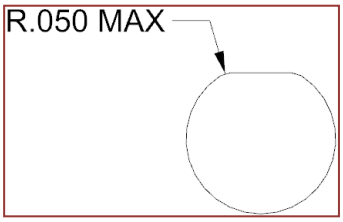
5.6.1. Internal square corners

These tolerances apply to corners shown as square corners in the model unless otherwise shown or specified in the product definition data set.

5.6.1.1. Machined tool end radii nominal size shall be R.010 inch maximum.

5.6.1.2. Lathed tool end radii nominal size shall be R.020 inch maximum.

5.6.2. Internal corners of D-shaped and DD-shaped holes shall be a .050 inch maximum radius where the master data set shows a cylindrical surface with a discontinuous transition to a flat surface.



5.7. Unless otherwise specified, where full thread length cannot be achieved without thread relief or where thread relief is represented in the CAD model: Thread relief shall be 1.5 to 2.5 times the thread pitch.

5.8. Press fit features shall allow for a .0002 minimum interference fit unless otherwise specified.

5.9. Blind Holes

5.9.1. Blind holes shall not breakout unless allowed by a data set note.

5.9.2. Depth

5.9.2.1. All blind holes, simple or threaded, that do not have an explicitly defined depth tolerance on the drawing shall have a minimum depth tolerance equal to the model depicted depth minus .010 inches.

5.9.2.2. All simple or threaded blind holes shall have a maximum depth tolerance as follows:
Maximum depth can be any depth that leaves a minimum material thickness of .020 inches before break through unless otherwise specified. Minimum material thickness to be measured from the drill point.

5.9.3. Drill points may be conical or flat.

5.10. All welds shall be Class C per AWS D17.1 unless otherwise specified.

5.11. The number of significant digits for an implicit nominal dimension is the same as the significant digits of the tolerance associated with it. The number of significant digits for an implicit basic dimension is the same as the number of significant digits used in the solid model to define the feature.

5.12. Rounding of dimensions either queried or resolved from the model shall be in accordance with IEEE/ASTM SI 10.

6. Datum Reference Frame (DRF)

6.1. Default Datum Reference Frame

All product definition data sets which contain undimensioned geometric features require implicit geometric controls for full product definition of their geometry. All such cases shall have a DRF explicitly defined in the data set. This singular DRF becomes the default DRF for all implicit feature definition.

6.1.1. The data set shall define the datum order of precedence for the default DRF, i.e., which datums are to be considered primary, secondary and tertiary as necessary.

6.1.2. The data set shall define the material condition (MMC, RFS or LMC) for all default DRF datums that are features of size.

6.1.3. The default DRF should have enough constraining degrees of freedom to fully locate and orient the product. For reference see ASME Y14.5 sections 4.2 – 4.4

6.2. All DRFs shall be one of the Datum Systems defined in ASME Y14.5.1M-1994 (R2012), Section 4.7.

7. Explicit Feature Definition

These rules and principles apply to all features given explicit nominal dimensions in the data set.

7.1. Feature Tolerance Order of Precedence

All feature tolerances in the data set shall have authority to bound feature attributes according to the following order of precedence:

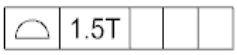
1) An explicitly defined tolerance specified in a view, table or note on the model or drawing.
2) Applicable tolerances check marked as “Explicit” in Tables 1 and 2.
3) Applicable title block tolerances.

8. Implicit Feature Definition

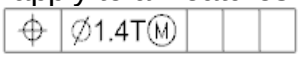
These rules and principles apply to all features not given explicit dimensioning or tolerancing in the data set.

8.1. For implicit feature definition the variable **T** is equal to the range of the 2 Place Title Block Tolerance given in a product definition data set. For example if a 2 Place Title Block Tolerance is specified on the drawing as +/- .01 then **T** = .02.

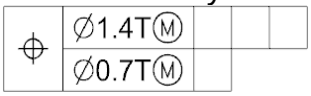
8.2. Implicit feature tolerances are expressed in multiples of T. For example 2**T** equals 2 times the range of the 2 PL Title Block Tolerance, or .04 if **T** = .02.

8.3. Unless otherwise specified the surface profile control 

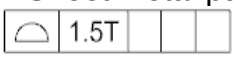
, relative to a products default DRF, shall apply to all features except as follows:

8.3.1. A Position control of 

shall apply to all cylindrical and circular features of size relative to the product default DRF.

8.3.2. Patterns of cylindrical or circular features of size shall be controlled by 

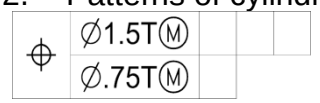
relative to the product default DRF.

8.3.3. Sheet metal parts: Unless otherwise specified the surface profile control 

, relative to a products default DRF, shall apply to all features except as follows:

8.3.3.1. A Position control of 

shall apply to all cylindrical and circular features of size relative to the product default DRF.

8.3.3.2. Patterns of cylindrical or circular features of size shall be controlled by 

relative to the product default DRF.

8.4. Features that are specified in Tables 1 and 2 shall be controlled by the associated tolerance given in their respective tables.

8.5. If a default DRF is not specified per paragraph 6.1, supplier will specify.

8.6. Coaxial Features

Unless otherwise specified the following features shall be considered a single, local coaxial pattern which is controlled by the feature control frame specified in 8.3.1 or 8.3.3.1 as described in ASME Y14.5-2009 section 7.4.2. A coaxiality tolerance shall also be applied as indicated with the second listed feature acting as the datum as exemplified in Figure 7-26 of ASME Y14.5-2009.

- Boss and included hole, 

- Counterbore and companion hole, 

- Countersink and companion hole (no coaxiality control beyond that provided in 8.3.1 and 8.3.3.1 is specified). Countersinks shall be controlled on an RFS basis.

9. Connector Holes

When it is desired to control the shape and size of D-shaped and DD-shaped connector holes with an all-around surface profile control as shown in Figure 9-1:

- An inspection gage or gage set may be used to inspect the hole for compliance to the ideal shape and size defined by the all-around profile control.
- The gage or gage set shall conform to the requirements of ASME Y14.43.
- The inspection gage or gage set must have the following properties:
  - The gage(s) shall have an extruded profile shape equal to the profile LMB (of the feature to be inspected) + .000 / - gage makers tolerance.
  - The gage(s) shall have an extruded profile shape equal to the profile MMB (of the feature to be inspected) + gage makers tolerance / - .000.
  - The length of the extruded profile MMB shape shall be at least three times the depth of the product feature.
  - The length of the extruded profile LMB shape shall be equal to or less than the depth of the product feature.
  - The inspected feature is acceptable if the following criteria are met:
    - Acceptance Criteria: A visual inspection comparing the LMB gage profile shall demonstrate that all of the inspected feature boundary lies within or is contained within the gage boundary.
    - Acceptance Criteria: The MMB gage profile shall be able to be fully inserted into the product feature being inspected and not rotate within it more than 30 degrees in any one direction.

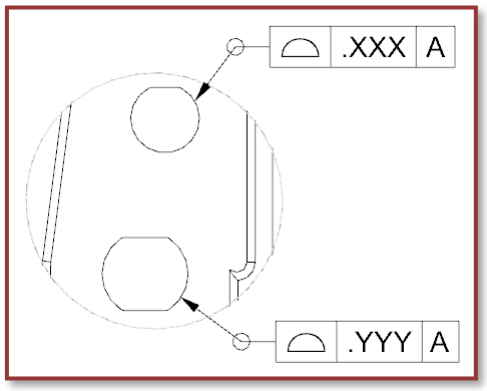


Figure 9-1: Example of Profile Control of Connector Holes.

10. Inseparable Assembly

Unless otherwise specified the following rules shall apply:

10.1. Welded, riveted and bonded assemblies shall be held to the two-place title block tolerance for overall size.

10.2. Clinch nuts, studs and other inseparable fasteners shall conform to recommended installation tolerances per manufacturer specifications.

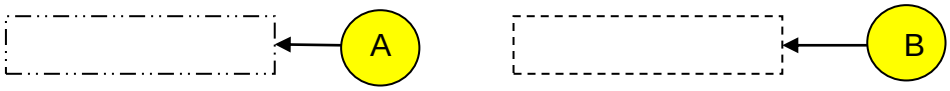
11. Surface Finish

Surface finishes shall conform to ASME B46.1 and shall have the following default values unless otherwise specified:

- Machined parts – 125 µin. Ra or better
- Lathed parts – 125 µin. Ra or better
- Sheet metal parts – 250 µin. Ra or better

12. Part Marking

12.1. Unless otherwise specified parts will be identified by permanent ink, laser marking or by a printed, self adhesive label and located as specified on the drawing.



A: The area indicated by a phantom line specifies the location of the marking to be on the “NEAR SIDE”.

B: The area indicated by a hidden line specifies the location of the marking to be on the “FAR SIDE”.

NOTE: The note “APPROXIMATELY AS SHOWN” on drawings is used to allow some flexibility in marking location.

13. Model Data Standards

L-3 CSW transfers model data in accordance with MIL-STD-1840C and is capable of releasing the following file types: Teamcenter/NX documents (*.jt), IGES documents (*.iges,*.igs), STL documents (*.stl), STEP documents (*.step,*.stp), ACIS documents (*.sat), Parasolid documents (*.x_b,*.x_t), Adobe Acrobat documents (*.pdf) and AutoCAD documents (*.dwg,*.dxf).

14. Tolerance Tables

The following tables define size and form tolerances for certain product features. These tolerances are established to best meet the dimensional characteristics of each feature in compliance with form, fit, and function of the designed part assemblies.

Table 1: Metal Fabrication Processes, Machined Parts

Machined Metal Fabrication Features (in.)	Tolerance (in.)	Application	
		Explicit	Implicit
Webs and flanges: size	**2 PL** Title Block Tolerance		✓
Slots, steps and pockets: width, & depth	**2 PL** Title Block Tolerance		✓
D and Double D holes larger than ∅0 through ∅.250	+ .003 / - .002	✓	✓
D and Double D holes larger than ∅.250 through ∅.500	+ .005 / - .003	✓	✓
D and Double D holes larger than ∅.500	+ .010 / - .005	✓	✓
Width of D (flat to extreme tangent) and Double D (between the two flats)	+ .010 / - .005	✓	✓
Holes; ∅0 through ∅.250	+ .003 / - .002	✓	✓
Holes; larger than ∅.250 through ∅.500	+ .005 / - .003	✓	✓
Holes; larger than ∅.500	+ .010 / - .005	✓	✓
Counterbore Depth	+/- .010	✓	✓
Countersink major ∅; 0 through .064 material thickness	+ .007 / - .003	✓	✓
Countersink major ∅; larger than .064 material thickness	+ .015 / - .005	✓	✓
Outside/overall dimensions	**2 PL** Title Block Tolerance		✓
Radii and fillet dimensions	**2 PL** Title Block Tolerance		✓
Flatness on planes	**3 PL** Title Block Tolerance Range		✓
Straightness, planes, walls flanges	**3 PL** Title Block Tolerance Range		✓
Roundness: boss, holes larger than 1.00 inch	**3 PL** Title Block Tolerance Range		✓
Mill steps tooling mismatch	.010 Maximum		✓
Axisymmetric Features			
Lathe turning: ID, OD	**2 PL** Title Block Tolerance		✓
Length and intersections, depths, steps	**2 PL** Title Block Tolerance		✓
Radii and fillet dimensions	**2 PL** Title Block Tolerance		✓
Mill steps tooling mismatch	.010 Maximum		✓

Table 2: Metal Fabrication Processes, Formed Sheet Metal Parts

Formed Sheet Metal Fab. Features	Tolerance (in.)	Application	
		Explicit	Implicit
Webs and flanges: size	**2 PL** Title Block Tolerance		✓
Slots, steps and pockets: width, & depth	**2 PL** Title Block Tolerance		✓
Material thickness; 0 through .050			
∅0 through ∅.250			
Cylindrical, D and Double D holes	+ .003 / - .002	✓	✓
Width of D (flat to extreme tangent) and DD (between the two flats)	+ .003 / - .002	✓	✓
larger than ∅.250 through ∅.500			
Cylindrical, D and Double D holes	+ .005 / - .003	✓	✓
Width of D (flat to extreme tangent) and DD (between the two flats)	+ .005 / - .003	✓	✓
larger than ∅.500			
Cylindrical, D and Double D holes	+ .010 / - .005	✓	✓
Width of D (flat to extreme tangent) and DD (between the two flats)	+ .010 / - .005	✓	✓
Material thickness; larger than .050			
∅0 through ∅.250			
Cylindrical, D and Double D holes	+ .005 / - .002	✓	✓
Width of D (flat to extreme tangent) and DD (between the two flats)	+ .005 / - .002	✓	✓
larger than ∅.250 through ∅.500			
Cylindrical, D and Double D holes	+ .007 / - .003	✓	✓
Width of D (flat to extreme tangent) and DD (between the two flats)	+ .007 / - .003	✓	✓
larger than ∅.500			
Cylindrical, D and Double D holes	+ .010 / - .005	✓	✓
Width of D (flat to extreme tangent) and DD (between the two flats)	+ .010 / - .005	✓	✓
Counterbore Depth	+/- .010	✓	✓
Countersink major ∅; 0 through .064 material thickness	+ .007 / - .003	✓	✓
Countersink major ∅; larger than .064 material thickness	+ .015 / - .005	✓	✓
Bending: Bend angle	+/- 2°		✓

15. Signatures

This document is approved and signed by:

Director of Hardware Engineering
Manager of Mechanical Design
Director of Manufacturing Operations
Manager of Manufacturing Operations: Machining and Fabrication
Director of Supply Chain Management
Manager of Inspection/OES

16. Nonmandatory Appendix

16.1. Examples of Explicit Tolerance Precedence

This section provides examples to help one understand the precedence criteria established in section 7.1. The content of this section provides nonmandatory information to clarify points and relationships of the standard for the reader. Text and tolerances in this section are tools to add clarification and do not represent requirements.

16.1.1. First Precedence: Explicit Tolerance

An explicit tolerance placed on a drawing or model in a view, table or note has first precedence over the tolerance tables in this document or block tolerances. This tolerance may be given with an explicit nominal dimension as shown in Figure 16-1, in a note or in a table such as Figure 16-2.

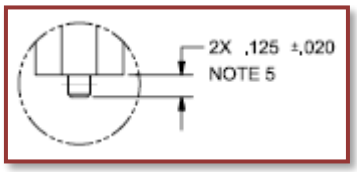


Figure 16-1: Example of Explicit Tolerance

7. UNLESS OTHERWISE SPECIFIED, HOLE SIZE TOLERANCE SHALL BE PER TABLE 1.

TABLE 1

HOLE DIAMETER	TOLERANCE	
0.0 THRU 0.125	(+) 0.003	(-) 0.002
> 0.125 THRU 0.250	(+) 0.004	(-) 0.002
> 0.250 THRU 0.500	(+) 0.005	(-) 0.003
> 0.500 THRU 0.750	(+) 0.008	(-) 0.003
> 0.750 THRU 1.000	(+) 0.010	(-) 0.004
> 1.000	(+) 0.010	(-) 0.005

Figure 16-2: Example of Explicit Tolerance Table & Referencing Note

16.1.2. Second Precedence: Tolerance Tables

Section 14 table tolerances take secondary precedence for explicit dimensions given that the following three criteria are satisfied:

- An explicit nominal dimension exists on the drawing or model where no tolerance is given
- The feature type is listed in the first column
- There is a check mark in the Explicit column

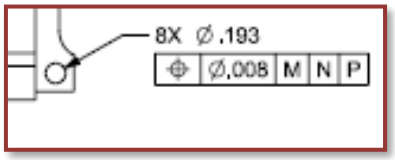


Figure 16-3: Example of a Table Tolerance Application

Figure 16-3 shows an example where a table tolerance takes precedence over a title block tolerance. In this case the explicit nominal dimension O.193 gives the size of the cylindrical hole feature but there is no explicit tolerance specified to the right of the dimension. The next step is to see if the feature in question is listed in a section 14 table. This is a machined hole. Table 1 gives tolerances for machined holes and the following line is found to apply to this feature.

Machined Metal Fabrication Features (in.)	Tolerance (in.)	Application	
		Explicit	Implicit
…	…		
Holes; ∅0 through ∅.250	+ .003 / - .002	✔	✔

The next step is to see if there is a check mark in the Explicit column. In this case there is a red check mark in that column so the appropriate tolerance for the O.193 feature is + .003 / - .002.

16.1.3. Third Precedence: Title Block Tolerances

When an explicit dimension is specified on a drawing or model and the criteria for primary or secondary precedence are not met then title block tolerances control the variation of the dimension. Two examples are given below in Figure 16-4.

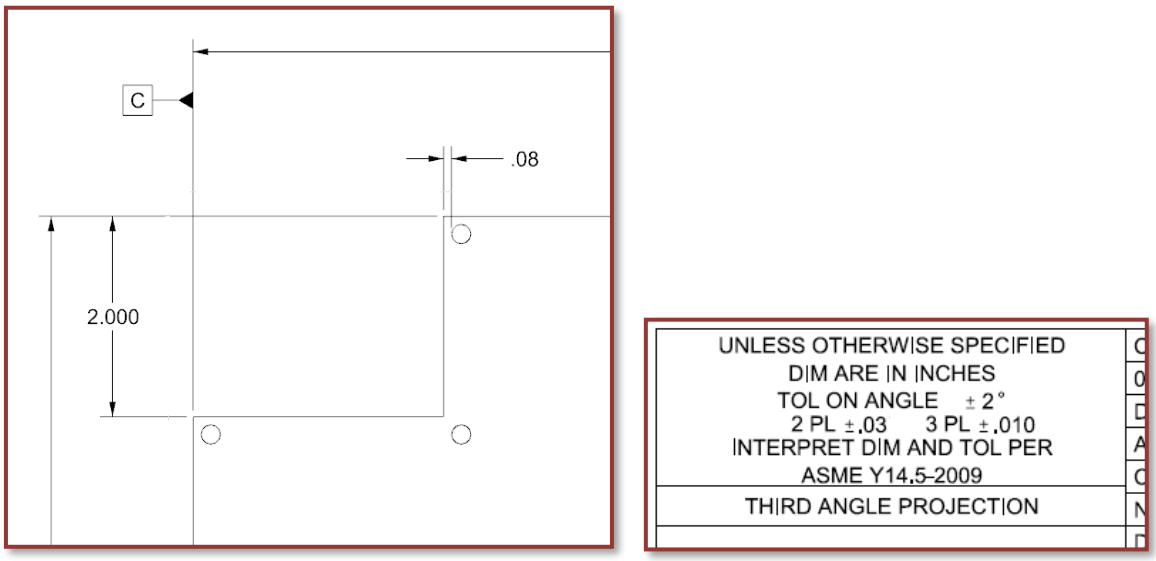


Figure 16-4: Examples of Title Block Tolerance Application

A portion of a sheet metal part drawing is shown in Figure 16-4. There are two dimensions shown: 2.000 and .08.

Let's consider the 2.000 dimension first. Once it is recognized that an explicit dimension has been defined for a feature the next step is to check the tolerance tables to see if that feature is listed there. The 2.000 dimension can be considered a **step** feature. Because this is a sheet metal part Table 2 is consulted and the following line is found.

Formed Sheet Metal Fab. Features (in.)	Tolerance (in.)	Application Explicit	Application Implicit
…	…		
Slots, steps and pockets: width, & depth	**2 PL** Title Block Tolerance	✓	

There is a line for step features in Table 2 but there is **no check mark in the Explicit column**. Therefore this dimension does not get its tolerance from Table 2 but is controlled by the title block tolerance. This dimension has three digits after the decimal so it is controlled by the 3 PL title block tolerance which is +/- .010 on this drawing.

Now let's consider the .08 dimension. Again, it is explicitly defined on the drawing. If no dimension were given for a feature then the feature variation would be controlled by the controls listed in section 8. This dimension controls either the location of the hole relative to the edge or it controls the location of the edge relative to the hole. On another part of the drawing a position control is found for the pattern of holes so the

.08 must control the location of the edge relative to the hole. Table 2 does not have a feature type description for locating edges therefore the tolerance will come from the title block. This dimension has two digits after the decimal so it is controlled by the 2 PL title block tolerance which is +/- .030 on this drawing.

17. Revision History Summary:

Revision	Reason for Update/Revision	Date:
A	Initial release.	4/11/2016
B	Standardized title page, header and footer. Added revision history summary.	11/12/2014
C	Added second sentence to paragraph 5.11. Paragraph 5.9 was expanded into a Blind Holes section to add new blind hole depth default tolerances.	4/11/2016

Copyright