APPLICATION		REVISION HISTORY			
NEXT ASSY	USED ON	REV	DESCRIPTION	DATE	APPR
N/A	N/A	A	Procedure Validation	20250922	AD
N/A	N/A	NC	Procedure Validation	20230913	HY
N/A	N/A	NC	NEW RELEASE	20180809	MCC

CUST: N/A
PRODUCT: N/A
DESC: N/A

DIMENSIONS ARE IN INCHES
TOLERANCES ARE:

ANGLES	DECIMALS
±0.5°	.XX ± .03
	.XXX ± .010

DO NOT SCALE DRAWING
PARTS SHALL BE FREE OF BURRS
BROKEN EDGES .005-.030
FILLETS R .005-.020
SURFACE ROUGHNESS 125

INTERPRET DRAWING IN ACCORDANCE WITH ASME Y14.100-2004 AND ASME Y14.5M-1994

MARVIN ENGINEERING CO. INC.

MARVIN ENGINEERING CO. INC.
260 W. BEACH AVE.
INGLEWOOD, CA. 90302

DRAWN/PREPARED	CS HOWARD	20180809
CHECKED BY	F AVINA	20180809
MFG ENGR	GG GOMEZ	20180809
PROG ENGR	E GUTIERREZ	20180809
APPROVED BY	M CHINFOO	20180809

DRAWING TITLE

STANDARD INSPECTION GUIDE FOR FORGINGS

SIZE	ID NO		REV
A	SP-12-03		A
SCALE	WEIGHT (lb)	CAGE CODE	SHEET
N/A	N/A	32067	1 of 6

1 Purpose

This inspection guide is used to provide guidelines for inspecting forgings.

2 Scope

This inspection standard specifies the interpretation of general notes on engineering drawings and includes the specific procedure for inspection of straightness tolerance. It shall be used for inspection of all forgings unless the engineering drawing specifies more rigid requirements..

3 Responsibilities

QC inspectors: Carry out detailed inspection of all features denoted on a drawing, inspectors must understand the proper uses and limitations of the equipment being considered throughout the inspection process.
Metrology: Provides inspection equipment that is calibrated and maintains calibration intervals.
IPT ME: Provides interpretation of design intent of drawing clarifies ambiguous drawing details and notes.
IPT QE: Interprets customer quality requirements.

4 Equipment

Coordinate Measurement Machine
Calipers
Micrometers
Surface plates
Angle plates
Height gages
Radius gages
Dial indicators
Gage blocks
Gage pins

5 Discussion on inspection process

5.1 Use of Coordinate Measurements Machines (CMMs) and Romer Arms

5.1.1 Appropriate uses of CMMs

Once a CMM program has been created for a given part, a CMM provides a fast inspection report with very little operator input after initial set up of Part Under Inspection (PUI).

5.1.2 Limitations of CMMs

Typical CMM report does not easily match to drawing features. The programmer needs to provide a map so that multiple circular features, linear features, etc. can be correlated to specific features on a dwg. It would better serve the quality community if CMM reports matched up to balloon callouts on balloon dwgs.

CMMs interpolate data (touches on a feature) according to internal algorithms to establish hole size, hole location, orientation of hole axis, datum planes, etc. CMMs tend to report to 4 decimal places. Because of this reporting to 4 places and the algorithms used to interpolate touch data, CMMs tend to provide an impression of precision that may exceed the accuracy of the measurement.

5.1.3 Appropriate uses of Romer arms

Romer arms perform very similarly to CMMs except they do not require a program to begin measurements. Romer arms are manually moved to probe locations. They are acceptable instruments for inspection of forgings.

5.2 Use of Standard Inspection Methods (SIMs)

Many of the bomb racks, ejection systems and missile launcher rails MEC manufactures were designed in the 1940s, 1950s and late 1960s through the mid 1980s. Examples of these programs include MA-4B, MAU-12, MAU-50, BRU-32, LAU-7, LAU-127,128, 129. ASME Y14.5 Geometric Dimensioning and Tolerancing had not yet been widely adopted at the time of design. Also CMMs were not in widespread use and standard inspection tools and go/no go tooling were what the designers anticipated would be used for inspecting parts.

In the mid-1990s through the 2010s is when Y14.5 was widely adopted as the design standard for dimensioning. Programs of this vintage include F-22 and JSF (F-35). In fact, for these programs, Computer Aided Design (CAD) was in widespread use and many of the above programs do not have complete dimensional and tolerance information on the drawings, This information is contained in the CAD models of the parts themselves.

Standard Inspection Methods include calipers, height gages and surface blocks, gage pins, etc., basically everything that is conventional methods of inspection outside of CMM.

5.2.1 Appropriate uses of SIMs

SIMs conforms to the inspection methodology the Original Equipment Manufacturers (OEMs) had in mind when designing our older bomb racks and missile rails. Therefore, SIMs most accurately measure features in accordance with original design intent.

5.2.2 Limitations of SIMs

Using calipers, height gages and surface blocks, joe blocks, pin gages, etc to fully inspect a part will require multiple setups depending on the feature and orientation being inspected. This can increase inspection time compared to CMMs. It also requires a fair amount of operator skill and knowledge to properly use the equipment and interpret the results.

6 Inspection process

6.1 Using Coordinate Measurements Machines (CMMs)

6.1.1 When CMM program available

When a part to be inspected has an MEC QC approved CMM program available, it is recommended to use a CMM for speed and efficiency.

6.1.2 When discrepancies are noted

If the CMM inspection process finds certain features to be Out Of Tolerance (OOT), the feature must be verified using SIMs. Use of SIMs is considered the gold standard and is the accepted value of the inspected dimension.

6.2 Using Standard Inspection Methods (SIMs)

6.2.1 No CMM program available

Use Standard inspection Methods when no CMM program is available for inspecting the parts.

6.2.2 Verification of CMM discrepancies

Use SIMs as noted in 1.2 above when there are features that are found to be OOT with the CMM.

NOTE: CMM program needs to be re-validated to ensure the accuracy of the program..

7 Detailed inspection process using SIMs

7.1 Calipers

Use calipers to measure OD of bosses and simple linear dimensions where the jaws can fit across the feature being inspected.

7.2 Height gages and surface plates

Use height gages with dial indicators and surface plates where linear dimensions cannot be measured with calipers.

Height gages and surface plates can also be used to determine true position location of holes.

This equipment is also very suitable for determining flatness, parallelism, etc

7.3 Gage pins

Class ZZ minus pins shall be used for inspecting all holes that have 3 place dimensions. These are also called standard pins. Standard pins come in increments of .001 and have a tolerance of +.0000/-.0002. The acceptance criteria is if the maximum hole size minus pin goes in but the next size up does not, the hole meets Dwg requirements. For example, if a hole is called out as .188 +.002/-.000 then if a .190 minus pin goes into the hole but a .191 minus pin does not, the hole meets requirements. In an AS9102 inspection sheet, the hole size as measured would be listed as .190.

Class X minus pins shall only be used when inspecting holes that have 4 place dimensions. These are also known as Deltronic pins. They come in increments of .0001 and have a tolerance of +.00000/-.000040

7.4 Angle or Sine plates

Use angle plates in conjunction with surface plates and right angle blocks to inspect angular dimensions.

8 Inspection processes specific to forgings

8.1 Interpretation of drawing notes

8.1.1 Edges of forging flash

In fulfillment of the general note "ALL CORNER RADII _____", no radisuing of edge corners of trimmed forging flash is required.

8.1.2 Ring or cylindrical tolerances

The diameter or radius tolerance given on unmachined surfaces is a blanket tolerance that includes straightness or roundness.

8.1.3 Straightness tolerances

Straightness tolerances are applied to continuous surfaces and are measured separately and independently of all other tolerances except for ring or cylindrical type parts (see paragraph 8.1.4). Inspection for straightness tolerances shall be performed per the procedure in paragraph 8.2. General notes on straightness of forgings may be of three types. Examples of these tolerances are as follows:

a. Incremental tolerance note: General notes such as straightness within 1/64 (.031) in 12 inch, etc, are incremental tolerances notes and shall be interpreted as step functions. For example: Straightness within .016 inch per 9 inch means a .016 inch tolerance is permitted for each 9 inch or fraction thereof. (A forging with a dimension of 22 inch carrying this note is allowed .048 inch tolerance).

b. Maximum tolerance note: General notes such as straightness within 3/64 (.047) max, etc, are maximum tolerance notes and denote the maximum allowable deviation from a true contour template anywhere on the part unless otherwise specified on the engineering drawing.
c. General straightness note: The general notes “FORGING MUST BE STRAIGHTENED BY THE FORGE SHOP” should be interpreted to allow the maximum deviation that will permit the finish-machined part to meet its required tolerances.

8.2 Method for inspection for straightness

Local pits, grinding marks, or mismatch are not a part of the straightness tolerance. If the above conditions result in failure to meet thickness or other specific tolerances, the parts are rejectable. Place the surface of the part on a surface plate or contour template. If the part can be rocked, convex warpage is indicated. If the part cannot be rocked, concave warpage is indicated. Inspect per paragraph 8.2.1 or 8.2.2 as applicable.

8.2.1 Concave warpage (see Figure 1)

Holding the part securely against the surface plate (or contour plate), inspect with feeler gages. Reject if a feeler gage thicker than the specified tolerance can be inserted at any point between the forging and the surface plate.

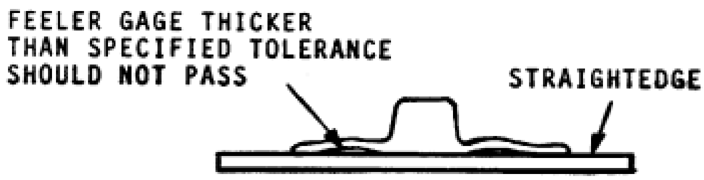

FIGURE 1
CONCAVE WARPAGE

8.2.2 Convex warpage (see figure 2)

Place a feeler gage of the specified thickness between the surface plate (or contour template) and one end of the cast surface to be inspected and hold the part down to prevent rocking. Reject if a feeler gage thicker than specified tolerance can be inserted between the surface of the forging and the surface plate.

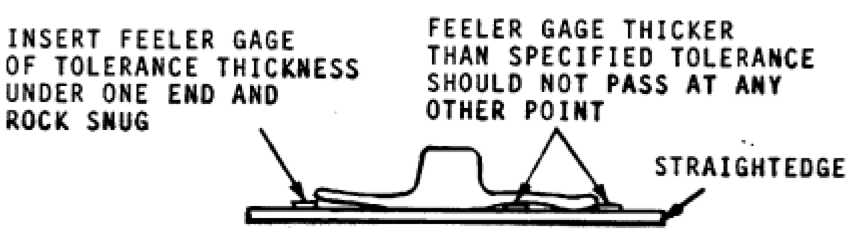

FIGURE 2
CONVEX WARPAGE

8.3 Discussion on tooling points

Forgings typically have some flat surfaces that are located parallel to the parting line of the forging. All other surfaces will have a draft to them to enable the forging to be easily removed from the forging die. The allowable draft angle is specified on the dwg but is typically 7 degrees or less.

Most forgings will show Tooling Points (TP) on the field of the drawing, they are not necessarily located on the forging datums but are typically offset from them. The tooling points will have an area associated with them such as .250 diameter. There will be 3 TPs for the tooling point A plane, 2 TPs for the tooling point B plane and 1 TP for the tooling point C plane.

Tooling point fixtures can be manufactured to create the tooling point planes and the offsets to the forging datums. Where the tooling points are located on flat surfaces, the tooling point can be simulated with a flat face pin of the specified diameter. Where the tooling points are located on forging draft surfaces, the tooling points are simulated with sharp pointed (.020 radius) pins.

Alternatively the forging tooling points can be simulated with a CMM or Romer arm. In this case the tooling points are located with multiple touches of the machine's probe using either the diameter specified on a flat surface or using a .010 diameter area for tooling points located on a forging draft surface. Note the 3 tooling point planes are orthogonal to each other. So orthogonal offsets are used from the forging datums even when the tooling points are on draft surfaces.

When all the tooling points have been entered into the measuring machine's data base the offsets are applied to simulate the datums of the forging. All measurements on the forging are derived from the forging datums, not the forging tooling points.

9 Drawing interpretation and ambiguity issues

Drawings are not always clear on their intention, are frequently double dimensioned, call out obsolete materials and processes. In these instances it the IPT ME and QE are the authorities as to how the dwg shall be interpreted and the feature inspected to meet the requirements of the dwg and our customers. QC inspectors should not hesitate to call in the IPT ME and QE to understand the requirements.