	DATE ISSUED:	NUMBER:		PAGE NO.
L3 communications Integrated Systems	22 OCTOBER 2004	TPS 2-106		1 of 23
	SUBJECT: HOLES, PREPARATION OF			

Originator:	AWCR:	Design Supv:	Project/CE:
S. R. Kramer 22 October 04	R.T. Johnson 22 October 04	J. Threlfall, Jr. 22 October 04	J. Threlfall, Jr. 22 October 04

TECHNICAL PROCESS SPECIFICATION

REVIEWED DATE	REV.	DATE	REV BY	PAGES AFFECTED	REMARKS
	H	13 Nov. 03	SRK	13	Deleted Table in 3.4.5.5 and referred to
					TPS 2-103 for recommended
					countersinks
1 June 04	J	1 June 04	SRK	2, 11	Revision date changed, paragraph
					reference changed
					Added drill speed requirements for
					reaming, countersinking and
					counterboring. Modified caution for
					retracting drill. Changed reference
					paragraph for reaming lubricants.
	K	22 Oct 04	SRK	2, 19, 20	Added paragraph for attribute gages and
					plug gage.

PROCESS SPECIFICATION

Active Amendments:
None

1. SCOPE.

1.1 Scope. This specification establishes the procedures, fabrication practices, and requirements for producing holes and related hole preparations (countersinks, deburring, reaming, etc.).

1.2 Effectivity. This specification is a complete revision of TPS 2-106J, dated 1 June 2004, and is effective upon release of the Document Release Notice (DRN). A solid black bar in the left-hand margin marks changed paragraphs from the previous revision.

1.3 Order of precedence. If there is a conflict between the Engineering drawing and this specification, the Engineering drawing shall take precedence.

2. APPLICABLE DOCUMENTS.

2.1 Government documents. The following document of the issue indicated forms a part of this specification to the extent specified herein. Unless otherwise specified, the current issue shall apply.

STANDARDS

Military

MIL-STD-403C	Preparation for and Installation of Rivets and Screws, Rocket, Missile, and Airframe Structures
NAS618	Fastener Recommended Shank, Hole, and Head to Shank Fillet Radius Limits for

2.2 Non-Government documents. The following documents form a part of this specification to the extent specified herein. Unless otherwise specified, the current issue shall apply.

STANDARDS

L-3 Communications Integrated Systems Component Engineering Drawing

TAS8980	Cutout, D-Hole

SPECIFICATIONS

Industry

American National Standards Institute (ANSI)

ANSI B46.1	Surface Texture (ASME)

L-3 Communications Integrated Systems Technical Process Specifications

TPS 1-415	Phosphate Coatings, Hand Application of
TPS 1-439	Brush Plating, Process for
TPS 1-442	Brush Chromate Conversion Coating for Aluminum Alloys
TPS 1-450	Sleeve Cold Working of Holes
TPS 2-6/3	Edge smoothing of Metal Parts, Requirements of
TPS 2-103	Rivets, Buck Type, Installation of
TPS 5-817	Temper Etch Inspection of Steel

3. REQUIREMENTS

3.1 Equipment.

3.1.1 **Drill motors.** No. 3P-1800 manufactured by Thor Power Tool Co., CAGE 11592 or equivalent.

3.1.2 **Angle air drill motors.** No. 2RL manufactured by Thor Power Tool Co., CAGE 11592 or equivalent.

3.1.3 **Angle drills.** No. AT300 90° or No. AT301 45° manufactured by Aircraft Tools, Inc., CAGE 00784 or equivalent.

3.1.4 **Flex drive drills.** No. AT302 manufactured by Aircraft Tools, Inc., CAGE 00784 or equivalent.

3.1.5 **Stop-countersink units.** No. AT-44OL manufactured by Aircraft Tools, Inc., CAGE 00784 or equivalent.

3.1.6 **Stop-countersink cutters.** No. AT-420 manufactured by Aircraft Tools, Inc., CAGE 00784 or equivalent.

3.1.7 **Hole finders.** No. AT500 manufactured by Aircraft Tools, Inc., CAGE 00784 or equivalent.

3.1.8 **Drill bits.** High speed steel (HSS) drills, solid carbide or carbide tipped, cobalt HSS drills, or equivalent.

3.1.9 **Reamers.** HSS reamers, carbide or carbide insert reamers, or equivalent.

3.2 **Materials.**

3.2.1 **Cool mist lubricant/coolant.** Tap Magic, made by Steco Corp., 2300 Cantrell Rd., Little Rock, AR, 72202, or equivalent as specified by Manufacturing Engineering.

3.2.2 **Drill and tap lubricant.** S.P.E.C., made by Standard Parts & Equipment Corporation, Fort Worth, TX, 76106, or equivalent as specified by Manufacturing Engineering.

3.3 **General requirements.**

3.3.1 **Safety and chemical handling.** Hazardous chemicals and waste disposal shall be per the appropriate Safety and Quality Control Procedures.

3.3.1.1 Always wear safety goggles or eyeglasses with side shields.

3.3.1.2 Do not hold small pieces of sheet metal in hand while drilling. Always use clamps or vise.

3.3.1.3 Never blow drill chips with compressed air.

3.3.2 **Pilot holes.** At the discretion of manufacturing, pilot holes may be used to locate holes from templates and as a centering guide for drilling specific size holes.

3.3.3 **Specific hole drilling**. Standard hole sizes for general machine and hand work shall be per Table I unless otherwise specified by Engineering Drawing, a standard repair procedure, a process specification, a design standard, or material review board disposition, as applicable.

3.3.4 **Standard tolerances.** Table II lists the tolerances for standard hole sizes. Holes .0135 thru 2.000 inches in diameter that are not toleranced on the engineering drawing, regardless of the method of manufacturing (drill, punch, mill, route, laser, electrical discharge machining (EDM), etc., shall conform to the tolerance ranges defined in Table II. When the Engineering Drawing specifies a four (4) place dimension, for example a 0.2499 diameter hole, the hole shall be reamed to that dimension per 3.4.6. Tolerances on non-fastener holes shall be per standard dimensional tolerances (i.e. .xx = ±.03, xxx = ±.01).

TABLE I
STANDARD DRILL SIZES

DRILL SIZE	DEC EQUIV	DRILL SIZE	DEC EQUIV	DRILL SIZE	DEC EQUIV	DRILL SIZE	DEC EQUIV
80	.0135	43	.089	8	.199	25/64	.3906
79	.0145	42	.0935	7	.201	X	.397
1/64	.0156	3/32	.0938	13/64	.2031	Y	.404
78	.016	41	.096	6	.204	13/32	.4062
77	.018	40	.098	5	.2055	Z	.413
76	.020	39	.0995	4	.209	27/64	.4219
75	.021	38	.1015	3	.213	7/16	.4375
74	.0225	37	.104	7/32	.2188	29/64	.4531
73	.024	36	.1065	2	.221	15/32	.4688
72	.025	7/64	.1094	1	.228	31/64	.4844
71	.026	35	.110	A	.234	1/2	.500
70	.028	34	.111	15/64	.2344	33/64	.5156
69	.0292	33	.113	B	.238	17/32	.5312
68	.031	32	.116	C	.242	35/64	.5469
1/32	.0312	31	.120	D	.246	9/16	.5625
67	.032	1/8	.125	1/4 (E)	.250	37/64	.5781
66	.033	30	.1285	F	.257	19/32	.5938
65	.035	29	.136	G	.261	39/64	.6094
64	.036	28	.1405	17/64	.2656	5/8	.625
63	.037	9/64	.1406	H	.266	41/64	.6406
62	.038	27	.144	I	.272	21/32	.6562
61	.039	26	.147	J	.277	43/64	.6719
60	.040	25	.1495	K	.281	11/16	.6875
59	.041	24	.152	9/32	.2812	45/64	.7031
58	.042	23	.154	L	.290	23/32	.7188
57	.043	5/32	.1562	M	.295	47/64	.7344
56	.0465	22	.157	19/64	.2969	3/4	.750
3/64	.0469	21	.159	n	.302	49/64	.7656
55	.052	20	.161	5/16	.3125	25/32	.7812
54	.055	19	.166	O	.316	51/64	.7969
53	.0595	18	.1695	P	.323	13/16	.8125
1/16	.0625	11/64	.1719	21/64	.3281	53/64	.8281
52	.0635	17	.173	Q	.332	27/32	.8438
51	.067	16	.177	R	.339	55/64	.8594
50	.070	15	.180	11/32	.3438	7/8	.875
49	.073	14	.182	S	.348	57/64	.8906
48	.076	13	.185	T	.358	29/32	.9062
5/64	.0781	3/16	.1875	23/64	.3594	59/64	.9219
47	.0785	12	.189	U	.368	15/16	.9375
46	.081	11	.191	3/8	.375	61/64	.9531
45	.082	10	.1935	V	.377	31/32	.9688
44	.086	9	.196	W	.386	63/64	.9844

TABLE II
STANDARD DRILLED HOLE TOLERANCES

HOLE DIAMETER.	TOLERANCES
.0135 thru .125	+.004/-.001
.126 thru .250	+.005/-.001
.251 thru .500	+.006/-.001
.501 thru .750	+.008/-.001
.751 thru 1.000	+.010/-.001
1.001 thru 2.000	+.012/-.001

3.3.5 **Surface finish.** Fastener hole surfaces shall comply with Table III unless otherwise specified by Engineering Drawing, a standard repair procedure, a process specification, a design standard, or material review board disposition, as applicable.

TABLE III
SURFACE CHARACTERISTICS IN FASTENER HOLES

ACCEPTABLE	UNACCEPTABLE
1. Circumferential scratches that are a minimum of .0625 inch or 10% of the part thickness from the surface of the part, whichever is smaller. 2. Spiral scratches that are a minimum of .0625 inch or 25% of the part thickness from the surface of the part, whichever is smaller 3. Longitudinal scratches not more than 50% of the length of the hole in any one part that neither start nor end within .0625 inch or 25% of the part thickness from the surface of the part, whichever is smaller. 4. Surface roughness of 125 Ra or smoother as defined in ANSI B46.1.	Scratches, nicks, or cuts intersecting a part surface or exceeding limits specified as acceptable.

3.3.6 **Blind holes.** All holes shall be considered through unless a depth is indicated on the Engineering Drawing. The depth of blind hole is measured to the depth of the cylindrical portion, not including the point.

3.3.7 **Edge smoothing and deburring.** Edge smoothing and deburring shall be accomplished per TPS 2-6/3 unless otherwise specified by Engineering Drawing, a standard repair procedure, a process specification, a design standard, or material review board disposition, as applicable.

3.3.8 **Punching or notching of aluminum alloy parts.** Punching or notching of aluminum parts in the heat treated condition (2024T3, 6061T6, 7075T6, etc.) to final (net) size shall not be performed to Type 2 (Fatigue or Sonic Critical) parts as defined in TPS 2-6/3 unless specified on the engineering drawing. These parts may be rough punched provided a minimum of .030 inches of material remains around the periphery of the punched or notched hole. The hole shall then be finished by drilling, boring, or reaming to final size. Type 2 parts that are heat treated after punching or notching may be finished to final (net) size. All Type 1 parts may be punched or notched to final (net) size.

3.3.9 **EDM holes**. Holes produced by Electrical Discharge Machining (EDM) shall be produced to be a minimum of .014 inch diameter undersize. The hole shall then be reamed to the correct size.

3.3.10 **Electrical connector mounting holes**. Unless otherwise specified, "D" shape electrical connector holes shall comply with TAS 8980.

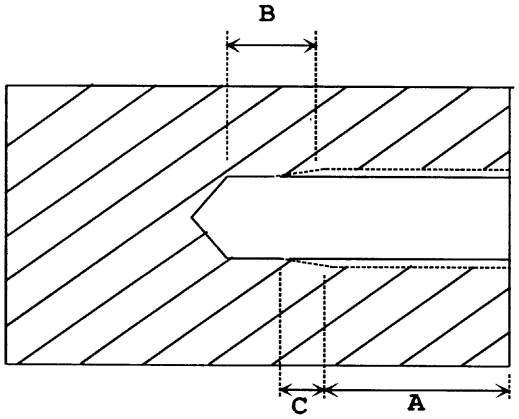
3.3.11 **Cold working**. Cold working of holes, when specified in engineering drawings, MRB disposition or specifications shall be performed per TPS 1-450.

3.3.12 **Tapped holes.** Whenever tapping a blind hole in which the Engineering Drawing does not specify the specific thread relief or the specific thread length, the following requirements shall apply.

3.3.12.1 **Depth of blind holes.** Blind hole depth is measured to the cylindrical portion, not to the depth of the point. The depth of the hole should permit threading in one operation. This means that beyond the last fully formed thread, clearance must be provided for the 5-pitch chamfer on the tap and the chips produced in cutting the thread. This distance should be approximately 1-1/2 times the diameter (1-1/2D). See Figure 1.

3.3.12.2 **Full form threads.** Tapped holes shall have the correct amount of perfect full form threads to accommodate the full form threads of the required fastener.

3.3.13 **Holes in fatigue critical or sonic areas.** Detail parts designed for fatigue critical or sonic areas shall be processed as specified on the Engineering Drawing.



A- Full Form Threads
B- Tap Clearance (1-1/2D)
C- Incomplete Threads

FIGURE 1
TAPPED HOLE

NOTE: The first and last one and one-half (1.5) threads on a fastener are incomplete threads.

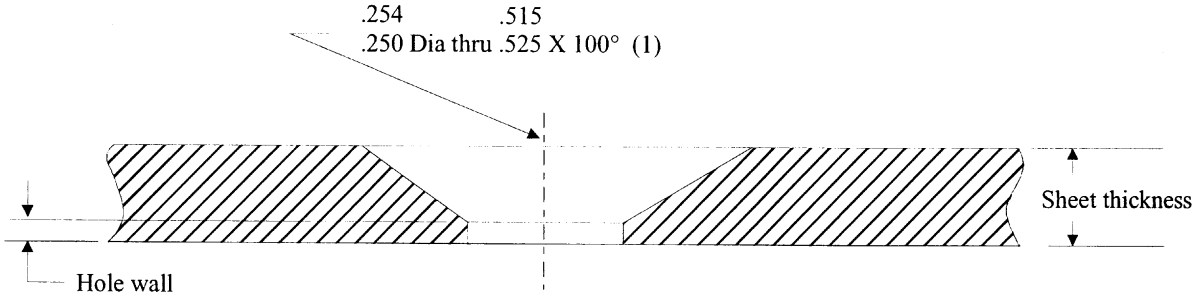
3.3.14 **Chamfer on threads.** All machine cut threaded ends, external and internal, except tapped holes are chamfered at 45°. Tapped holes are to be countersunk approximately 120°, angle included, to minimize the effect of the burr due to tapping, except where thin stock makes this impractical. See Table IV for countersink hole diameters.

NOTE: Countersinks are to be concentric within ±.0-05 inch Full Indicator Runout (FIR).

TABLE IV
MAXIMUM COUNTERSINK DIAMETER

TAP SIZE	COUNTERSINK DIAMETER
0	.080 MAX
1	.093
2	.109
3	.125
4	.140
5	.156
6	.171
8	.203
10	.234
1/4	.296
OVER 1/4 TO 1/2	TAP SIZE + .046

3.3.15 **Wall depth.** Countersunk holes with a wall depth of 10% or less of the sheet thickness are not required to meet drilled hole diameter tolerance. See Figure 2.



(1) Drilled hole diameter

FIGURE 2
WALL DEPTH EXAMPLE

3.3.16 **Drilling and reaming of high-strength alloy steels.** For holes in steel over 180 ksi tensile strength, the following guidelines apply. The drilling of holes, including chamfering and spot facing in steels subsequent to hardening to strength levels of 180 ksi ultimate tensile strength and above shall be avoided whenever possible. When such drilling and reaming is unavoidable because of manufacturing sequence, tooling and techniques necessary to avoid overheating shall be used. Tooling and processes used shall be qualified by demonstration. Microhardness and metallurgical examinations of test specimens should be used to determine the depth of disturbed metal and possible overheated areas resulting from drilling. Parts shall be inspected for overheating per TPS 5-817.

3.3.17 **Normalcy.** Unless otherwise specified on the engineering drawing or as specified in 3.3.22 and 3.3.23., all holes shall be normal within ± 2° of the surface being drilled.

3.3.18 **Perpendicularity of a machined hole intersecting a machined surface.** Unless otherwise specified, the centerline of a finished hole intersecting and perpendicular to a machined surface shall be square within reference plane representing the finished surface within .001 inch per inch of axial length.

3.3.19 **Perpendicularity of holes drilled into a machined surface.** Unless otherwise specified, when a drawing shows a hole drilled into a machined surface, the centerline of the finished hole shall be square with the reference plane of the finished surface per Table V.

TABLE V
SQUARENESS TOLERANCE

HOLE DIAMETER (Inch)		SQUARENESS TOLERANCE PER INCH OF DRILL DEPTH (Max)
0	.1249	.010
.125	.499	.008
.500	.749	.006
.750	2.000	.005
Over 2.000		.002

3.3.20 **Perpendicularity of intersecting machined holes**. When a drawing shows two machined holes with their center lines in the same plane and intersecting at right angles, unless otherwise specified, the center lines of the finished holes shall be square within .001 inch per inch.

3.3.21 **Parallelism of a machined hole to a machined surface.** Unless otherwise specified, if a machined hole is indicated on a drawing as being parallel to a machined surface, and located from it, the centerline of the finished hole shall be parallel to the reference plane representing the finished surface within .001 inch per inch.

3.3.22 **Parallelism of machined holes.** Unless otherwise specified, when two machined holes are shown on a drawing as being parallel and one is dimensioned with respect to the other, the centerlines of the finished holes shall be parallel with .001 inch per inch of hole length.

3.3.23 **Concentricity.** Unless otherwise specified, two diameters shown concentric on a drawing shall not run eccentrically more than the arithmetical sum of tolerances indicated on the drawing for those diameters, when checked against each other. Examples are counterbores, spotfaces, and countersinks.

3.3.24 **Flanged holes.**

3.3.24.1 **Hole sizes**. Dimensions and tolerances shall comply with the drawing or the applicable design standard.

3.3.24.2 **Flange direction**. Flange direction shall be as shown on the engineering drawing. If not specified on the drawing, flange direction shall be optional.

3.3.24.3 **Workmanship.**

3.3.24.3.1 **Burrs and sharp edges.** Burrs shall be removed and sharp edges shall be broken .005 to .015 inch unless otherwise specified by Engineering Drawing, a standard repair procedure, a process specification, a design standard, or material review board disposition, as applicable.

3.3.24.3.2 **Tool marks**. Tool marks or scratches shall be blended and shall not exceed the depth values listed in Table VI.

TABLE VI
BLENDING DEPTH

Material Thickness	Limit of Depth of Depression (After Blending)(Inch)
.008 - .011	.001
.012 - .020	.002
.021 - .040	.003
.041 - .080	.005
Over .080	.006

3.3.24.3.3 **Cracks**. There shall be no visible cracks or splits.

3.3.24.3.4 **Wrinkles.** Wrinkle height shall not exceed 5% of the metal thickness.

3.3.24.3.5 **Cleanliness.** Metal sheet shall be cleaned prior to punching or drilling the holes. Any lubricant used shall be removed afterward. Slight staining is acceptable.

3.3.25 **Finishing**. Whenever possible, finishes shall be applied after holes and threads are completed. Holes, including threads, spotfaces, and blended areas, produced after other finishes have been applied shall be touched up per the following guidelines unless otherwise specified by the Engineering drawing:

a. Aluminum shall receive chemical conversion treatment per TPS 1-442.

b. Corrosion resistant stainless steels shall be mechanically polished with aluminum oxide paper, silicon carbide paper, or skotchbrite pads.

c. Carbon and alloy steels shall either be brush cadmium plated per TPS 1-439 or, after solvent wiping shall be phosphate treated per TPS 1-415. Only low embrittling cadmium plating method shall be used on steels over 160 KSI tensile strength.

3.4 **Procedure.**

3.4.1 **Basic drilling practices.**

3.4.1.1 Always use sharp drills. Exchange dull drills for sharp drills with tool crib.

CAUTION Never personally sharpen drills.

3.4.1.2 Before drilling, make sure drill is centered in drill chuck and secured in position by locking with a chuck key.

3.4.1.3 When drilling thin sheet metal parts, support the back side of the part with a wooden block.

3.4.1.4 Make sure drill is perpendicular (± 2°) to work or elongated hole will result.

3.4.2 **Drilling tools.** Consult the tool crib for common or special drilling tools and accessories.

3.4.3 Drill speeds and feeds.

3.4.3.1 Drill speeds (rpm) for holes sizes and materials shall be per Table VII.

TABLE VII
MAXIMUM RATED MOTOR SPINDLE SPEED (RPM)

Drill Diameter (Inch)	Aluminum and Magnesium Alloys Dry or with Lubricant	Steel, Titanium, and Inconel using Lubricant.
Up to 3/16	5000	750
Up to 1/4	5000	500
Up to 3/8	2800	300
Up to 1/2	1200	250
Up to 3/4	1000	200
Up to 1	750	150
Up to 1-1/2	500	100

3.4.3.2 For reaming, countersinking, and counterboring operations, the spindle speed shall be approximately 1/2 drill speed for all metals listed in Table VII. The feed shall be approximately two times that used for drilling.

3.4.3.3 Proper hand drilling feeds shall be achieved by using sufficient force to keep the drill cutting continuously.

CAUTION: Drilling with excessive force may unwind or bow the drill flutes, causing elongated and double holes and tool breakage. Lack of force to keep drill cutting will dull the drill rapidly.

3.4.3.4 When machine drilling, the drill feed per revolution shall be in the range from .0015 to .006 inch for aluminum alloys. Harder alloys (i.e. 2024, 7075) require lower feeds, while higher feeds are more suitable for softer alloys (i.e. 6061, 5052).

3.4.3.5 When drill or reamer passes through the material or joint being drilled, the motor shall be stopped for hand drilling.

CAUTION: To prevent axial marks, always remove the drill or reamer from the hole by slowly hand rotating the chuck counterclockwise while retracting the drill.

3.4.3.6 Drilling aids such as drill tables, bushings, or guide blocks shall be used for best resulting hole angularity, tolerance, concentricity, and finish.

3.4.3.7 Drilling and reaming lubricants per 3.2.2 shall be used.

3.4.3.8 Close tolerance holes shall be defined as holes with a total dimensional tolerance of .002 inch or less. When close tolerance holes are specified, holes shall be either pilot drilled/core drilled/reamed or pilot drilled/double margin step drilled. Extreme caution shall be taken when using double margin drills to insure that the total material thickness being drilled is less that the body length of the pilot portion (First step). The first step shall have drilled through the material completely before the final step of the drill begins any cutting.

3.4.3.9 Care shall be exercised to insure that drills and reamers are replaced before they become dull and cause unacceptable finishes or hole sizes and that chips are removed from the drill and reamer flutes between each hole.

3.4.4 **Deburring.**

3.4.4.1 **Solid rivets.** Burrs under either head of solid rivets do not as a rule result in unacceptable riveting, provided the final installation meets the limitations of this specification and TPS 2-103.

3.4.4.2 **Blind fasteners.** Burrs under either head of blind fasteners shall be removed to prevent unacceptable gaps, low rivet preload and loose joints.

3.4.4.3 **Excessive removal of material.** When deburring is necessary, care shall be taken to remove as little material as possible beyond the burr. Removal of any appreciable amount of material from the edge of the hole (excessive chamfering) could result in a joint of lowered strength. Follow the directions of TPS 2-6/3. (See 3.3.8)

3.4.4.4 **Stacked parts.** If parts are stacked to be assembled later, burrs shall be removed to prevent scratching of adjacent assemblies and possible hazard to personnel.

3.4.4.5 **Separation of parts.** Deburr and remove any drill chips or foreign matter which will cause separation of parts when fastened together.

3.4.4.6 **Minimizing burrs.** Burrs and chips may be minimized by clamping the sheets securely before drilling and by backing up the work if the rear member is not sufficiently rigid.

3.4.5 **Countersinking.**

3.4.5.1 **Machine countersinking.** Machine countersinking shall be accomplished by using stop-countersinking tools whenever possible. The tools shall be set by countersinking on trial material. A flush head rivet of the type and size required to be installed shall be used as a gage to determine the countersink depth. All countersinking tools must have an angle corresponding to the angle of the fastener to be installed (i.e., use 100° countersink for 100° rivet).

3.4.5.2 **Stop-countersink.** A stop-countersink must be used in all machine countersinking except where there is not enough clearance. Stop-counters consist of four principal parts; the housing, the cutter, the pilot and the skirt or stop. To change from one rivet size to another, only the pilot cutter needs to be changed. The skirts are adjustable, permitting the cutter to be raised or lowered. When using a stop-countersink, always hold the skirt firmly with one hand. If the skirt turns or vibrates, the material being worked will be scarred.

3.4.5.3 **Back (inverted) countersinking.** When access for countersinking is difficult, it may be necessary to use a back countersink. The back countersink consists of two pieces; a rod of the same diameter as the drilled hole, and a cutter which can be attached to the rod. A countersink hole is made on the far side by pulling on a drill motor attached to the near end of the shaft. Since there is no skirt or stop on this type of countersink, extreme caution should be exercised to prevent making the countersunk hole too deep.

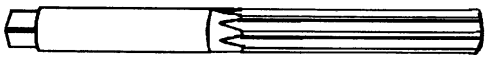
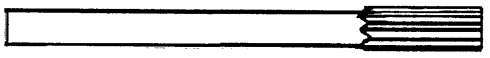
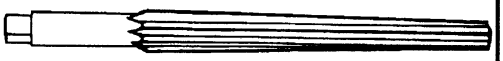
3.4.5.4 **Special countersinks.** In places where access is difficult, special or threaded countersinks can be used in right angle drills or even flex drills. These types of countersinks do not have an adjustable stop and must be used with caution to prevent making the countersunk hole too deep.

3.4.5.5 **Countersinking, flush both sides.** When the Engineering Drawing specifies machine countersinking for rivets that must be flush on both sides, the countersink for the manufactured head shall be per 3.4.5.1 and TPS 2-103. TPS 2-103 provides countersink diameters for upset heads. See TPS 2-103 for Rivet Code Designation.

3.4.5.6 **Punch formed countersinking.** Punch formed countersinking shall be accomplished by the use of a countersink tool on the punch press. The tools may be set by countersinking on trial material. A flush head rivet or fastener of the type and size required to be installed shall be used as a gage to determine the countersink depth. All countersinking tools must have an angle corresponding to the angle of the fastener to be installed (i.e., use 100° countersink for 100° rivet). Punch formed countersinking is not allowed for parts fabricated per edge smoothing requirements of TPS 2-6/3 Type 2. Hole tolerances shall be per 3.3.3.

3.4.6 **Reaming.** Certain types of fasteners require holes that are more accurate than can be obtained by drilling. Usually these holes are specified by a four (4) place decimal. In such instances the holes should be brought to size using a reamer. The size of the reamer shall be checked with a micrometer before being used. Commonly used reamers are listed in Table VIII.

TABLE VIII
REAMERS

STRAIGHT FLUTE HAND REAMERS This reamer is used in a tap handle. The user must take care to hold the tool perpendicular to the surface being reamed.	
STRAIGHT FLUTE MACHINE REAMERS This reamer must be used carefully to avoid enlarging the hole on the reverse side. It is recommended that a slow speed drill motor or press with a regulator be used with this reamer.	
STRAIGHT FLUTE TAPERED REAMER This reamer does not tend to enlarge the reverse side of the hole. It can be used only where space permits running the reamer all the way through the hole. The size of this reamer is checked at the larger end.	

3.4.7 **Drilling of stainless steel.** Drill at a slow speed and use a split tip drill (split point drill). Apply a steady pressure so that the drill will cut at all times. When the drill slips, the stainless steel becomes work hardened and is very difficult to cut. Use a lubricant coolant per 3.2.

3.4.8 **Drilling glass fiber reinforced laminate.** Fiber reinforced laminate material is quite hard and very abrasive. Special modified drill bits must be provided for economical drilling of good quality holes. The drill material shall be cobalt high speed steel, solid carbide or carbide tipped high speed steel, ceramic, diamond coated or equivalent.

3.4.8.1 **Modified drill bits.** A special tip grinding procedure was developed for modifying standard drills capable of long run and quality fiber reinforced laminate drilling. The modified tip configuration is shown in Figure 3.

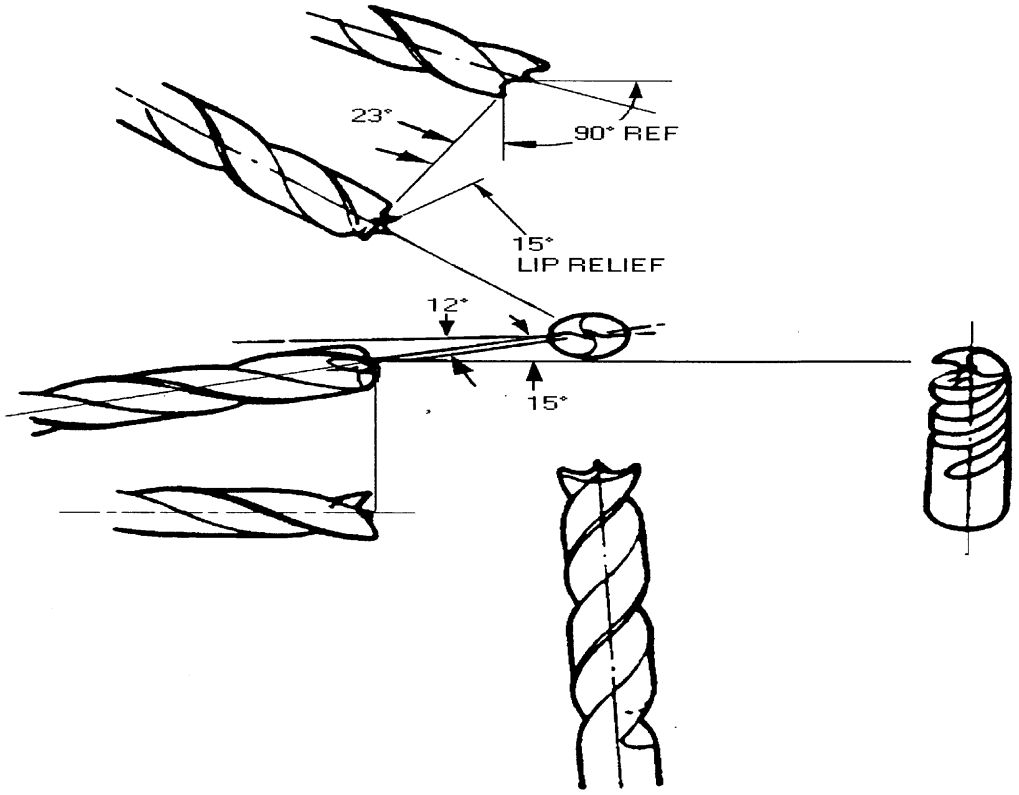


FIGURE 3
MODIFIED DRILL TIP

Modify drill bits per the following procedure.

a. The setup for steps 1 and 2 shows the tool rest at a 15 degree angle, the lip relief angle of the drill. The front view of the grinding wheel shows a 35 degree angle line. This is the angle at which the drill is fed into the grinding wheel. The 35 degree angle allows the center of the drill to extend beyond the forward cutting points of the land after the drill has been ground. See Figure 4.

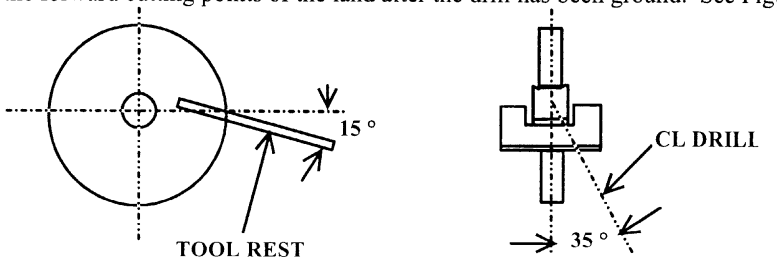


FIGURE 4
SETUP FOR GRINDING STEPS 1 & 2

b. In grinding steps 1 and 2, the rotational position of the drill bit is very important. The chisel edge must be in a vertical position so that the grinding cut is parallel to the chisel edge angle. After grinding one lip, rotate the drill 180 degrees to grind the opposite lip. See Figure 5.

c. The final grinding, steps 3 and 4, are done without the tool rest. The bit is ground on each side to reduce the helix angle forming a cutting edge at the center of the web and bringing the chisel edge to a point. See Figure 5.

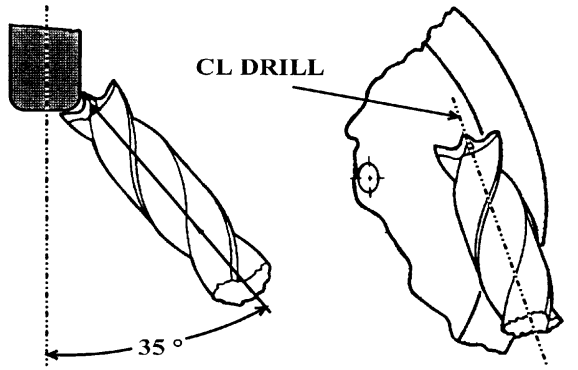


FIGURE 5
GRINDING PROCEDURE FOR MODIFYING STANDARD DRILL BIT

3.4.8.2 **Back-ups.** Back-ups such as wood or metal shall be used on the opposite side of the laminated material to prevent lamination separation, and to eliminate chipping as the drill breaks through the material.

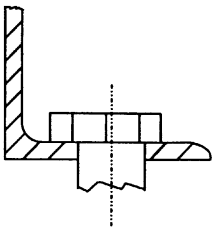
3.4.9 **Spotfacing and counterboring.**

3.4.9.1 **Spotfacing.** Spotfacing is a method of machining a flat surface around a hole, usually to provide a true bearing surface for a bolt head or nut where the original surface is sloping or uneven. Figure 6 illustrates the use of this method with the installation of a standard fastener.

3.4.9.2 **Counterboring.** Counterboring is a method of machining a cylindrical recess around a hole. A standard counterbore has a flat bottom, resulting in a situation similar to a spotface; however, the two methods differ in the depth of the machining and in their manner of use. When more than 0.030 inches of material is to be removed, the operation is considered to be counterboring. Figure 6 illustrates a common use of counterboring.

SPOTFACE

To provide a flat, true surface for seating a bolt head.



COUNTERBORE

To depress a bolt head entirely below the surface.

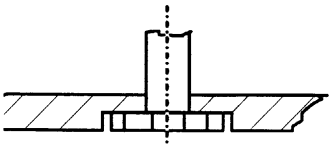


FIGURE 6
SPOTFACING AND COUNTERBORING

3.4.9.3 **Procedure.** Spotfacing and counterboring operations are usually done with the same standard tool which is equipped with interchangeable pilots of various sizes. The tool may be used with any drilling device. For best results, the surface to be penetrated should be as nearly perpendicular to the axis of the hole as possible.

Designs requiring the use of long pilots to engage the hole before cutting often cause tool breakage, and should be avoided (Ref. Figure 7). Extreme slopes and large fillet radii should be reconsidered for possible design improvement in this regard. Holes should be located far enough from adjacent walls to provide tool clearance with the most adverse accumulation of tolerances.

3.4.9.4 **Standard sizes.** Standard spotface and counterbore diameters, fillet radii, and pilot diameters are listed in Table IX.

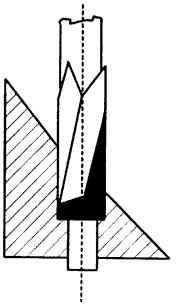


FIGURE 7
UNDESIRABLE SPOTFACING CONDITION

TABLE IX
SPOTFACE OR COUNTERBORE SIZE FOR GENERAL USE

Bolt or Screw Size	Diameter	Minimum Pilot Diameter	Fillet Radii
4	.38	.09	.031
6	.44	.12	.031
8	.44	.16	.031
10	.56	.16	.062
1/4	.62	.16	.062
5/16	.69	.22	.062
3/8	.81	.22	.062
7/16	.88	.22	.062
1/2	1.00	.22	.062
9/16	1.19	.34	.062
5/8	1.31	.34	.062
3/4	1.44	.34	.062
7/8	1.62	.34	.062

3.4.9.5 **Back spotfacing and back counterboring.** Where accessibility is limited, a special cutting tool may be used with the driving pilot applied to the backside of the material to be spotfaced or counterbored. The cutting tool is separable from the driving pilot, and must be installed and removed at each hole. Table X lists standard size back spotface and counterbore tools.

3.4.9.6 **Spotfacing of bosses.** If a boss is to be spotfaced, a spotfacing tool larger than the diameter of the boss should be used, to prevent the formation of a sharp ridge as a result of the rounded corner on the tool or eccentricity between the spotface and the boss.

3.4.9.7 **Surface texture.** The finish for spotfacing and counterboring shall not exceed 125 Ra.

3.4.9.8 **Standard radius.** A corner radius of 0.03 (+0.01, - .000) inches is required for cutters up to and including 0.50 inches in diameter, and a radius of 0.06 (+0.02, -.000) inches is allowed for cutters larger than 0.50 inches in diameter.

TABLE X
STANDARD BACK SPOTFACE SIZES

DIAMETER	LENGTH
.25	.38
.31	.38
.38	.50
.44	.50
.50	.62
.56	.62
.62	.62
.69	.62
.75	.62
.81	.62
.88	.62
.94	.62
1.00	.62

4. QUALITY ASSURANCE PROVISIONS.

4.1 Responsibility for inspection. Organizationally assigned personnel responsible for performing the requirements described herein are also primarily responsible for ensuring the quality of the process activities. Other personnel within the operating department, or others as deemed appropriate by management may be assigned to verify compliance with requirements listed herein. Quality Assurance departments may perform inspections or audits as necessary to provide adequate oversight of process controls.

4.2 Monitoring procedures for equipment used in process. The process owner or department performing the process shall verify by process audits or inspections that all equipment used in this process are per the requirements specified herein.

4.3 Monitoring procedures for materials used in process. The process owner or department performing the process shall verify by process audits or inspection that all materials used in this process are per the requirements specified herein and are within any applicable shelf life limits.

4.4 Inspection. Inspection shall consist of visual, dimensional, and nondestructive evaluation of the requirements of this drawing.

4.5 Hole dimensions. Hole dimensions and roundness shall be verified by used of ball gages which are rotated 360° in the holes. This includes counterbores.

4.6 Attribute gages. Attribute gages are fixed gauges that are designed to check a single dimension and/or tolerance limit. There is a difference between gaging and measuring a dimension. A gage is designed to check the dimension with Go or NoGo (good or bad) results. Measuring instruments measure the dimension with variable (discrete values) results. All gages are designed with gage maker's tolerance and wear allowance. Wear allowance can cause a gage to be produced exactly at the limit of the dimension it was designed to check. In this case, the gage will not fit the part if the part has been made exactly at that limit (an acceptable part). Therefore, it is possible, with all gauges, to reject a part that is on the borderline of being acceptable. The rule of thumb for using all gauges is to reinspect a borderline suspect part that the gage has rejected using a measuring instrument before making the "reject" decision.

4.6.1 Plug gages. These are simply hardened steel "pins" that are accurate in size, with a handle. There are usually two members, the Go and the NoGo (See Figure 9). They are used to check inside diameters such as drilled holes on an accept or reject basis. There are three basic kinds of plug gages:

1. Single purpose. A Go or NoGo member only (not both).
2. Double end. Those that have a Go and NoGo.
3. Progressive. Those where the Go and NoGo are on the same side (made from the same piece of steel). The Go is the first part used.

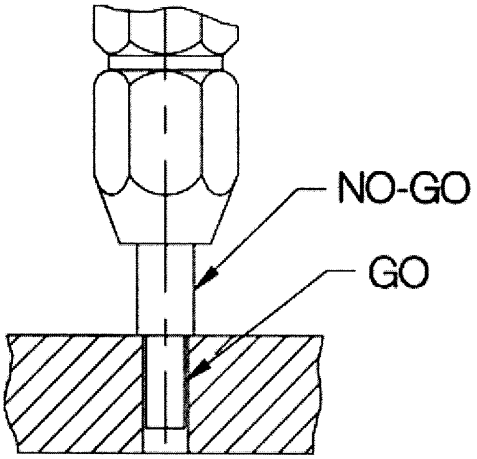


FIGURE 8
PLUG GAGE

4.6.2 **Plug gage application.** Start the member at a slight angle into the hole, rotate it to a 90° position, and allow it to go into the hole (if it will) under its own weight.

4.6.3 **Plug gage limitations**. There are two main limitations of plug gages.

2. They will not detect out of roundness, taper, or barrel-shaped or bell-mouthed holes (which can be a problem).

4.7 **Countersinks.** Countersinks shall be inspected by means of rotating countersink gages 360° in the holes and observing closely for out-of-round or out-of-concentricity conditions.

4.8 **Surface characteristics in fastener holes**. Surfaces in holes shall be inspected per Table III.

5. **PREPARATION FOR DELIVERY.** This section is not applicable to this specification.

6. **NOTES.**

6.1 **Intended use.** This specification is intended to provide a standard hole size and tolerance for holes along with the procedures and requirements for producing holes and related applications, such as reaming, countersinking, spotfacing, counterboring and deburring.

6.2 **Drawing note.** When specified on the engineering drawing, the drawing note shall read "Install hole per TPS 2-106."

Numerical Index

Numerical Index (continued)

Numerical Index (continued)