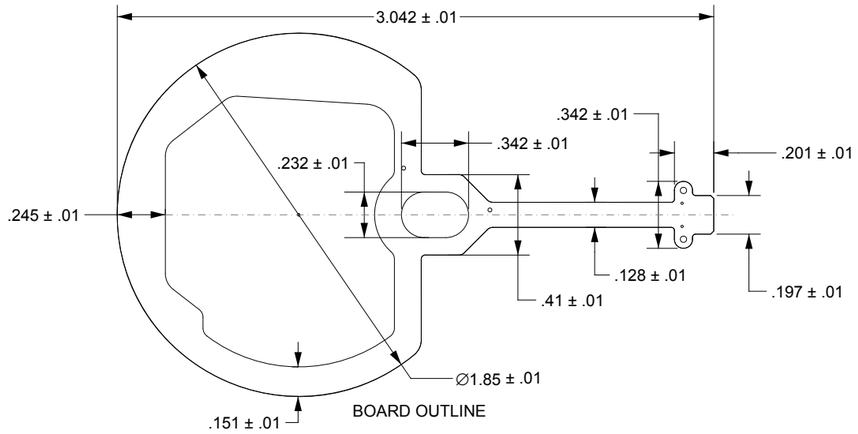
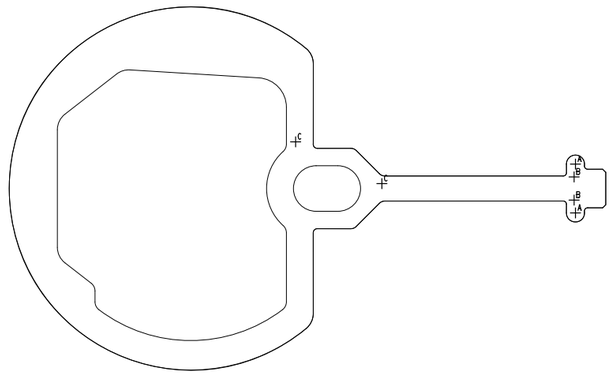
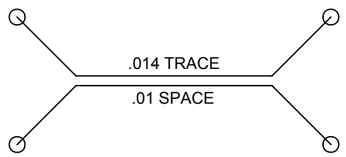




DRILL PATTERN

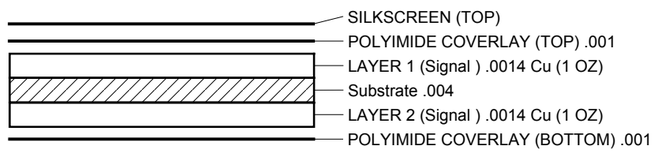


SCALE: NONE

DETAIL A - SMALLEST TRACE WIDTH

Drill Chart				
Size	Sym	Qty	Plated	Tolerance
.0118	B	2	Yes	+0/-.0118
.02	C	2	Yes	+0/-.02
.032	A	2	No	+/-.002
Total		6		

LAYER STACK-UP



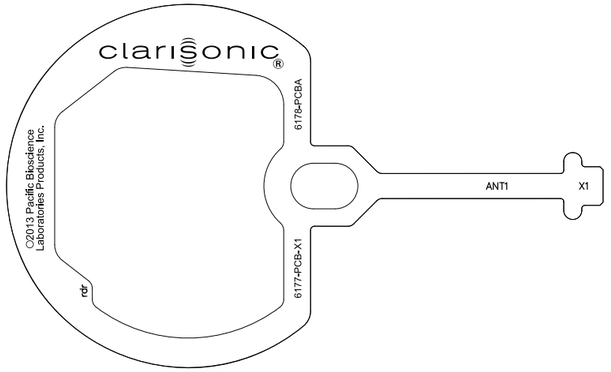
REV	REVISION NOTES	AUTHOR	APPROVED
X1	initial flex antenna design for Pedi RFID	R. Rutledge	2013-07-03

FAB NOTES: UNLESS OTHERWISE SPECIFIED

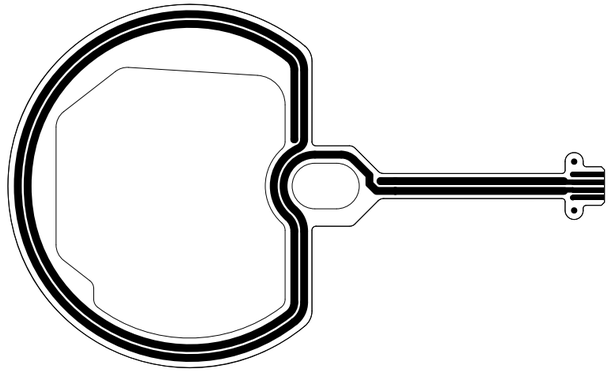
1. Dimensions are given in inches if no units are specified
2. Refer to 6178-SCH (Schematic) and 6177-GBR (Gerbers) for design data.
3. Fabricate Flex in accordance with IPC-6013 Class 2 using customer-supplied data
4. Material: Double-layer flex circuit with polyimide coverlay and adhesiveless substrate
   A. Copper Weight: .5 oz. rolled annealed Cu with .5 oz. Cu plating. Finished Cu thickness 1 oz.
   B. Laminate shall be per IPC-4202,4203, and 4204, with UL 94 V-0 flammability rating
   C. Overall flex thickness to be .01 ±10%.
5. Finish:
   A. Apply .001 thick polyimide coverlay to both sides of PCB per IPC-4203 using "Solder Mask" CAD data for areas to remove coverlay.
   B. Exposed copper shall be plated with Electroless Nickel Immersion Gold (ENIG) per IPC-4552
      a) Minimum 3um Ni and 0.05um Au plating
   C. Silkscreen/Overlay shall be white, permanent, non-conductive ink. Silkscreen shall not encroach on any solderable component pad.
6. Tolerances:
   A. Unless otherwise specified all hole dimensions apply after plating. Plated through holes to have a minimum of .001 copper wall plating.
   B. All holes shall be located within .005 diameter of true position. All holes surrounded by land pads shall have a minimum annular ring of .001. Tangency on holes with pad breakout is acceptable.
   C. Conductor outlines shall be within .001 of artwork originals.
   D. Maximum radius on any inside corner: .020
   E. Layer to layer registration shall be within .003
   F. Board outline contours shall be within .005
7. Test Requirements:
   A. 100% electrical verification of opens/shorts
8. A certification of conformance shall be supplied to the circuit board assembler with each lot indicating that PCBs were built and tested in accordance with these requirements.
9. Panelization shall be at the fabricator's discretion to minimize cost and maximize yield.

clarisonic
Pacific Bioscience Laboratories Products, Inc.
PO Box 429
Redmond, WA 98075
TEL: 425-285-4000
FAX: 425-952-5335
www.clarisonic.com

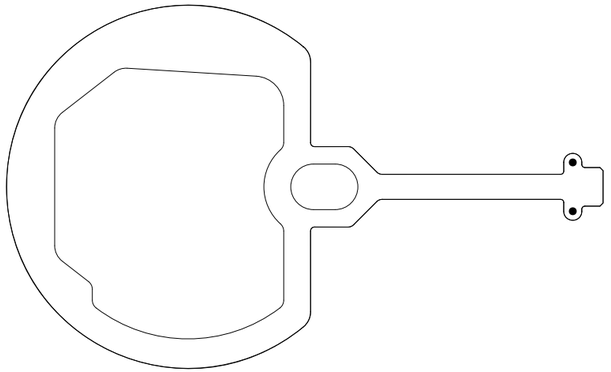
DRAWN R. Rutledge	DATE 2013-07-03	DWG TITLE Flex Antenna for RFID, Pedi Fabrication Drawing		
ENGINEER R. Rutledge	DATE 2013-07-03			
CHECKED D. Gunderson	DATE 2013-07-03			
APPROVED S. Straka	DATE 2013-07-03	SIZE C	DWG NO 6177-PCB	REV X1
ISSUED R. Rutledge	DATE 2013-07-03	SCALE 2 : 1	SHEET NAME Bare Board	SHEET 1 OF 3



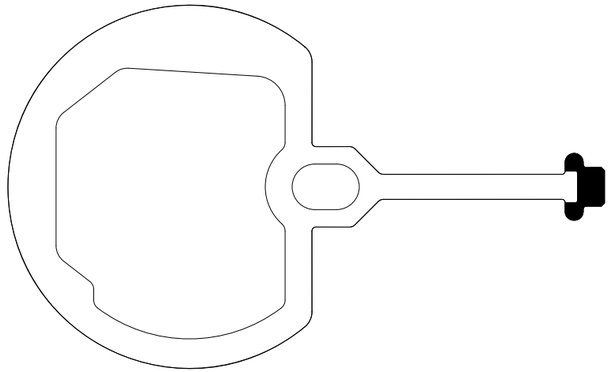
Layer : Top Overlay Gerber: .gto



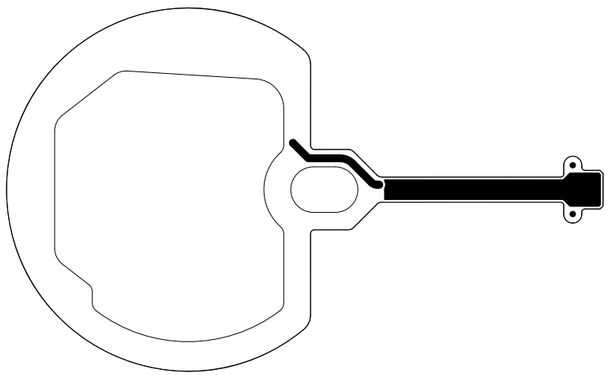
Layer : Bottom Layer Gerber: .gbl



Layer : Top Coverlay (neg) Gerber: .gts



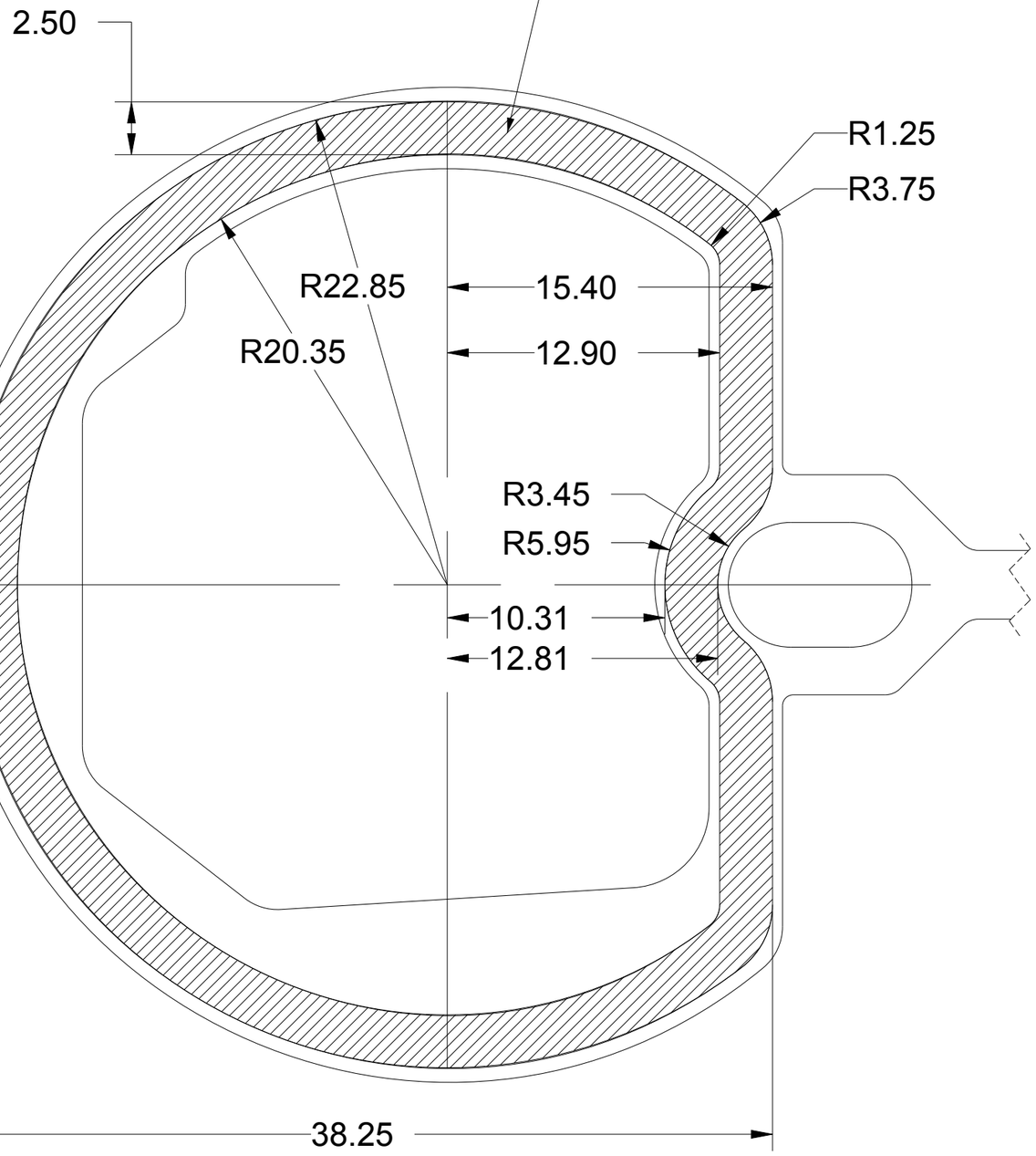
Layer : Bottom Coverlay (neg) Gerber: .gbs

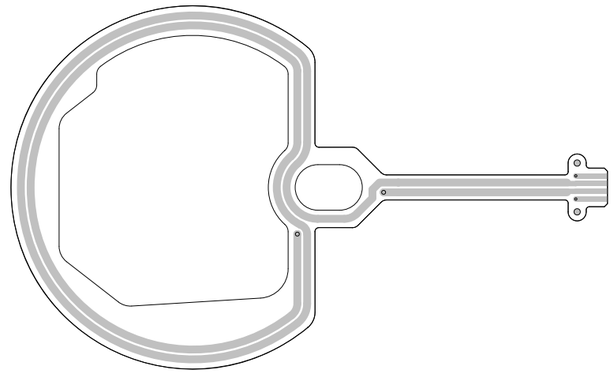


Layer : Top Layer Gerber: .gtl

SHEET NAME:	Layer Prints	
SIZE C	DWG NO 6177-PCB	REV X1
SCALE 2 : 1	Pacific Bioscience Laboratories Products, Inc.	SHEET 2 OF 3

Attach strips of TESA 8853 FPC Mounting Tape or equivalent die-cut to the dimensions shown. Place approximately as indicated, ensuring that tape does not hang over any edges.





Bottom Side

SHEET NAME:	Pressure Sensitive Adhesive Strips	
SIZE C	DWG NO 6177-PCB	REV X1
SCALE 2 : 1	Pacific Bioscience Laboratories Products, Inc.	SHEET 3 OF 3