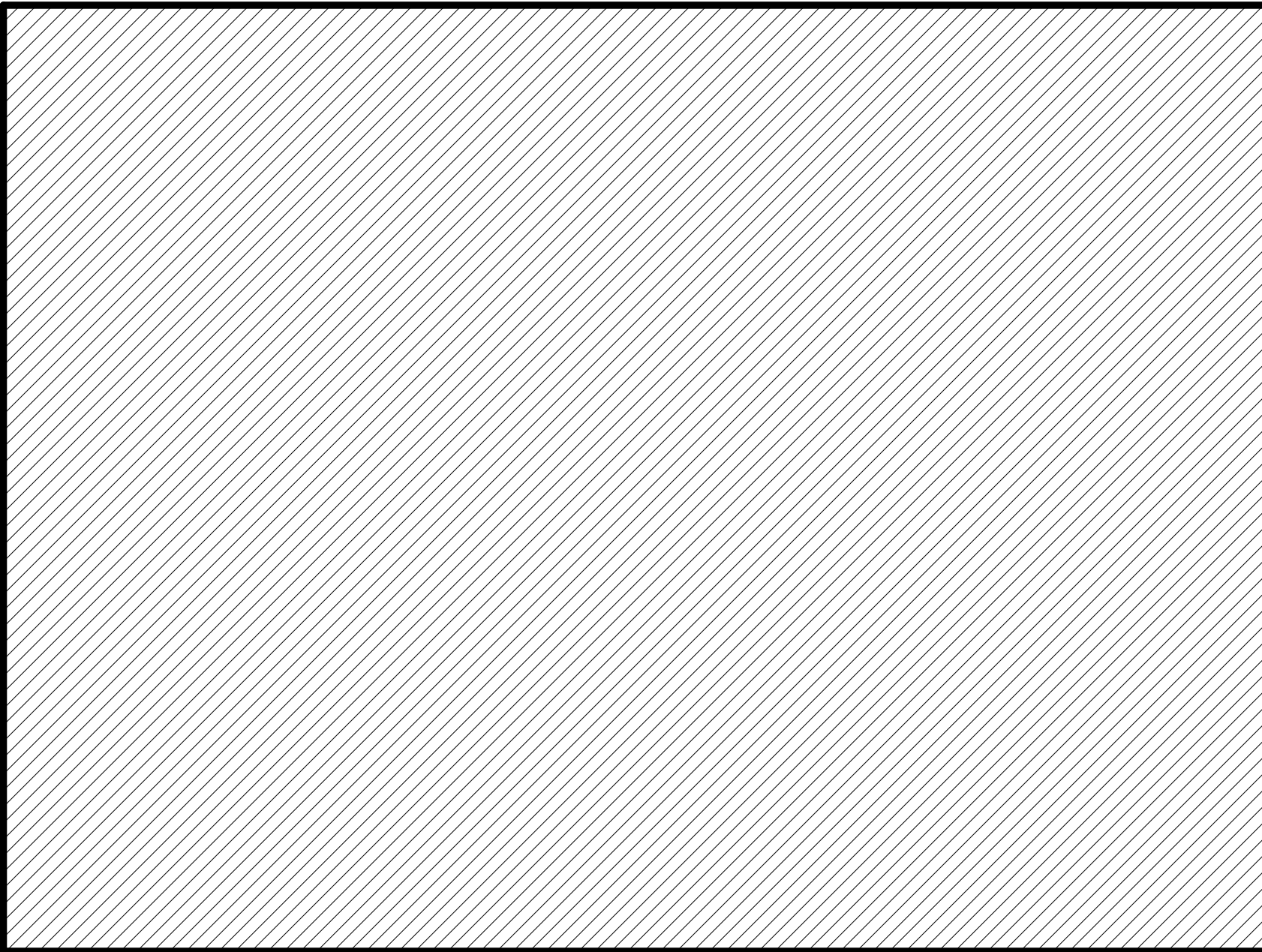


Symbol	Count	Hole Size	Plated	Hole Type	Drill Layer Pair	Via/Pad	Pad Shape	Template	Description	Hole Tolerance (+)	Hole Tolerance (-)
○	2	40.00mil (1.016mm)	PTH	Round	Top Layer - Bottom Layer	Pad	(Mixed)	(Mixed)			
☆	5	25.00mil (0.635mm)	PTH	Round	Top Layer - Bottom Layer	Via	Rounded	v114h64m124			
▽	19	16.00mil (0.406mm)	PTH	Round	Top Layer - Bottom Layer	Via	Rounded	v89h41m99			
□	27	12.00mil (0.305mm)	PTH	Round	Top Layer - Bottom Layer	Via	Rounded	v71h30m0mx0			
	53 Total										

Layer	Name	Material	Thickness	Constant	Board Layer Stack
	Top Overlay				
	Top Solder	Solder Resist	0.40mil	3.5	
1	Top Layer		1.40mil		
	Dielectric 1	FR-4	27.40mil	4.8	
2	Bottom Layer		1.40mil		
	Bottom Solder	Solder Resist	0.40mil	3.5	
	Bottom Overlay				

Total board thickness: 31.00mil



IC-901 Tone Unit (RevA) CONTACT INFO:

Joe Haas

joeh@rollanet.org

4301 Yellowstone Dr.

Taylor, TX 76574

+1 512-577-5575

IC901_TU, RevA PCB FAB Notes:

ALL DIMENSIONS IN MILS UNLESS NOTED.

MTL: FR-4, 0.031 (0,8mm) THK +/- 5%

BOARD EDGE DIMENSION TOLERANCE: +/- 10%

NUMBER OF COPPER LAYERS: 2, WT: 1 OZ

FINISH: LEADED HASL (LEAD-FREE OR ENIG MAY BE SUBSTITUTED)

SOLDERMASK: SMOBC, TWO SIDES, GREEN

SILKSCREEN: 2-SIDE, WHITE

HOLE SIZES ARE FINISHED DIMENSION

BOARD OUTLINE IS 1mil TRACE ON ALL PLOT LAYERS. REMOVE FOR FAB

GERBER LAYER MAP:

TOP SILK: IC901_TU.GTO

TOP SOLDERMASK: IC901_TU.GTS

TOP COPPER: IC901_TU.GTL

BOTTOM COPPER: IC901_TU.GBL

BOTTOM SOLDERMASK: IC901_TU.GBS

BOTTOM SILK: IC901_TU.GBO

DRILL FILE: IC901_TU.TXT

DRILL TOOL FILE: IC901_TU.DRR (HOLE TOLERANCE +0/-0.003)

DRILL/FAB DRAWING: IC901_TU.GD1 (THIS DWG)

PLACE PCB FAB DATE/MFR CODE ON BACK-SIDE SILKSCREEN

