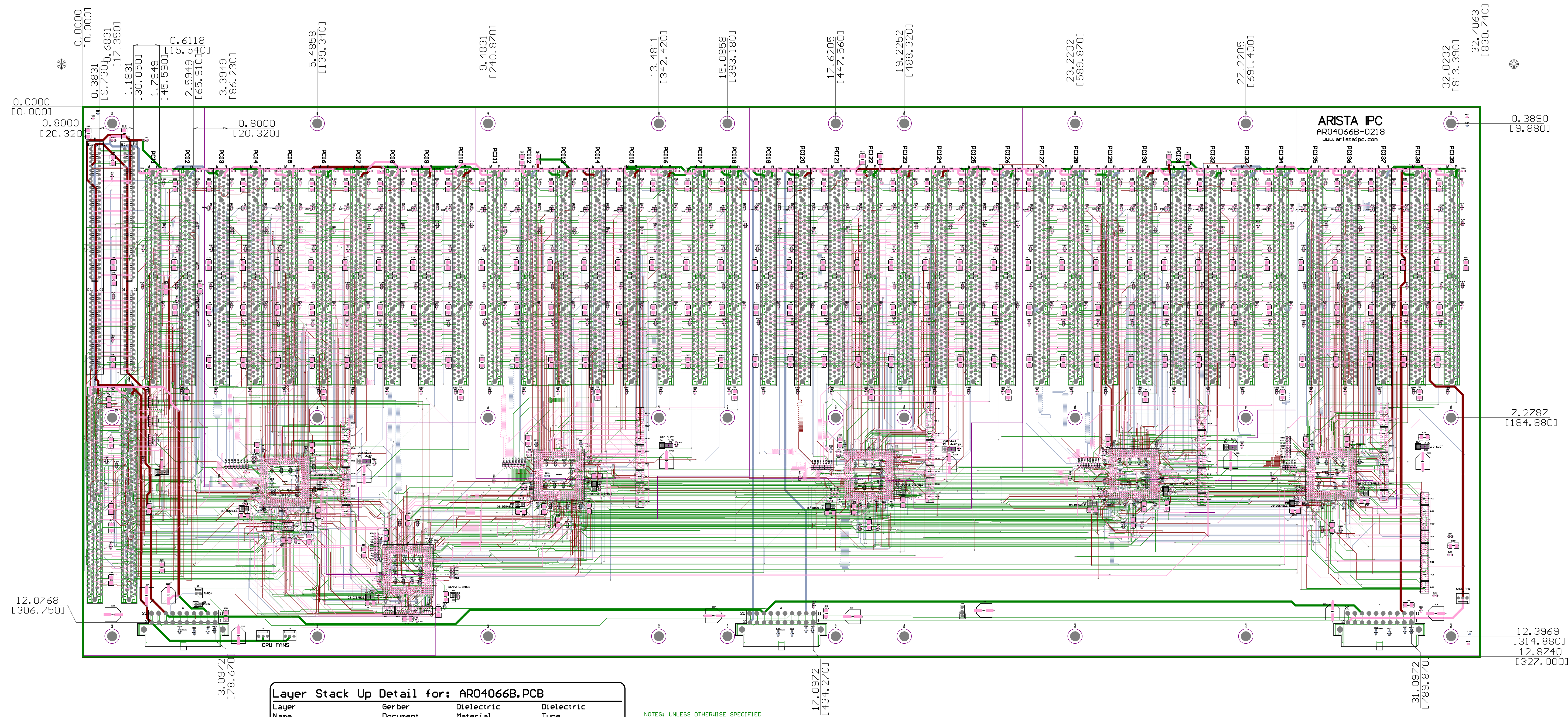




Layer Stack Up Detail for: AR04066B.PCB			
Layer Name	Gerber Document	Dielectric Material	Dielectric Type
TopLayer	(.GTL)		
MidLayer1	(.G1)	FR-4	Core
InternalPlane1	(.GP1)	FR-4	PrePreg
InternalPlane3	(.GP3)	FR-4	Core
InternalPlane2	(.GP2)	FR-4	PrePreg
InternalPlane4	(.GP4)	FR-4	Core
MidLayer2	(.G2)	FR-4	PrePreg
BottomLayer	(.GBL)	FR-4	Core

NOTES: UNLESS OTHERWISE SPECIFIED

MATERIAL: EPOXY FIBERGLASS FR4 OR EQUIVALENT, 0.0625 +/- 0.002 FINISHED THICKNESS.
FINISHED COPPER THICKNESS TO BE 1 OZ. EXTERNAL LAYERS, 1 OZ. INTERNAL LAYERS
MINIMUM COPPER PLATING .001 THICK FOR PLATED THROUGH HOLES. ANNULAR RING TO BE 0.004 MINIMUM.
SOLDER MASK OVER BARE COPPER, BOTH SIDES, GREEN IN COLOR.
SOLDER MASK REGISTRATION +/- 0.003 . ALL TRACES ADJACENT TO PADS SHALL BE COMPLETELY COATED WITH SOLDER MASK.
ALL EXPOSED COPPER AREAS 500KRON WHITE TIN OR EQUIV.
SILKSCREEN BOTH SIDES USING WHITE EPOXY INK.
HOLE DIAMETER TOLERANCE IS +/- 0.003 AFTER PLATING
HOLE LOCATION +/- 0.002 . MAXIMUM LAYER TO LAYER MISREGISTRATION SHALL BE 0.005 .
MEASUREMENT METHOD MUST COMPLY TO MIL-P-55110D, FIGURE 1.
FINISHED CONDUCTOR WIDTH SHALL NOT VARY MORE THAN +/- 10% FROM ARTWORK MASTER.
WARP AND TWIST OF BOARD SHALL NOT EXCEED 0.010 INCH PER INCH.
ALL DIMENSIONS ARE IN INCHES. TOLERANCES: .XX = +/- 0.010 , .XXX = +/- 0.005 .
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E
VENDORS UL APPROVED LOGO TO BE LOCATED ON SOLDER SIDE OF BOARD.
DRAWING IS VIEWED FROM COMPONENT OR PRIMARY SIDE.
LAYER STACK UP IS VIEWED FROM COMPONENT SIDE
ALL HOLES ARE PLATED THROUGH EXCEPT AS NOTED.
GOLD PLATE FINGERS 0.000030 THICK GOLD OVER 0.000100 NICKEL.
DRILL SIZE UNITS ARE THOUSANDTHS OF AN INCH.
VIA TENCING AS PER GERBER FILES
RADIUS INTERNAL / EXTERNAL SHARP CORNERS 0.062R MIN. IF TAB ROUTED.

AR04066B.PCB
Multilayer Composite Print
Multilayer Composite Print
4/13/2008