



Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	.GTS		0.4mil	Solder Resist	3.50	
Top Layer	.GTL	2.1mil				
Power Plane	.GP1	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	.GP2	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	.GBL	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	.GBS		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE

MATERIAL ->

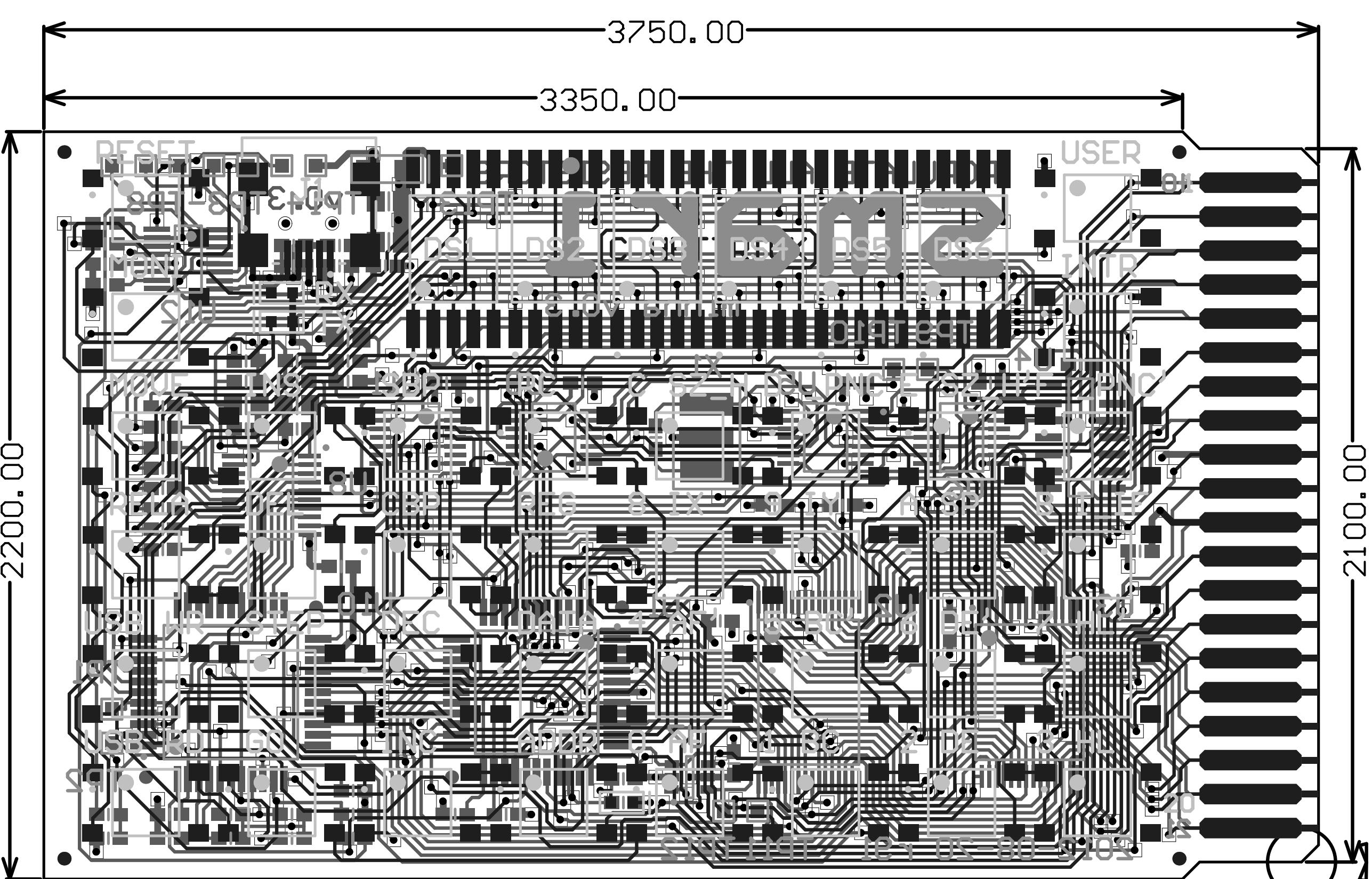
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT, LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26. MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING. ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS. THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED) SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING SHALL BE IMMERSION GOLD FINISH. SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK. THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD. UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON THE TOP SIDE OF THE BOARD

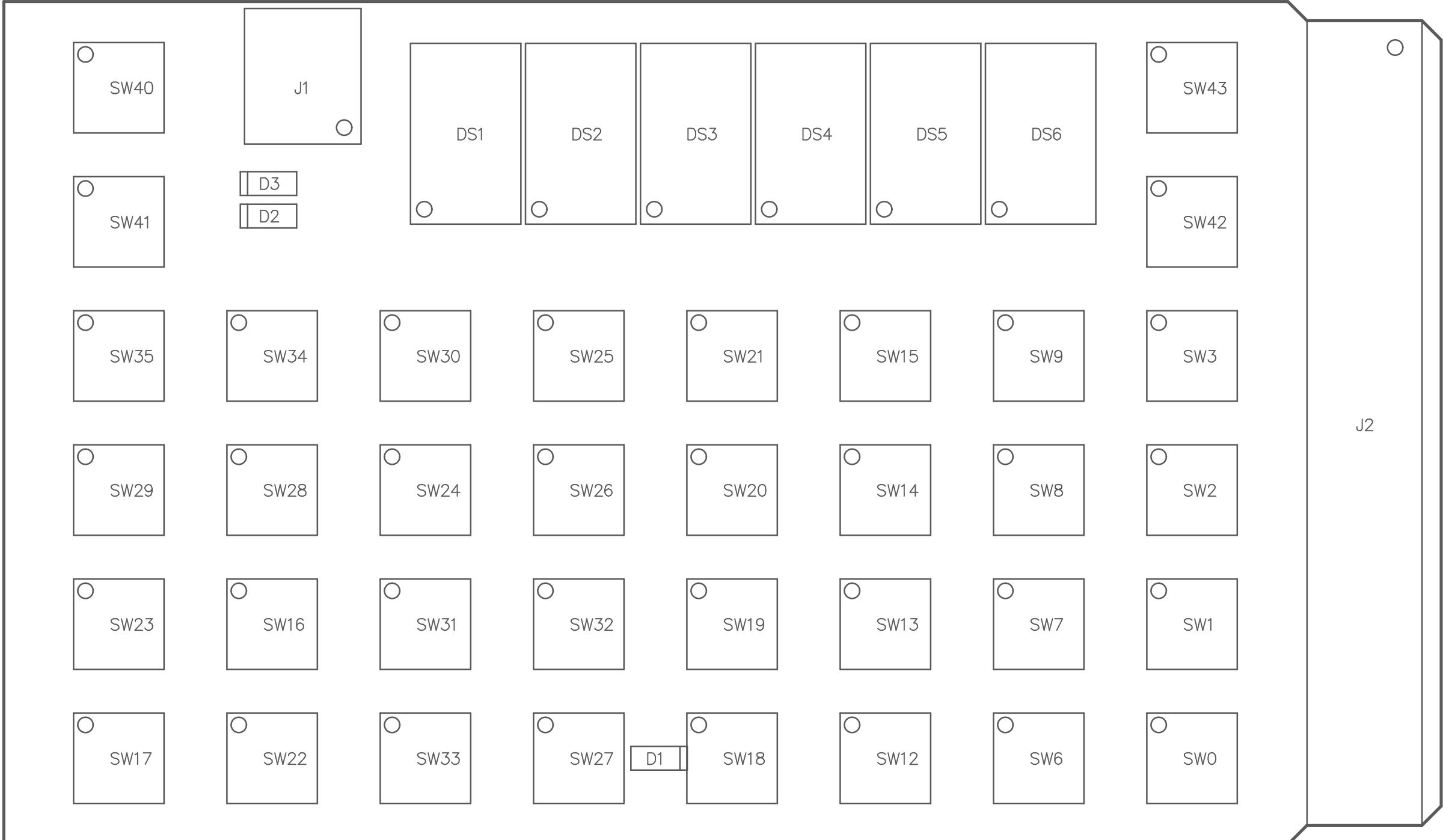
TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%. CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA. HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005" HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM. SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE. MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.

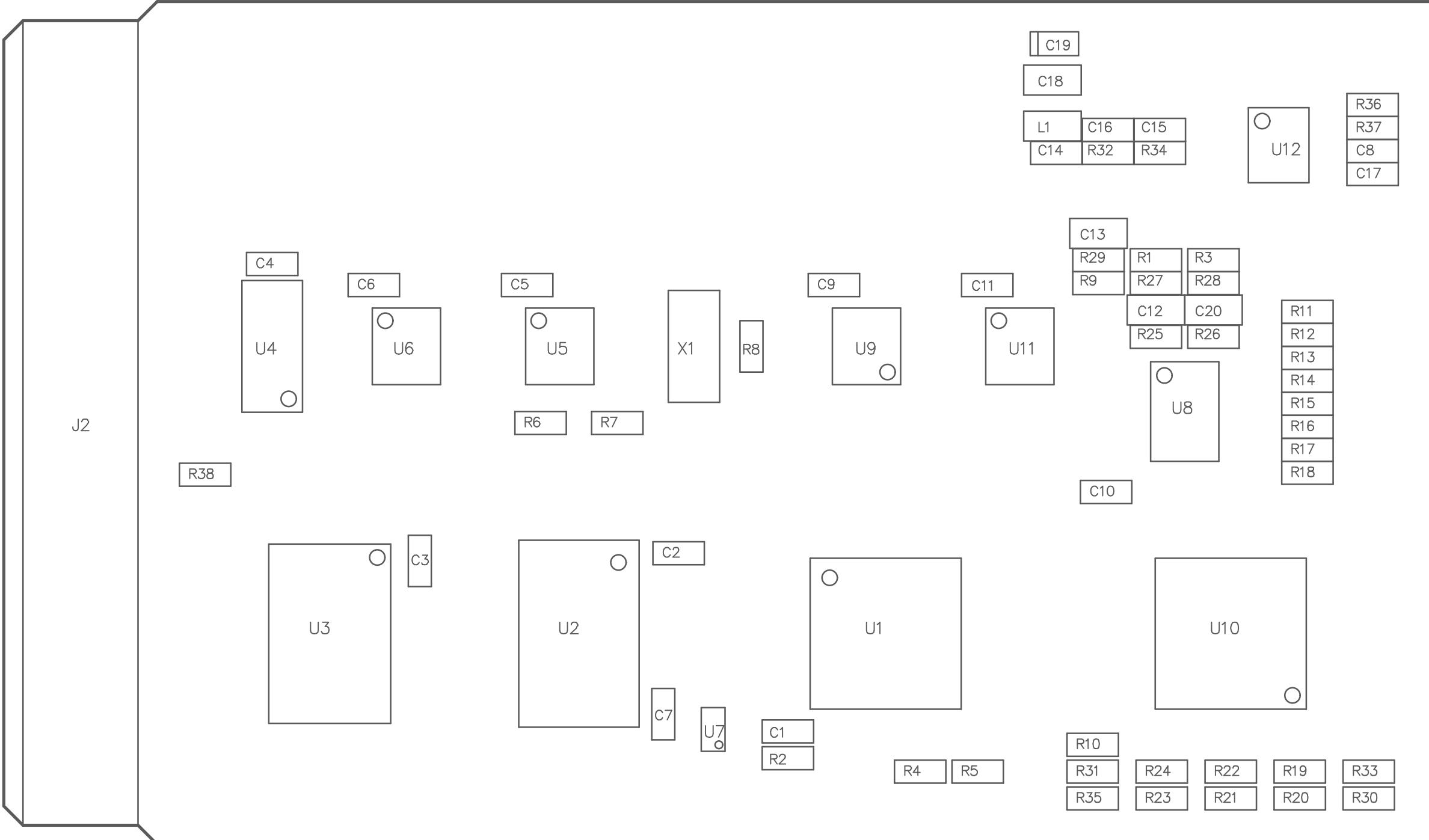


Bob D Design
smari minna
v0.3

ENGINEER: Chris Hettrick	PROJECT TITLE: smari minna	
PCB DESIGNER: Chris Hettrick		
DATE: 2012-08-20	COMPANY NAME: Populate All The Resistors	
FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.3



Assembly Top
smari minna
v0.3
2012-08-20
Not To Scale



Assembly Bottom
smari minna
v0.3
2012-08-20
Not To Scale

Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	.GTS		0.4mil	Solder Resist	3.50	
Top Layer	.GTL	2.1mil				
Power Plane	.GP1	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	.GP2	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	.GBL	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	.GBS		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE

MATERIAL ->

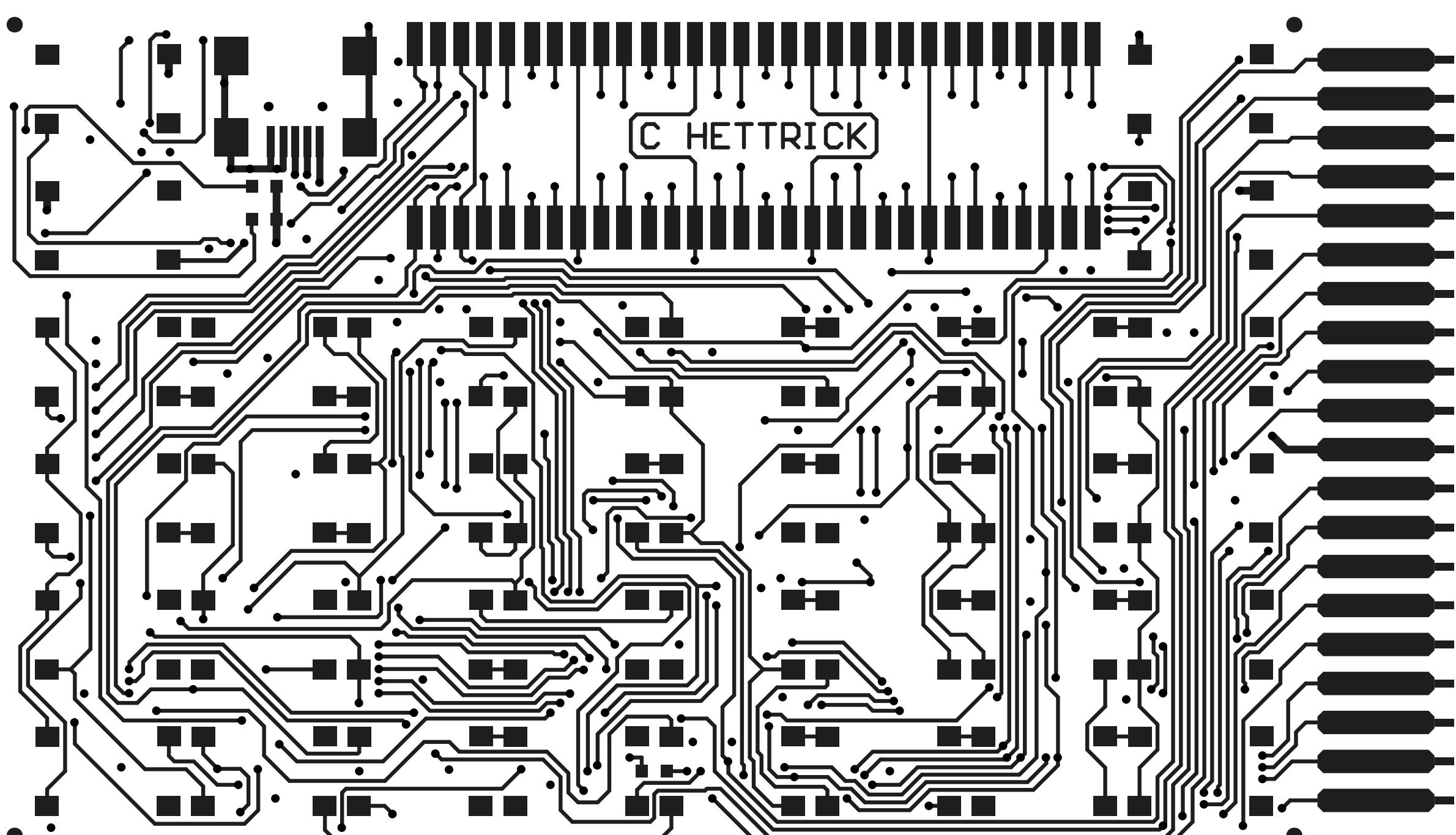
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,
 LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.
 MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.
 ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.
 THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED)
 SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS
 ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING
 SHALL BE IMMERSION GOLD FINISH.
 SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK.
 THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD.
 UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON
 THE TOP SIDE OF THE BOARD

TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%.
 CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA.
 HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005"
 HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING
 REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM.
 SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE.
 MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.



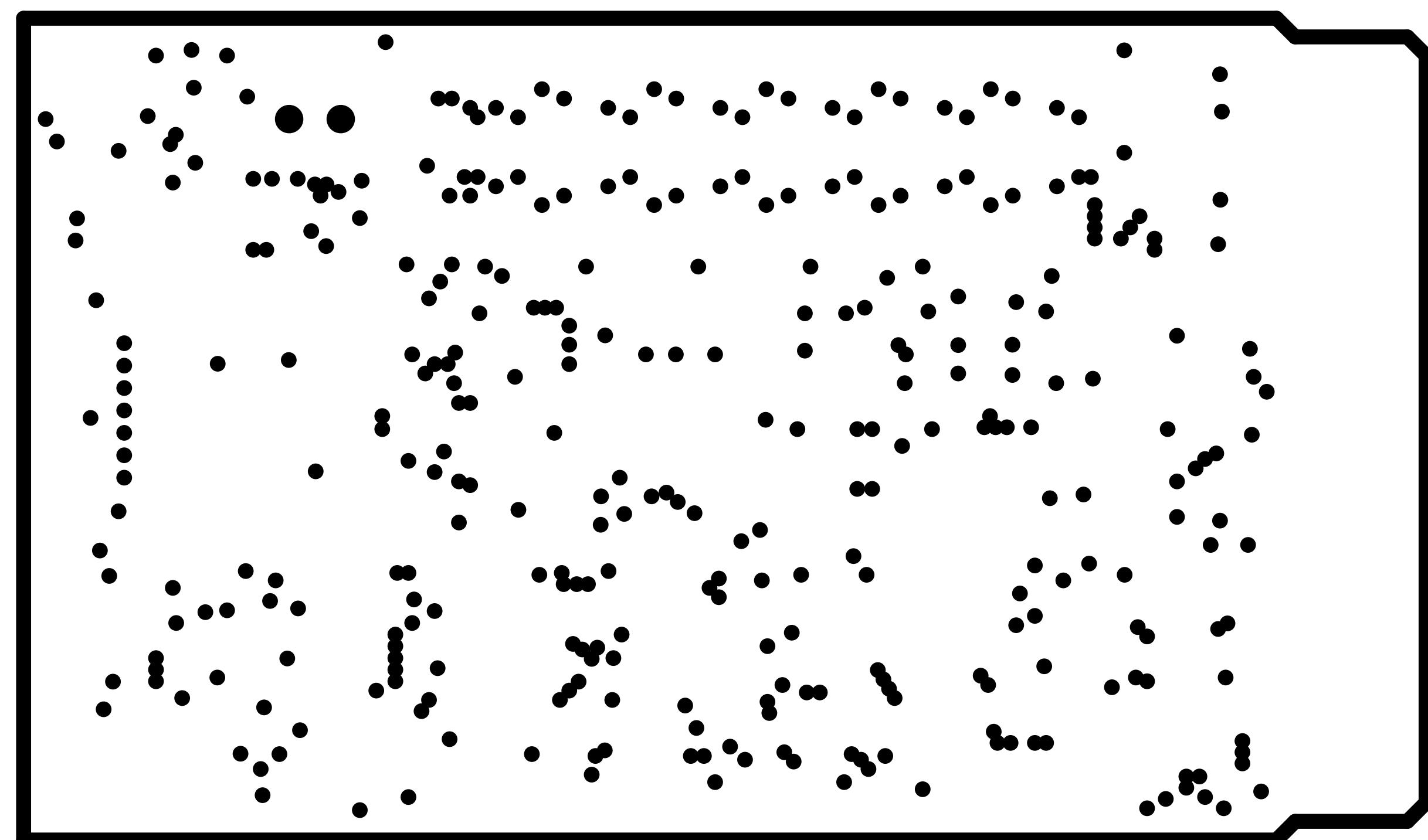
Top Layer
 smari minna
 v0.3

	ENGINEER: Chris Hettrick	PROJECT TITLE:	
	PCB DESIGNER: Chris Hettrick		smari minna
	DATE: 2012-08-20	COMPANY NAME: Populate All The Resistors	
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.3

Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	<.GTS>		0.4mil	Solder Resist	3.50	
Top Layer	<.GTL>	2.1mil				
Power Plane	<.GP1>	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	<.GP2>	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	<.GBL>	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	<.GBS>		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE



MATERIAL ->

MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT, LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26. MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING. ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS. THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED) SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING SHALL BE IMMERSION GOLD FINISH. SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK. THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD. UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON THE TOP SIDE OF THE BOARD

TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%. CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA. HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005" HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM. SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE. MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.

Power Plane
smari minna
v0.3

	ENGINEER: Chris Hettrick	PROJECT TITLE:		
	PCB DESIGNER: Chris Hettrick			smari minna
	DATE: 2012-08-20		COMPANY NAME: Populate All The Resistors	
	FILE NAME: minna_pcb.PcbDoc		COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.3

Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	<.GTS>		0.4mil	Solder Resist	3.50	
Top Layer	<.GTL>	2.1mil				
Power Plane	<.GP1>	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	<.GP2>	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	<.GBL>	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	<.GBS>		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE

MATERIAL ->

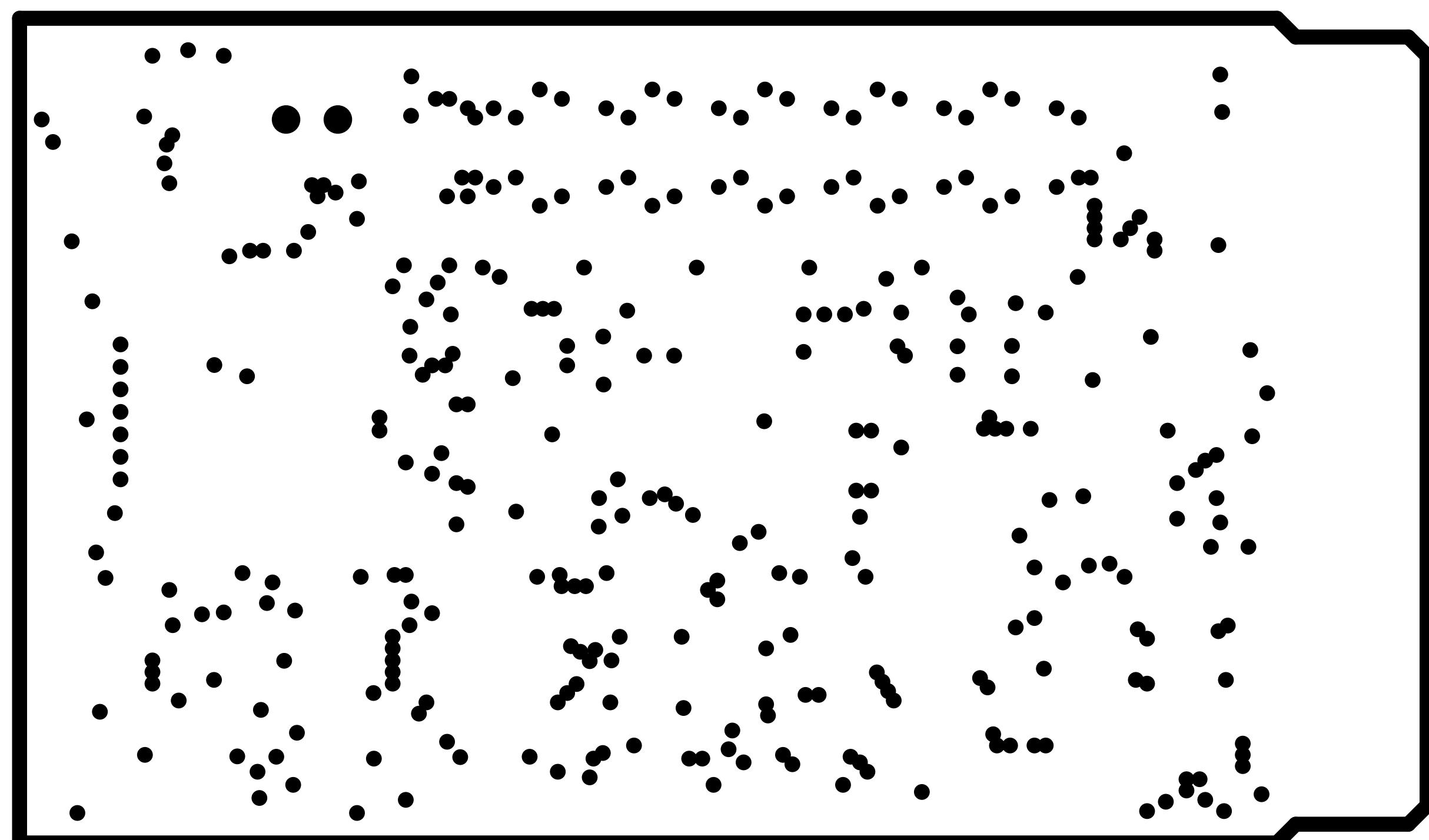
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,
LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.
MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.
THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED)
SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS
ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING
SHALL BE IMMERSION GOLD FINISH.
SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK.
THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD.
UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON
THE TOP SIDE OF THE BOARD

TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%.
CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA.
HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005"
HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING
REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM.
SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE.
MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.



Ground Plane
smari minna
v0.3

	ENGINEER: Chris Hettrick	PROJECT TITLE: smari minna
	PCB DESIGNER: Chris Hettrick	
	DATE: 2012-08-20	
	FILE NAME: minna_pcb.PcbDoc	COMPANY NAME: Populate All The Resistors

Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	.GTS		0.4mil	Solder Resist	3.50	
Top Layer	.GTL	2.1mil				
Power Plane	.GP1	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	.GP2	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	.GBL	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	.GBS		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE

MATERIAL ->

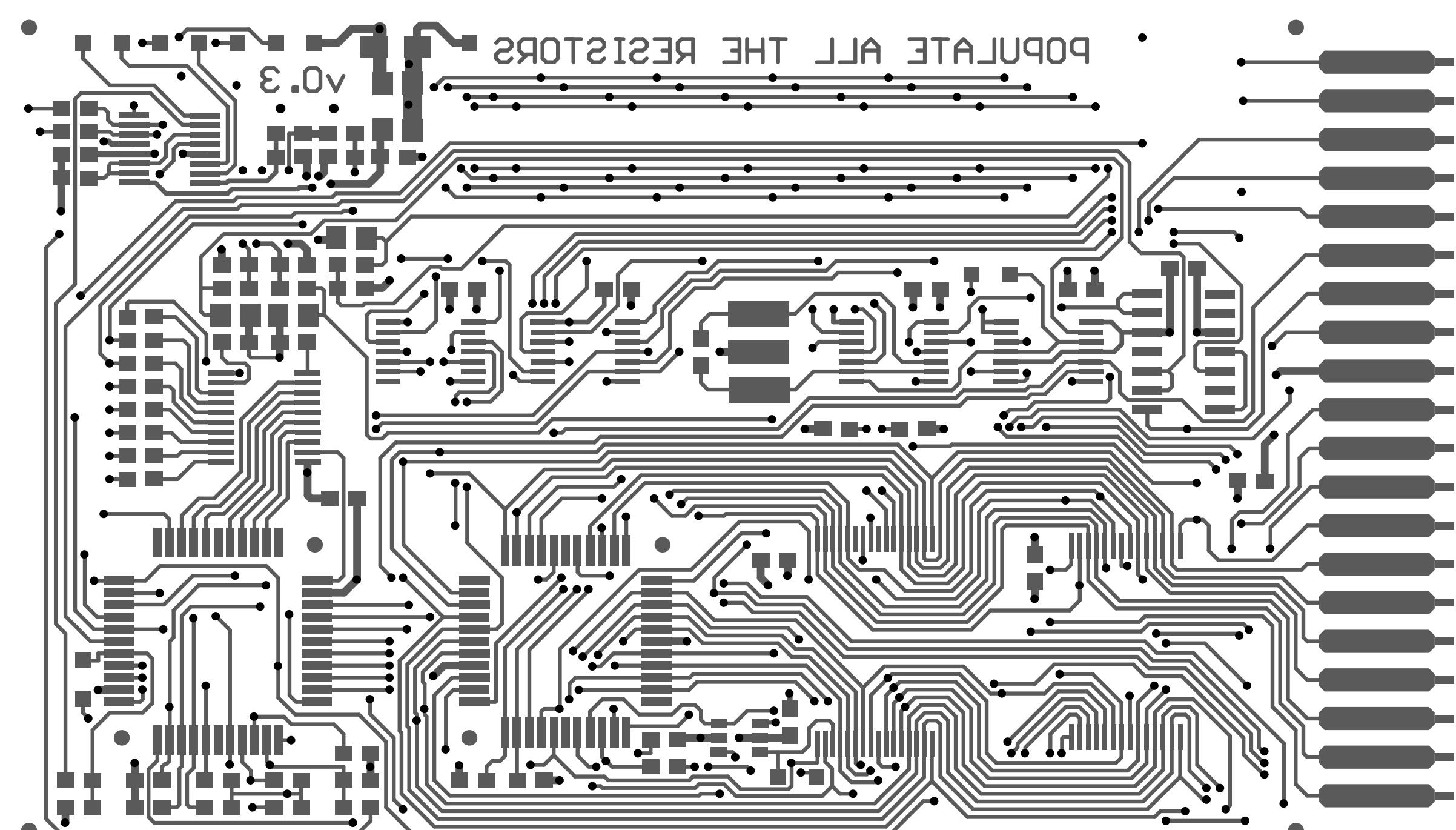
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,
LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.
MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.
THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED)
SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS
ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING
SHALL BE IMMERSION GOLD FINISH.
SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK.
THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD.
UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON
THE TOP SIDE OF THE BOARD

TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%.
CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA.
HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005"
HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING
REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM.
SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE.
MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.



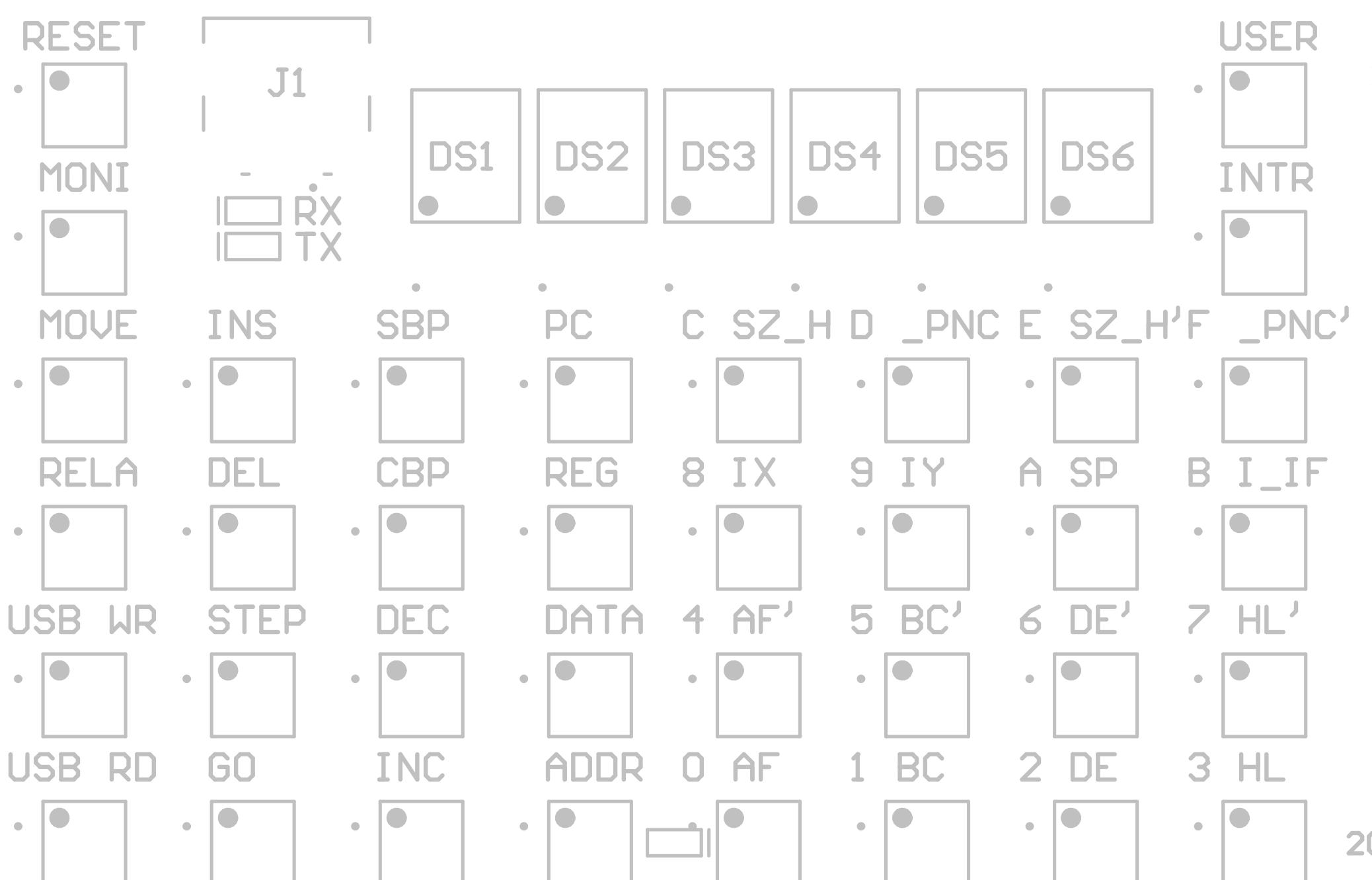
Bottom Layer
smari minna
v0.3

ENGINEER: Chris Hettrick	PROJECT TITLE: smari minna
PCB DESIGNER: Chris Hettrick	
DATE: 2012-08-20	COMPANY NAME: Populate All The Resistors
FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com
	REVISION: v0.3

Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	.GTS		0.4mil	Solder Resist	3.50	
Top Layer	.GTL	2.1mil				
Power Plane	.GP1	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	.GP2	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	.GBL	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	.GBS		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE



MATERIAL ->

MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT, LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26. MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING. ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS. THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED) SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING SHALL BE IMMERSION GOLD FINISH. SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK. THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD. UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON THE TOP SIDE OF THE BOARD

TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%. CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA. HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005" HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM. SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE. MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.

Top Overlay
smari minna
v0.3

	ENGINEER: Chris Hettrick	PROJECT TITLE:		
	PCB DESIGNER: Chris Hettrick	smari minna		
	DATE: 2012-08-20	COMPANY NAME: Populate All The Resistors		
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.3	

Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	.GTS		0.4mil	Solder Resist	3.50	
Top Layer	.GTL	2.1mil				
Power Plane	.GP1	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	.GP2	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	.GBL	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	.GBS		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE

MATERIAL ->

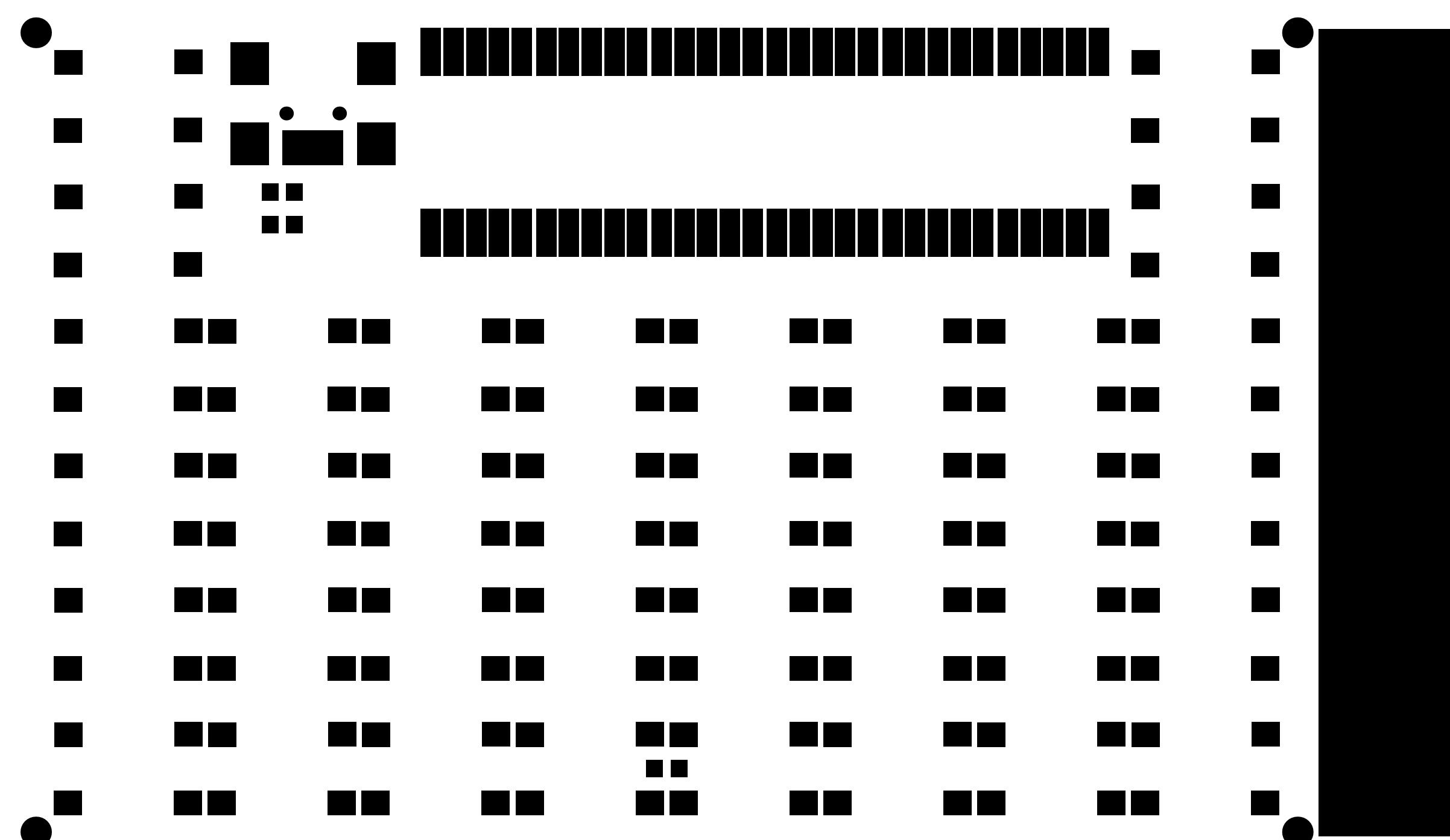
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,
LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.
MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.
THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED)
SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS
ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING
SHALL BE IMMERSION GOLD FINISH.
SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK.
THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD.
UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON
THE TOP SIDE OF THE BOARD

TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%.
CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA.
HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005"
HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING
REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM.
SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE.
MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.



Top Solder
smari minna
v0.3

	ENGINEER: Chris Hettrick	PROJECT TITLE:		
	PCB DESIGNER: Chris Hettrick			smari minna
	DATE: 2012-08-20		COMPANY NAME: Populate All The Resistors	
	FILE NAME: minna_pcb.PcbDoc		COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.3

Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	<.GTS>		0.4mil	Solder Resist	3.50	
Top Layer	<.GTL>	2.1mil				
Power Plane	<.GP1>	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	<.GP2>	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	<.GBL>	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	<.GBS>		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE

MATERIAL ->

MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,
 LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.
 MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.
 ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.
 THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED)
 SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS
 ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING
 SHALL BE IMMERSION GOLD FINISH.
 SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK.
 THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD.
 UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON
 THE TOP SIDE OF THE BOARD

TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%.
 CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA.
 HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005"
 HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING
 REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM.
 SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE.
 MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.

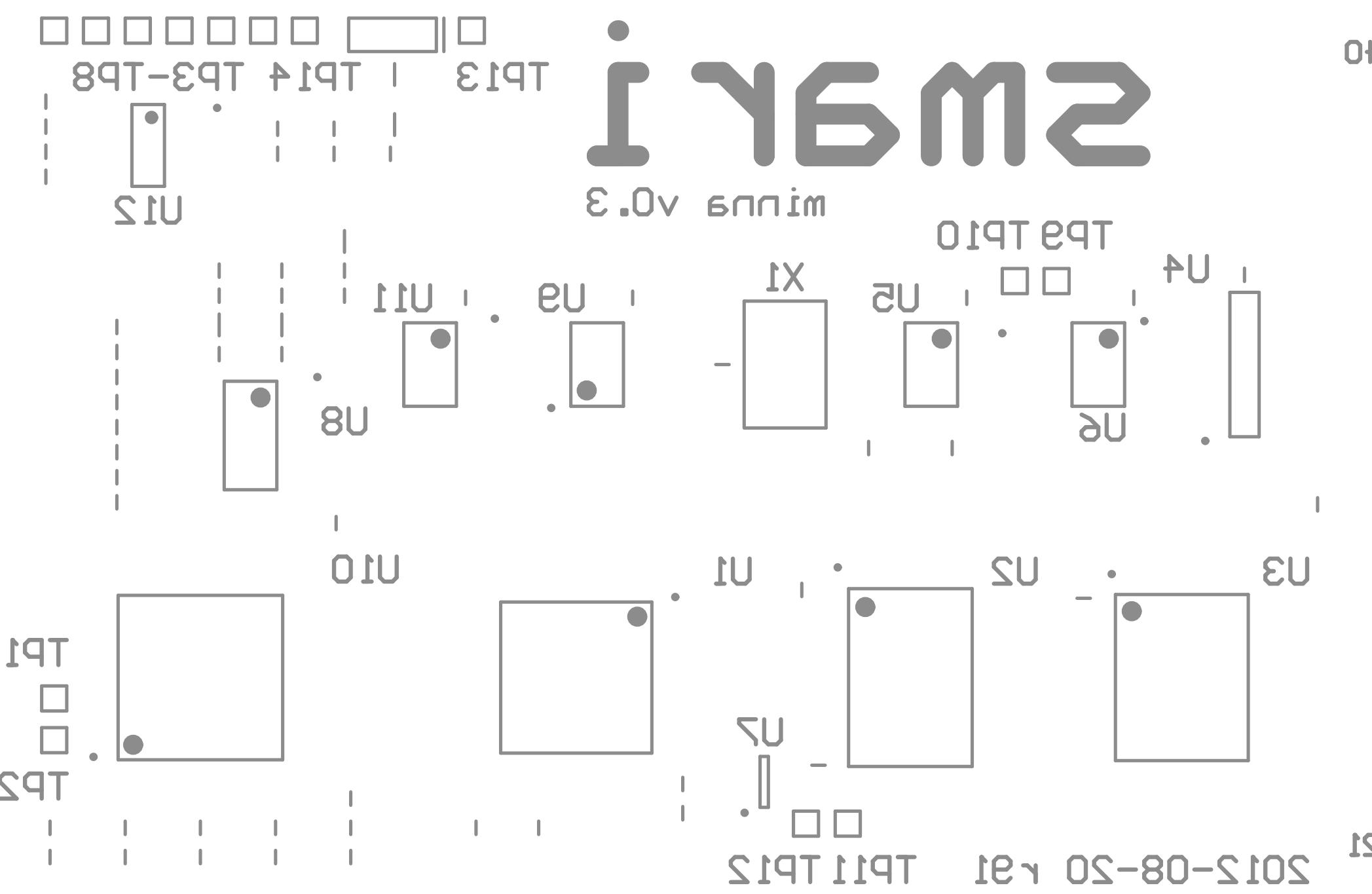
Top Paste
 smari minna
 v0.3

	ENGINEER: Chris Hettrick	PROJECT TITLE:		
	PCB DESIGNER: Chris Hettrick			smari minna
	DATE: 2012-08-20		COMPANY NAME: Populate All The Resistors	
	FILE NAME: minna_pcb.PcbDoc		COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.3

Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	.GTS		0.4mil	Solder Resist	3.50	
Top Layer	.GTL	2.1mil				
Power Plane	.GP1	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	.GP2	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	.GBL	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	.GBS		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE



Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	.GTS		0.4mil	Solder Resist	3.50	
Top Layer	.GTL	2.1mil				
Power Plane	.GP1	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	.GP2	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	.GBL	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	.GBS		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE

MATERIAL ->

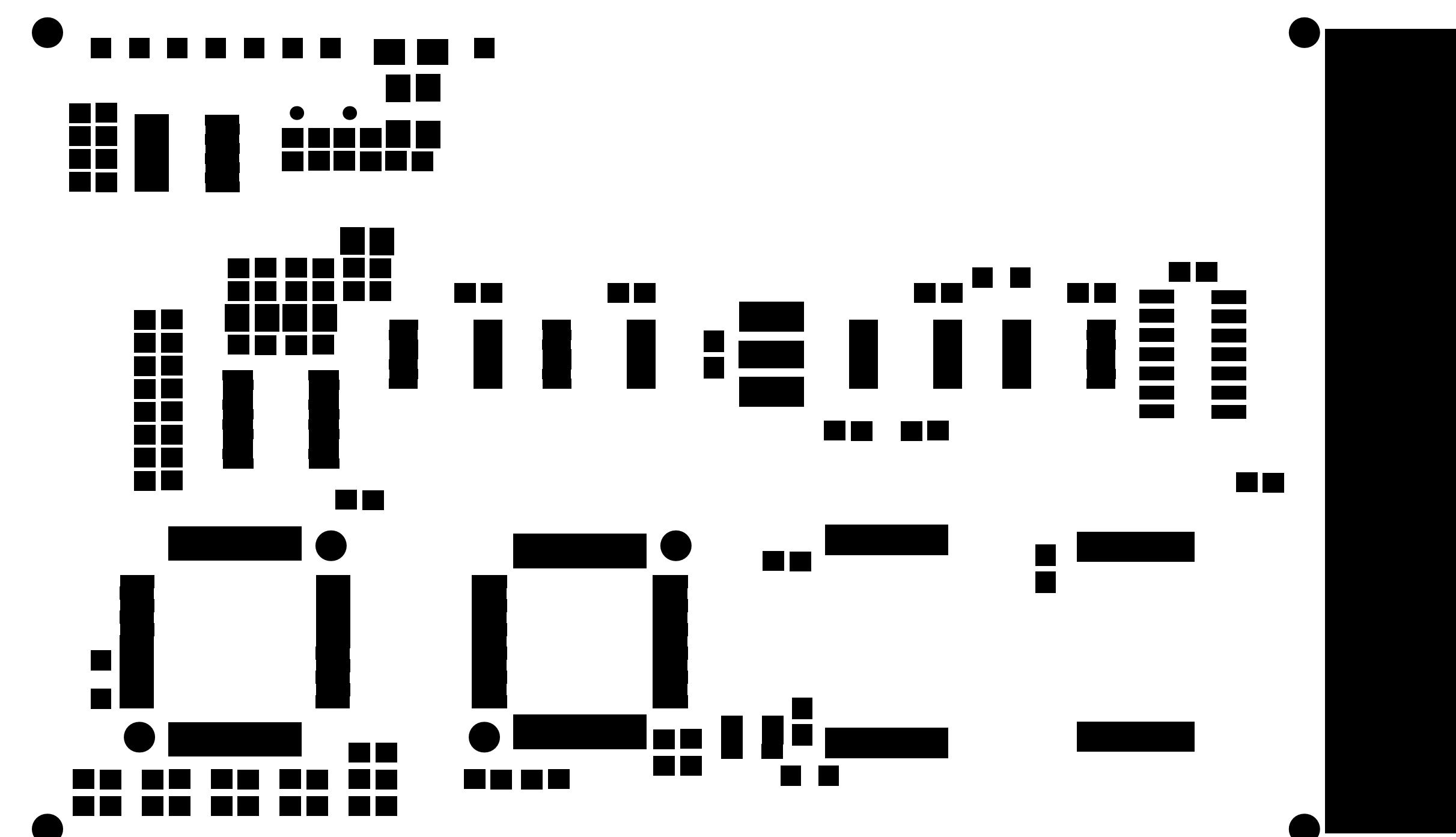
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,
LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.
MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.
THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED)
SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS
ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING
SHALL BE IMMERSION GOLD FINISH.
SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK.
THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD.
UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON
THE TOP SIDE OF THE BOARD

TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%.
CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA.
HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005"
HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING
REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM.
SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE.
MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.



Bottom Solder
smari minna
v0.3

	ENGINEER: Chris Hettrick	PROJECT TITLE:		
	PCB DESIGNER: Chris Hettrick			smari minna
	DATE: 2012-08-20	COMPANY NAME: Populate All The Resistors		
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.3	

Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	<.GTS>		0.4mil	Solder Resist	3.50	
Top Layer	<.GTL>	2.1mil				
Power Plane	<.GP1>	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	<.GP2>	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	<.GBL>	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	<.GBS>		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE

MATERIAL ->

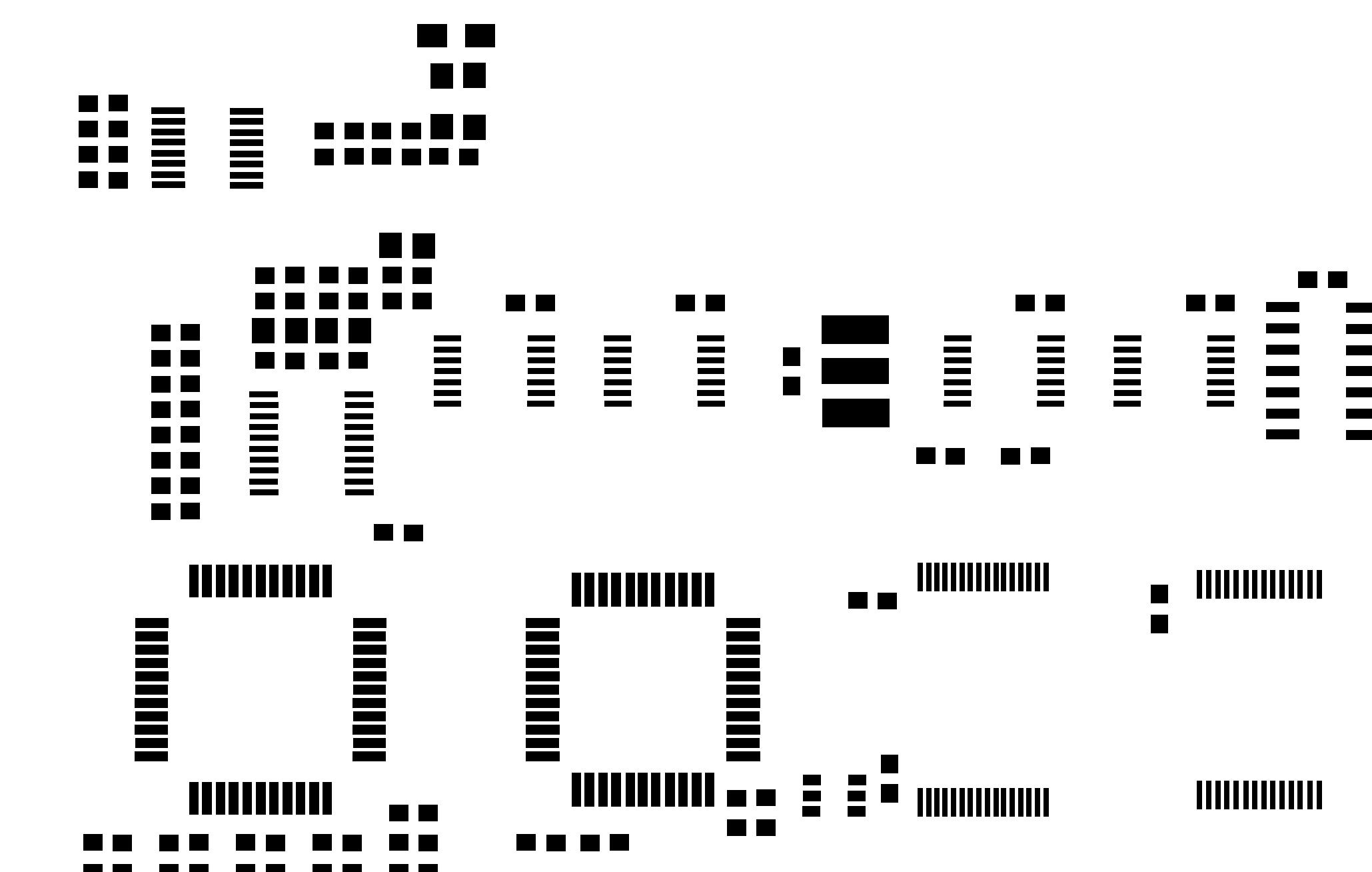
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,
LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.
MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.
THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED)
SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS
ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING
SHALL BE IMMERSION GOLD FINISH.
SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK.
THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD.
UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON
THE TOP SIDE OF THE BOARD

TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%.
CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA.
HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005"
HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING
REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM.
SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE.
MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.



Bottom Paste
smari minna
v0.3

	ENGINEER: Chris Hettrick	PROJECT TITLE:		
	PCB DESIGNER: Chris Hettrick			smari minna
	DATE: 2012-08-20	COMPANY NAME: Populate All The Resistors		
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.3	



Symbol	Hit Count	Tool Size	Plated	Hole Type
□	368	12mil (0.3048mm)	PTH	Round
○	2	35.433mil (0.9mm)	NPTH	Round
	370 Total			

Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	.GTS		0.4mil	Solder Resist	3.50	
Top Layer	.GTL	2.1mil				
Power Plane	.GP1	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	.GP2	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	.GBL	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	.GBS		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE

MATERIAL ->

MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT, LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26. MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING. ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS. THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED) SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING SHALL BE IMMERSION GOLD FINISH. SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK. THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD. UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON THE TOP SIDE OF THE BOARD

TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%. CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA. HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005" HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM. SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE. MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.

SEE DETAIL A

Drill Drawing
smari minna
v0.3

ENGINEER: Chris Hettrick	PROJECT TITLE: smari minna
PCB DESIGNER: Chris Hettrick	
DATE: 2012-08-20	COMPANY NAME: Populate All The Resistors
FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com
	REVISION: v0.3

Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	<.GTS>		0.4mil	Solder Resist	3.50	
Top Layer	<.GTL>	2.1mil				
Power Plane	<.GP1>	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	<.GP2>	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	<.GBL>	2.1mil		FR-4	4.20	Core
Bottom Solder Mask	<.GBS>		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE

MATERIAL ->

MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,
 LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.
 MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.
 ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.
 THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED)
 SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS
 ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING
 SHALL BE IMMERSION GOLD FINISH.
 SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK.
 THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD.
 UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON
 THE TOP SIDE OF THE BOARD

TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%.
 CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA.
 HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005"
 HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING
 REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM.
 SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE.
 MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.

Drill Guide
 smari minna
 v0.3

	ENGINEER: Chris Hettrick	PROJECT TITLE:	
	PCB DESIGNER: Chris Hettrick		smari minna
	DATE: 2012-08-20	COMPANY NAME: Populate All The Resistors	
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.3

Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	.GTS		0.4mil	Solder Resist	3.50	
Top Layer	.GTL	2.1mil				
Power Plane	.GP1	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	.GP2	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	.GBL	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	.GBS		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE

MATERIAL ->

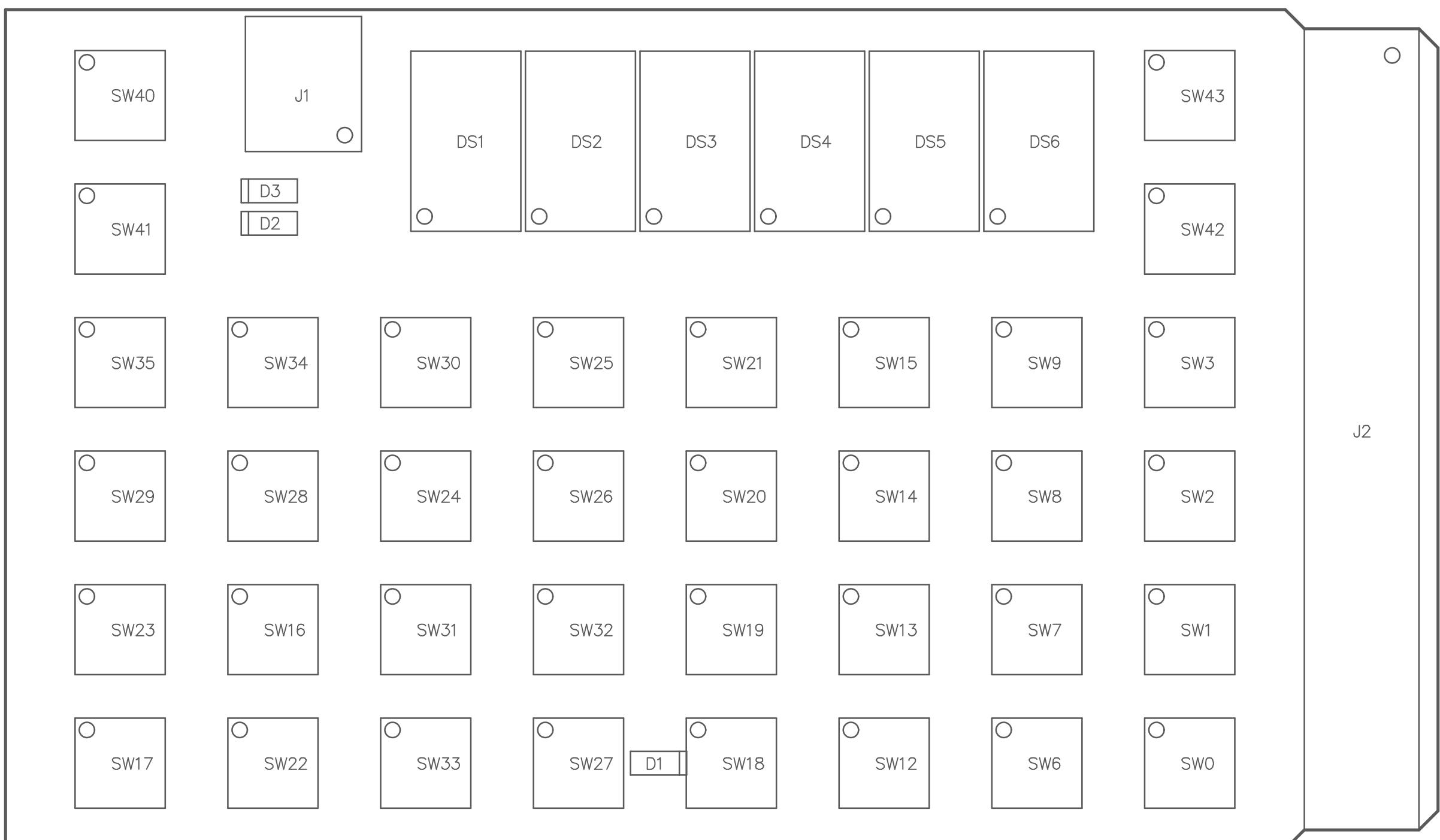
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT, LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26. MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING. ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS. THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED) SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING SHALL BE IMMERSION GOLD FINISH. SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK. THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD. UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON THE TOP SIDE OF THE BOARD

TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%. CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA. HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005" HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM. SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE. MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.



Assembly Top
smari minna
v0.3
2012-08-20
Not To Scale

	ENGINEER: Chris Hettrick	PROJECT TITLE: smari minna
	PCB DESIGNER: Chris Hettrick	
	DATE: 2012-08-20	COMPANY NAME: Populate All The Resistors
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com

Layer Stack Up Detail for: minna_pcb.PcbDoc

Layer Name	Gerber Document	Copper Thickness	Dielectric Height	Dielectric Material	Dielectric Constant	Dielectric Type
Top Solder Mask	.GTS		0.4mil	Solder Resist	3.50	
Top Layer	.GTL	2.1mil				
Power Plane	.GP1	0.7mil	14mil	FR-4	4.20	Core
Ground Plane	.GP2	0.7mil	28mil	FR-4	4.20	PrePreg
Bottom Layer	.GBL	2.1mil	14mil	FR-4	4.20	Core
Bottom Solder Mask	.GBS		0.4mil	Solder Resist	3.50	

ALL HOLES INDICATE FINISHED SIZE

MATERIAL ->

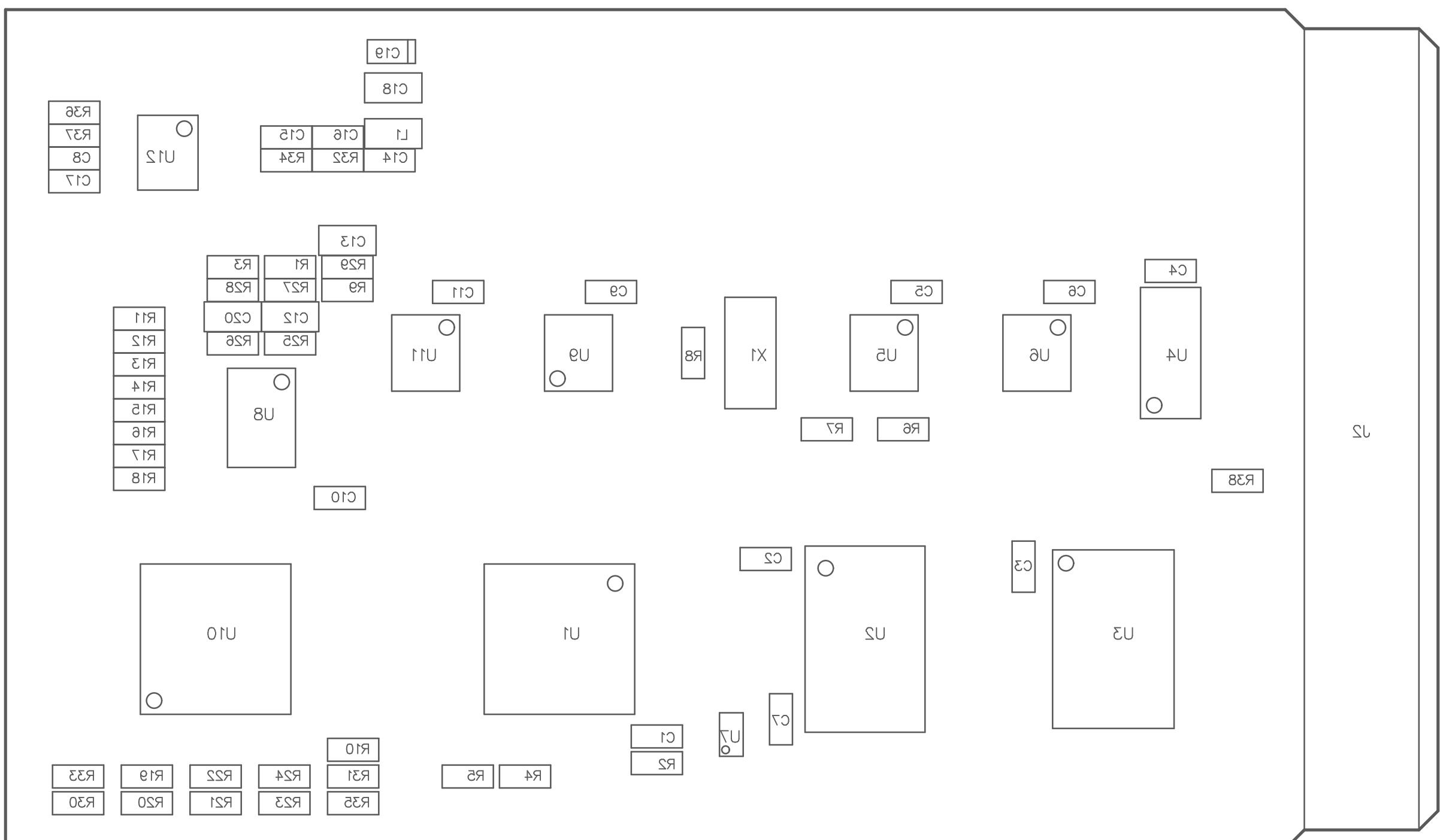
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,
LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.
MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.
THIS IS A 4 LAYER BOARD.

FINISH ->

FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED)
SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS
ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING
SHALL BE IMMERSION GOLD FINISH.
SILKSCREEN SHALL BE WHITE, PERMANENT, ORGANIC, NON-CONDUCTIVE INK.
THERE SHALL BE NO SILKSCREEN ON ANY SOLDERABLE COMPONENT PAD.
UL LOGO, MANUFACTURER'S IDENTIFICATION AND DATE CODE LETTER SHALL BE RENDERED IN ETCH ON
THE TOP SIDE OF THE BOARD

TOLERANCES ->

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%.
CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN +/- 0.001" OF GERBER DATA.
HOLE LOCATION +/- 0.003". MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005"
HOLE DIAMETER TOLERANCE IS +/- 0.003" AFTER PLATING
REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM.
SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.003") ABOVE BOARD SURFACE.
MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.



Assembly Drawing
Bottom
v0.3
2012-08-20
Not To Scale

	ENGINEER: Chris Hettrick	PROJECT TITLE: smari minna
	PCB DESIGNER: Chris Hettrick	
	DATE: 2012-08-20	COMPANY NAME: Populate All The Resistors
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com

