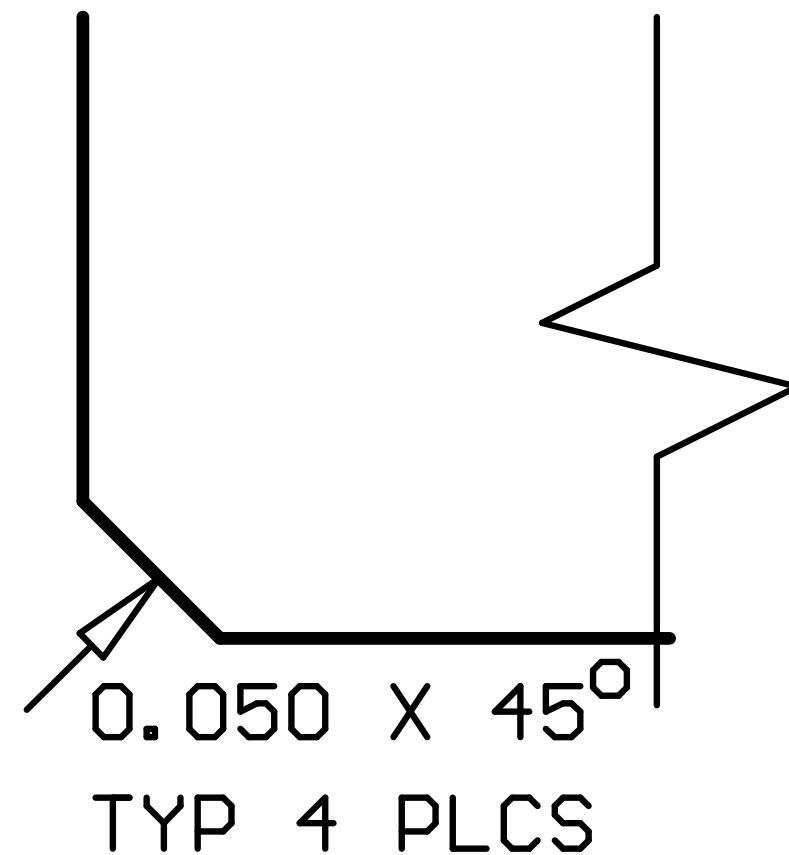


# DETAIL

**0.030 X 30° CHAMFER  
BOTH SIDES  
ENTIRE LENGTH OF  
EDGE CONNECTOR**



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		FR-4	4.20	Core
Power Plane	<.GP1>	0.7mil	14mil			
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, L.P.I., 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

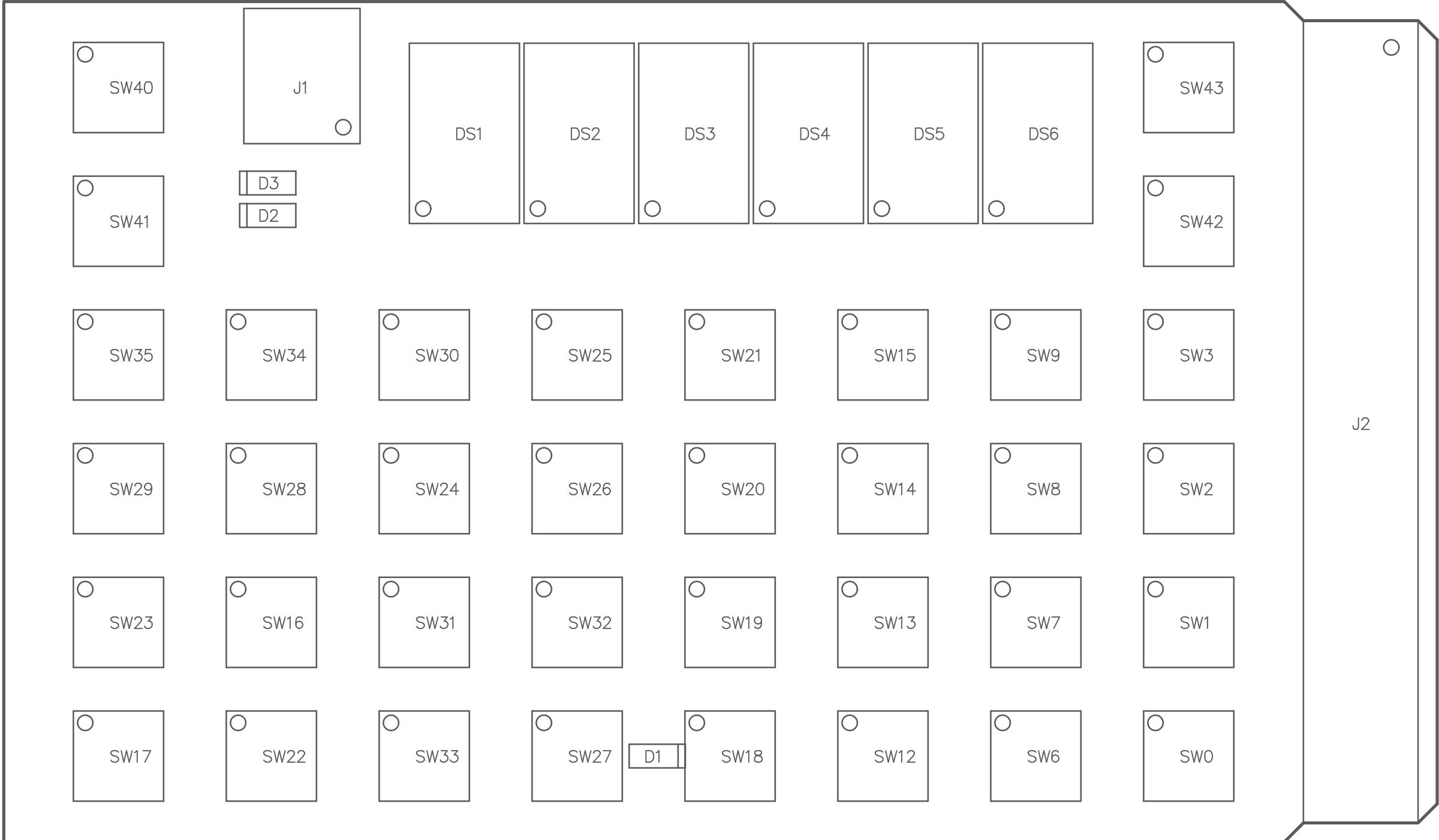
## TOLENCES =>

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

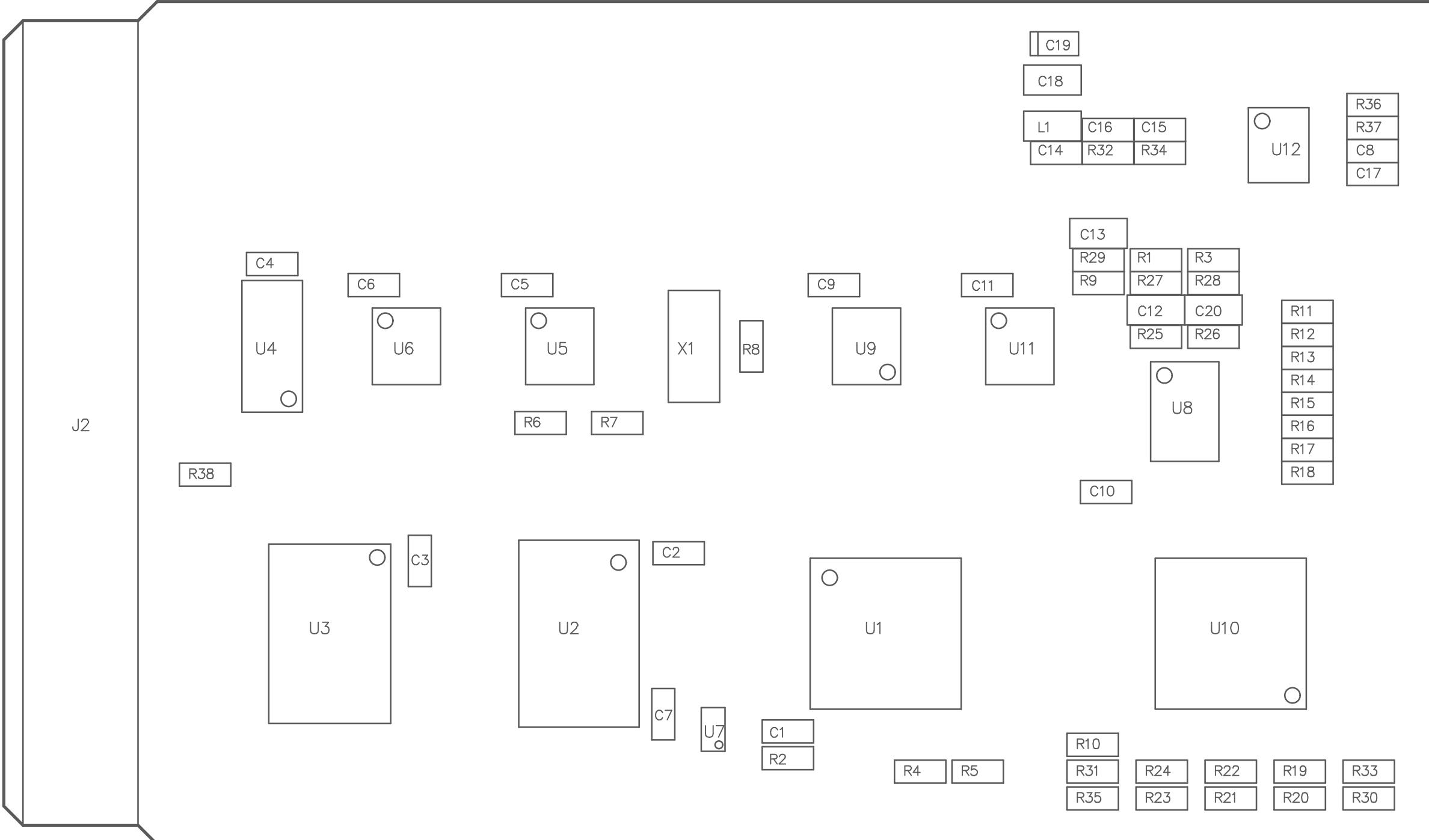
SFF DETAIL

Baptized by  
smari minna  
v0.4

ENGINEER: Chris Hettrick	PROJECT TITLE:  smari minna
PCB DESIGNER: Chris Hettrick	
DATE: 2012-08-21	COMPANY NAME: Populate All The Resistors
FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: <a href="mailto:info@populatealltheresistors.com">info@populatealltheresistors.com</a>



Assembly Top  
smari minna  
v0.4  
2012-08-21  
Not To Scale



Assembly Bottom  
smari minna  
v0.4  
2012-08-21  
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 ->

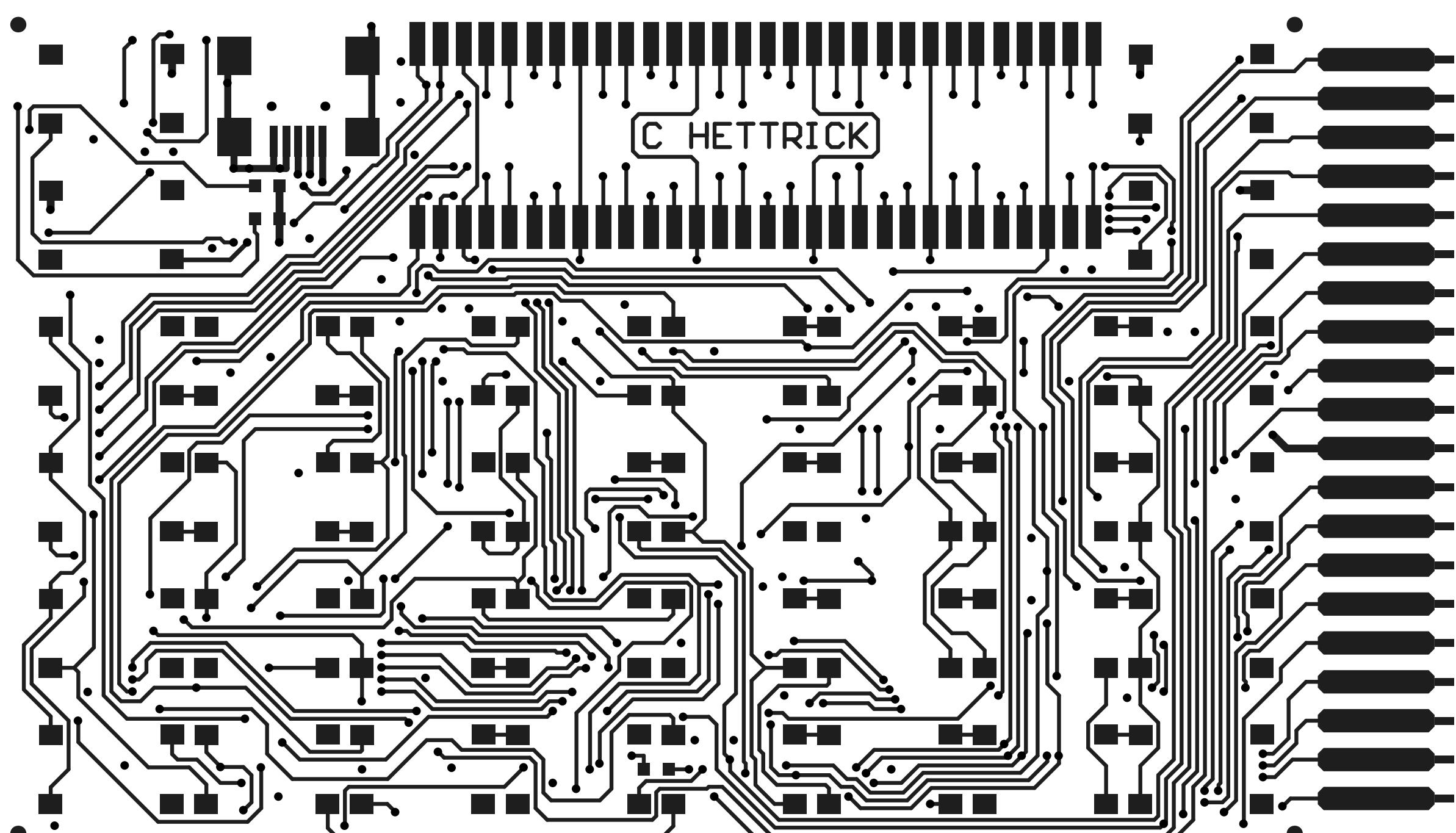
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,  
 LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.  
 MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.  
 ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.  
 THIS IS A 4 LAYER BOARD.

FINISH ->

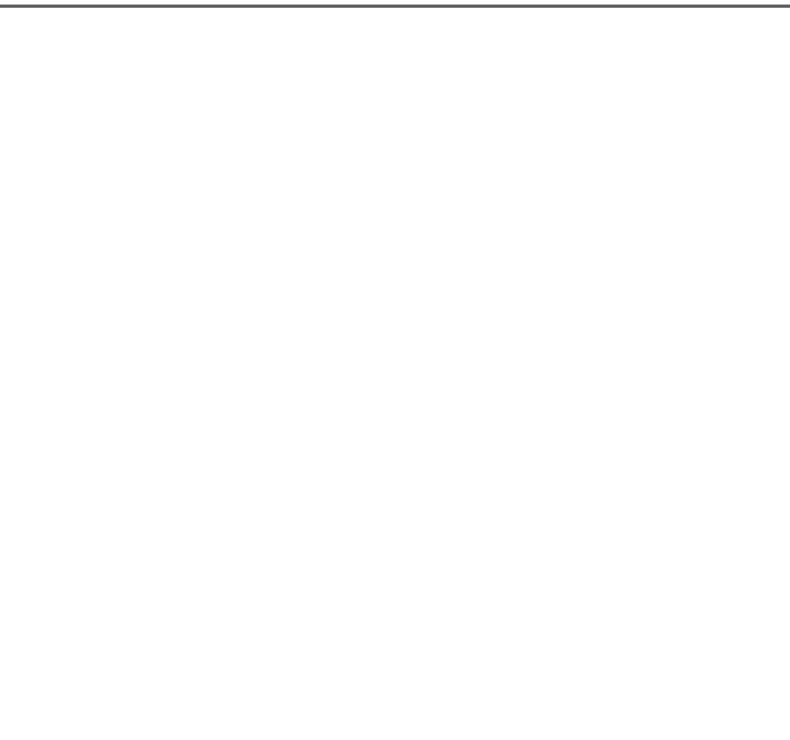
FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED)  
 SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS  
 ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING  
 SHALL BE IMMERSION GOLD FINISH.  
 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  
 v0.4

	ENGINEER: Chris Hettrick	PROJECT TITLE:  <b>smari minna</b>
	PCB DESIGNER: Chris Hettrick	
	DATE: 2012-08-21	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 ->

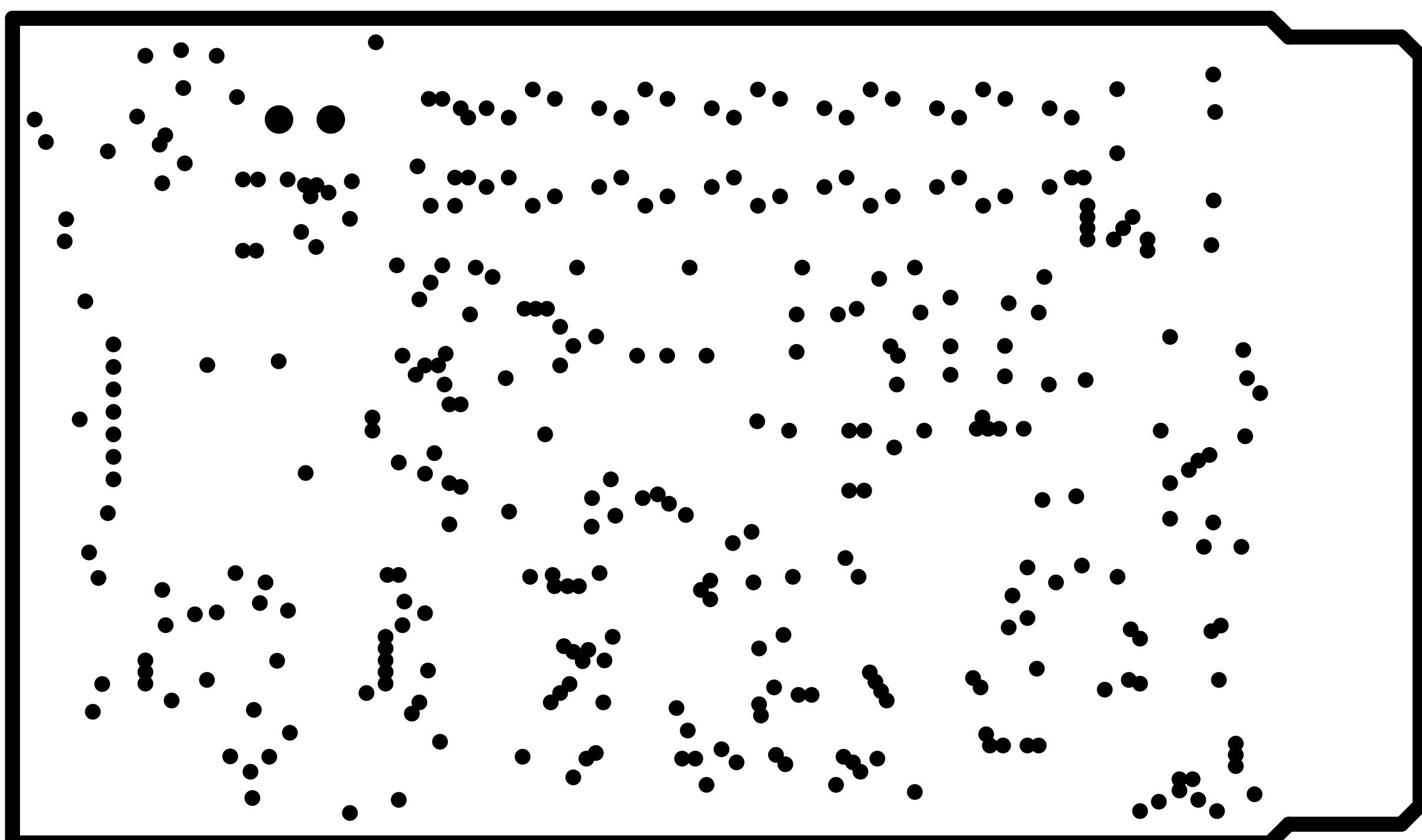
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,  
LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.  
MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.  
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.  
THIS IS A 4 LAYER BOARD.

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.



Power Plane Negative View

smari minna

v0.4

	ENGINEER: Chris Hettrick	PROJECT TITLE:  smari minna	
	PCB DESIGNER: Chris Hettrick		
	DATE: 2012-08-21	COMPANY NAME: Populate All The Resistors	
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.4

## 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 ->

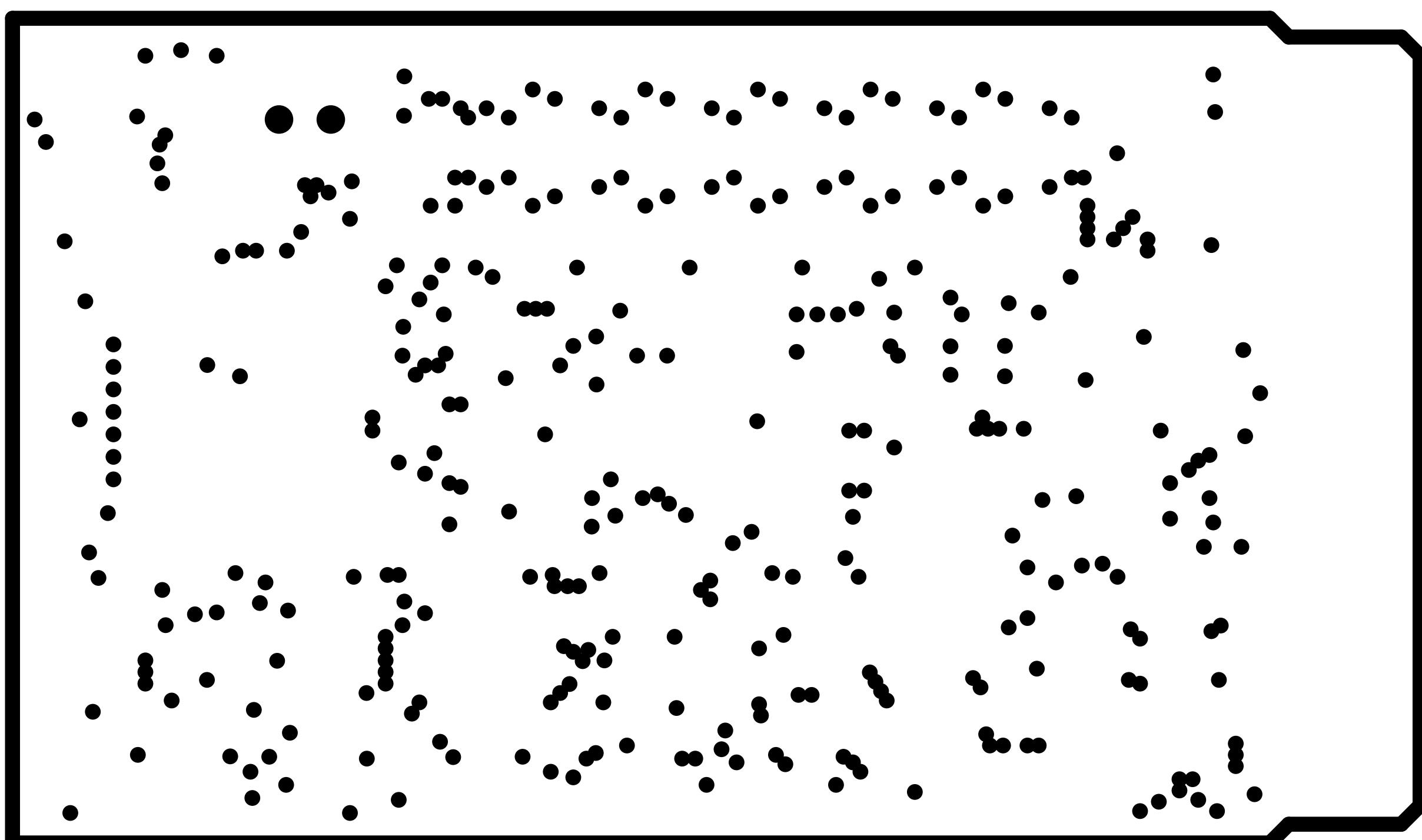
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,  
LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.  
MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.  
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.  
THIS IS A 4 LAYER BOARD.

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 Negative View

smari minna

v0.4

	ENGINEER: Chris Hettrick	PROJECT TITLE:  <b>smari minna</b>		
	PCB DESIGNER: Chris Hettrick			
	DATE: 2012-08-21	COMPANY NAME: Populate All The Resistors		
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com	REVISION:	v0.4

## 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 ->

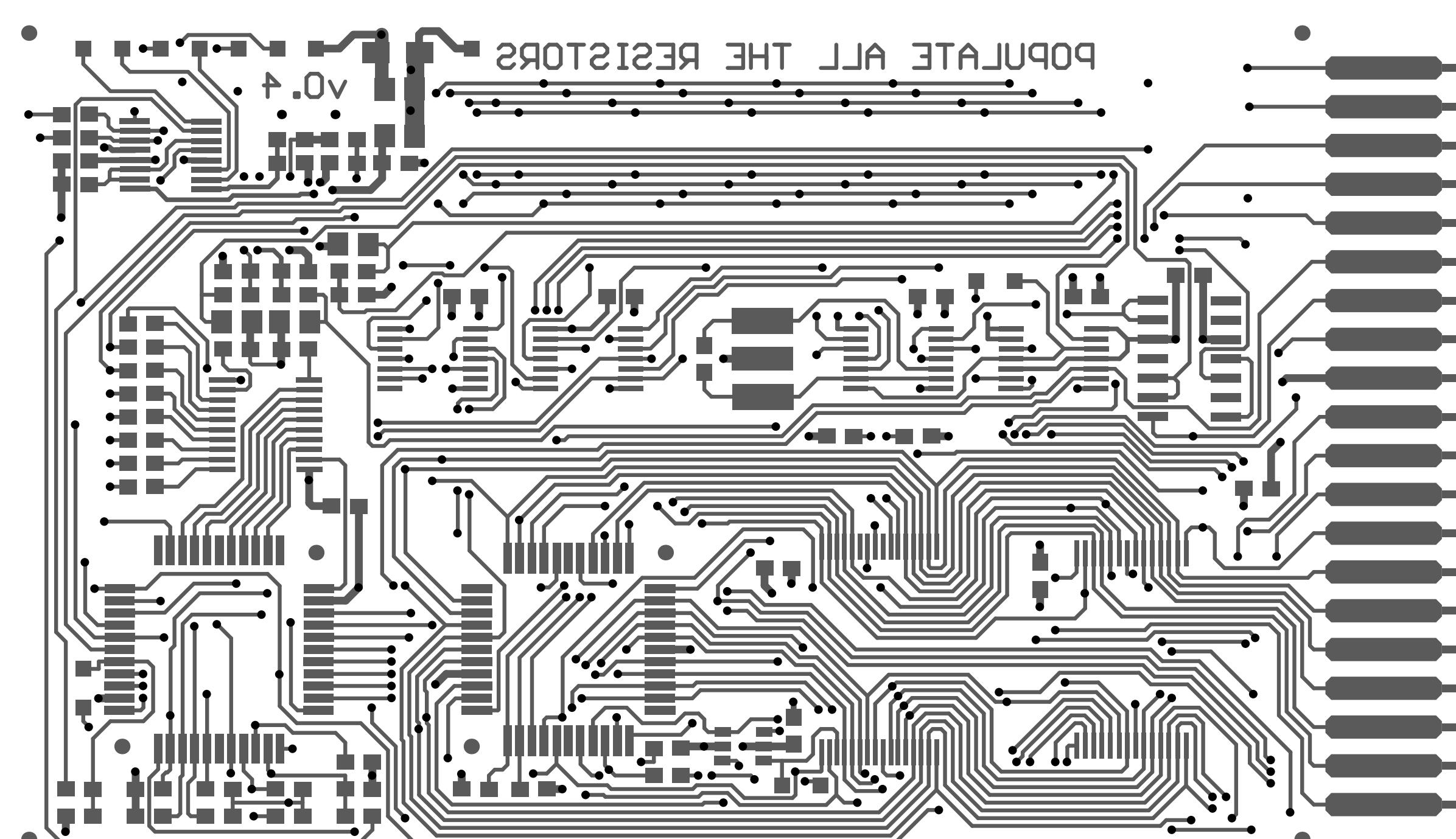
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT, LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26. MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING. ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS. THIS IS A 4 LAYER BOARD.

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.



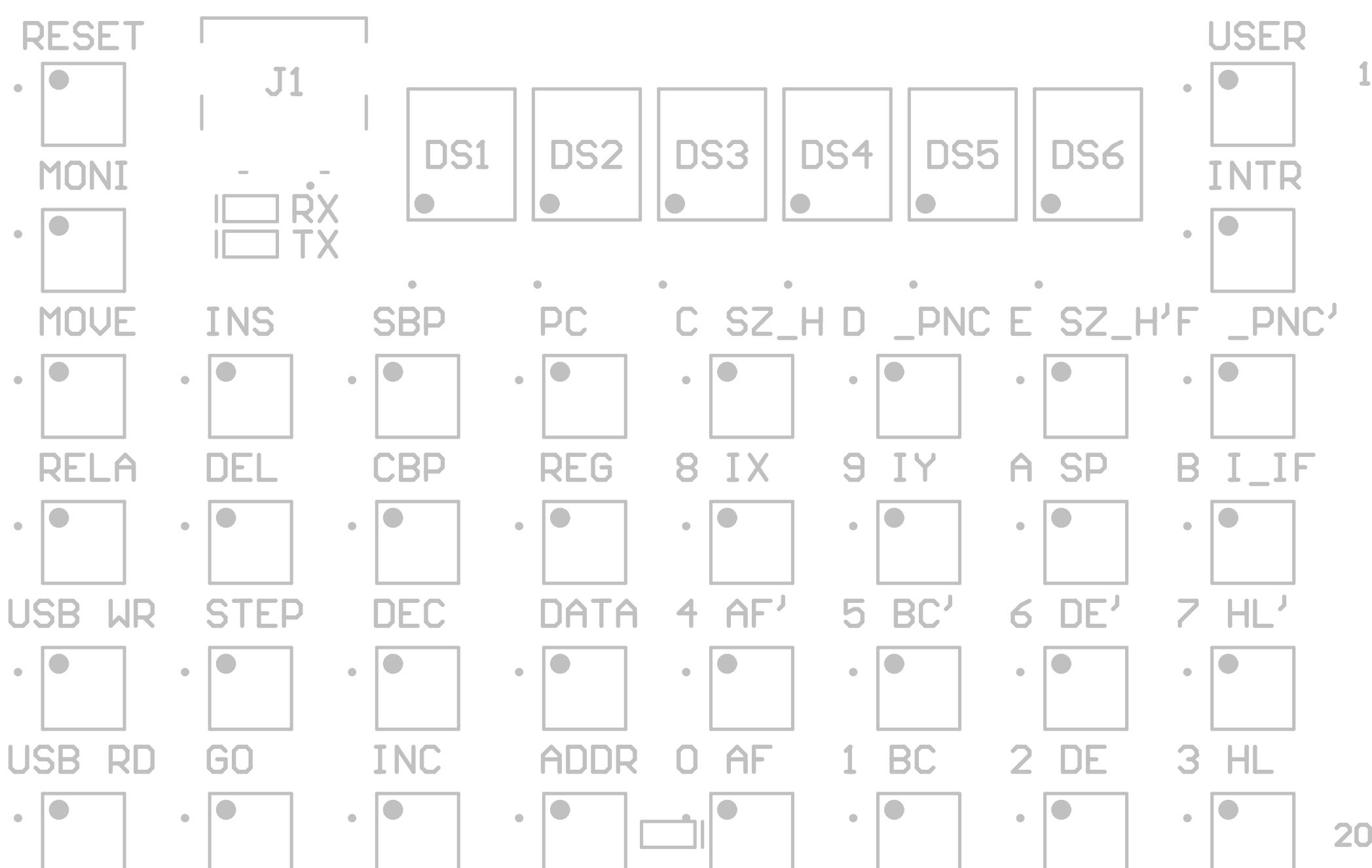
Bottom Layer  
smari minna  
v0.4

ENGINEER: Chris Hettrick	PROJECT TITLE:  smari minna
PCB DESIGNER: Chris Hettrick	
DATE: 2012-08-21	COMPANY NAME: Populate All The Resistors
FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com
	REVISION: v0.4

## 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  
v0.4

	ENGINEER: Chris Hettrick	PROJECT TITLE:  <b>smari minna</b>		
	PCB DESIGNER: Chris Hettrick			
	DATE: 2012-08-21	COMPANY NAME: Populate All The Resistors		
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.4	

## 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 ->

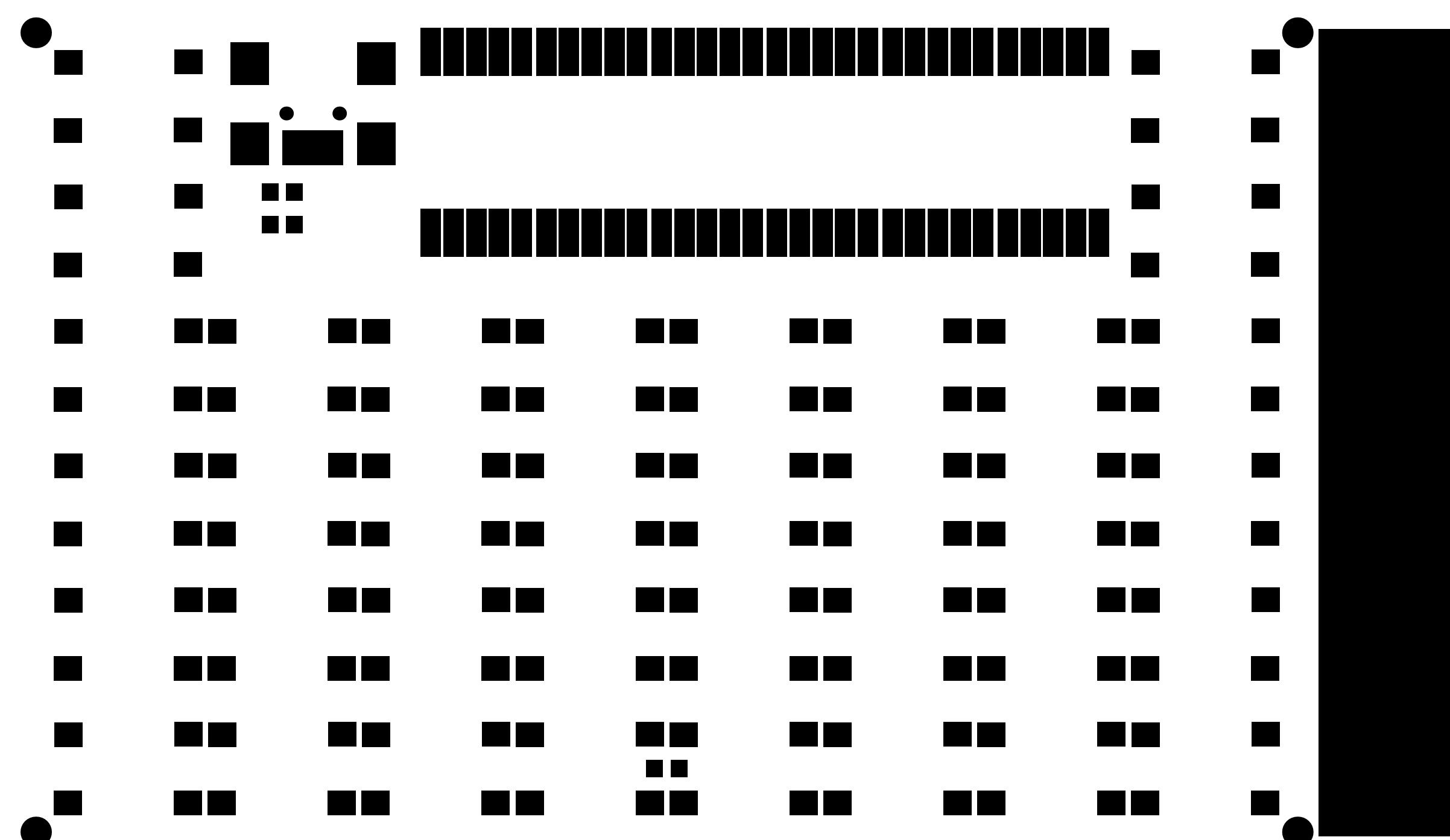
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,  
LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.  
MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.  
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.  
THIS IS A 4 LAYER BOARD.

FINISH ->

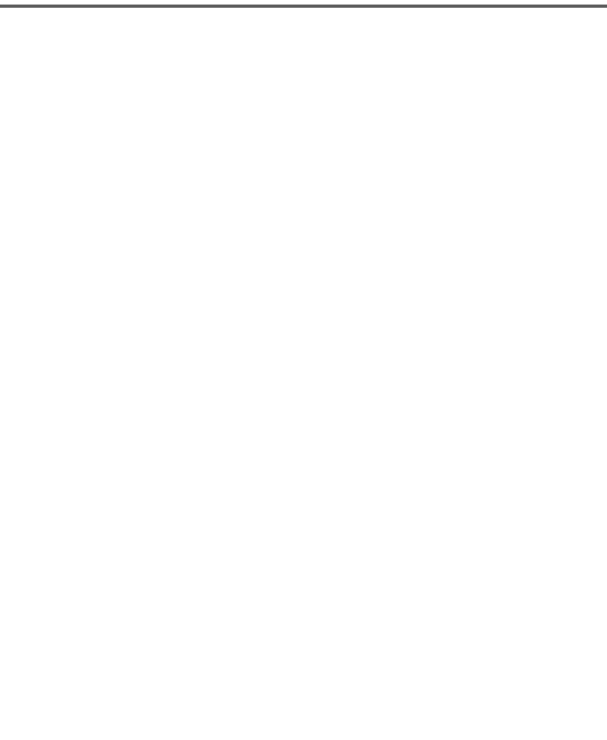
FINISHED COPPER THICKNESS TO BE 1.0 oz. EXTERNAL LAYERS, 0.5 oz. INTERNAL LAYERS (MULTI-LAYERED)  
SOLDER MASK OVER BARE COPPER, BOTH SIDES, LPI., VIOLET IN COLOR. 0.002" MAX THICKNESS  
ALL EXPOSED CONDUCTIVE PATTERN AREAS NOT COVERED WITH SOLDERMASK OR OTHER PLATING  
SHALL BE IMMERSION GOLD FINISH.  
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  
v0.4

	ENGINEER: Chris Hettrick	PROJECT TITLE:  smari minna
	PCB DESIGNER: Chris Hettrick	
	DATE: 2012-08-21	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.

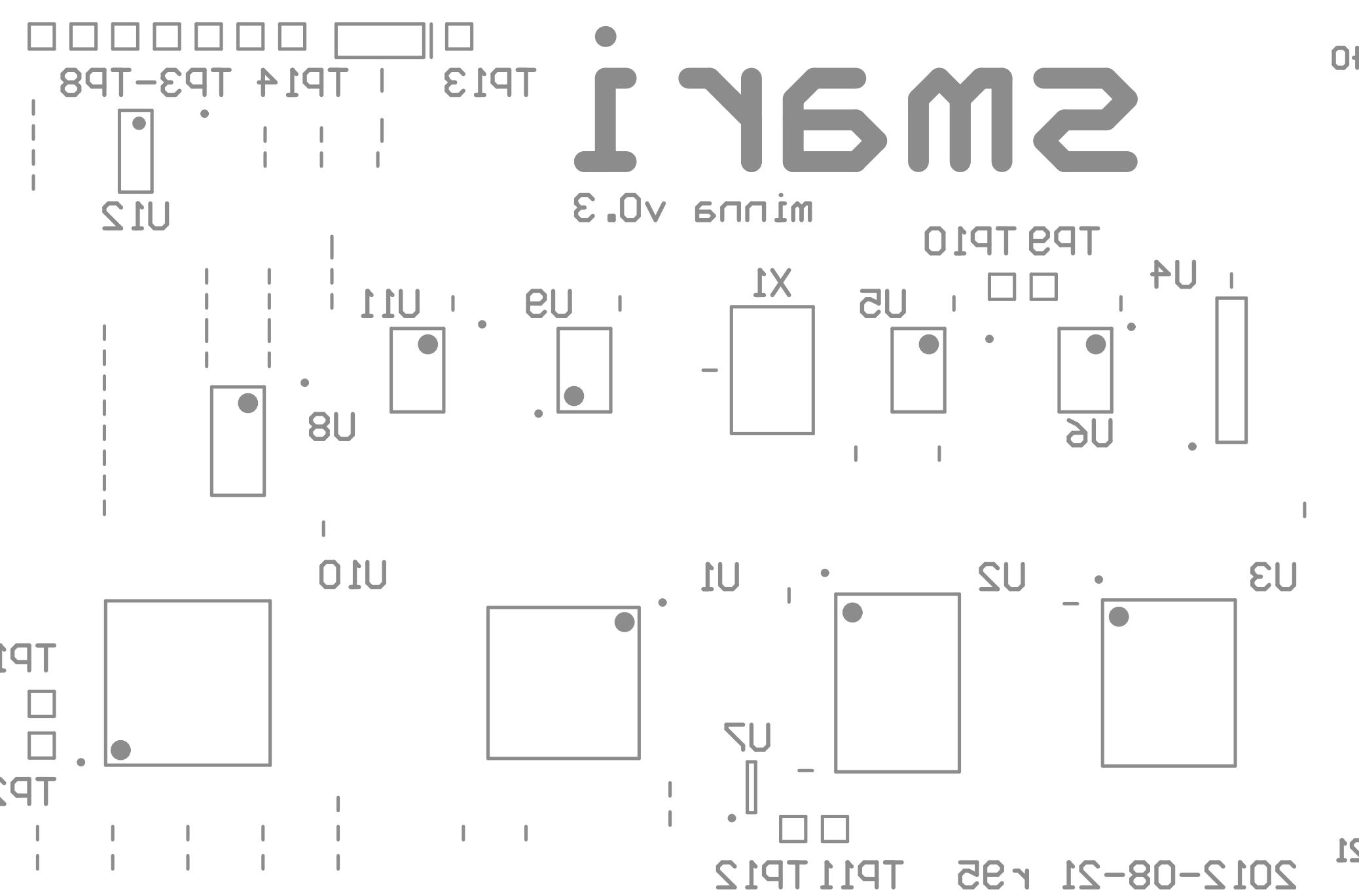
Top Paste  
smari minna  
v0.4

	ENGINEER: Chris Hettrick	PROJECT TITLE:		
	PCB DESIGNER: Chris Hettrick			smari minna
	DATE: 2012-08-21		COMPANY NAME: Populate All The Resistors	
	FILE NAME: minna_pcb.PcbDoc		COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.4

# 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		FR-4	4.20	Core
Power Plane	<.GP1>	0.7mil	14mil	FR-4	4.20	PrePreg
Ground Plane	<.GP2>	0.7mil	28mil	FR-4	4.20	Core
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**



Bottom Overlay  
smari minna  
v0.4

## 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, L.P.I., 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

TOI FRANCES ->

## TOLERANCES →

WARP OR TWIST OF BOARD SHALL NOT EXCEED 1%.  
CONDUCTOR WIDTHS AND SPACING SHALL BE WITHIN

HOLE LOCATION  $\pm 0.003"$  MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005".

HOLE LOCATION  $\pm$  0.003 . MAXIMUM LAYER TO LAYER MIS-REGISTRATION SHALL BE 0.005  
HOLE DIAMETER TOLERANCE IS  $\pm$  0.003" AFTER PLATING

HOLE DIAMETER TOLERANCE IS  $\pm 0.005$  AFTER PLATING  
REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAX

REMOVE ALL BURRS AND BREAK SHARP EDGES 0.015" MAXIMUM.  
SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF

SURFACE MOUNT PAD PLATING MUST BE FLAT TO A MAXIMUM OF 0.08 (0.005") ABOVE BOARD SURFACE.  
MINIMUM COPPER PLATING 0.001" THICK FOR PLATED THROUGH HOLES.

	ENGINEER: Chris Hettrick	PROJECT TITLE:  smari minna
	PCB DESIGNER: Chris Hettrick	
	DATE: 2012-08-21	COMPANY NAME: Populate All The Resistors
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: <a href="mailto:info@populatealltheresistors.com">info@populatealltheresistors.com</a>
		REVISION: v0.4

## 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 ->

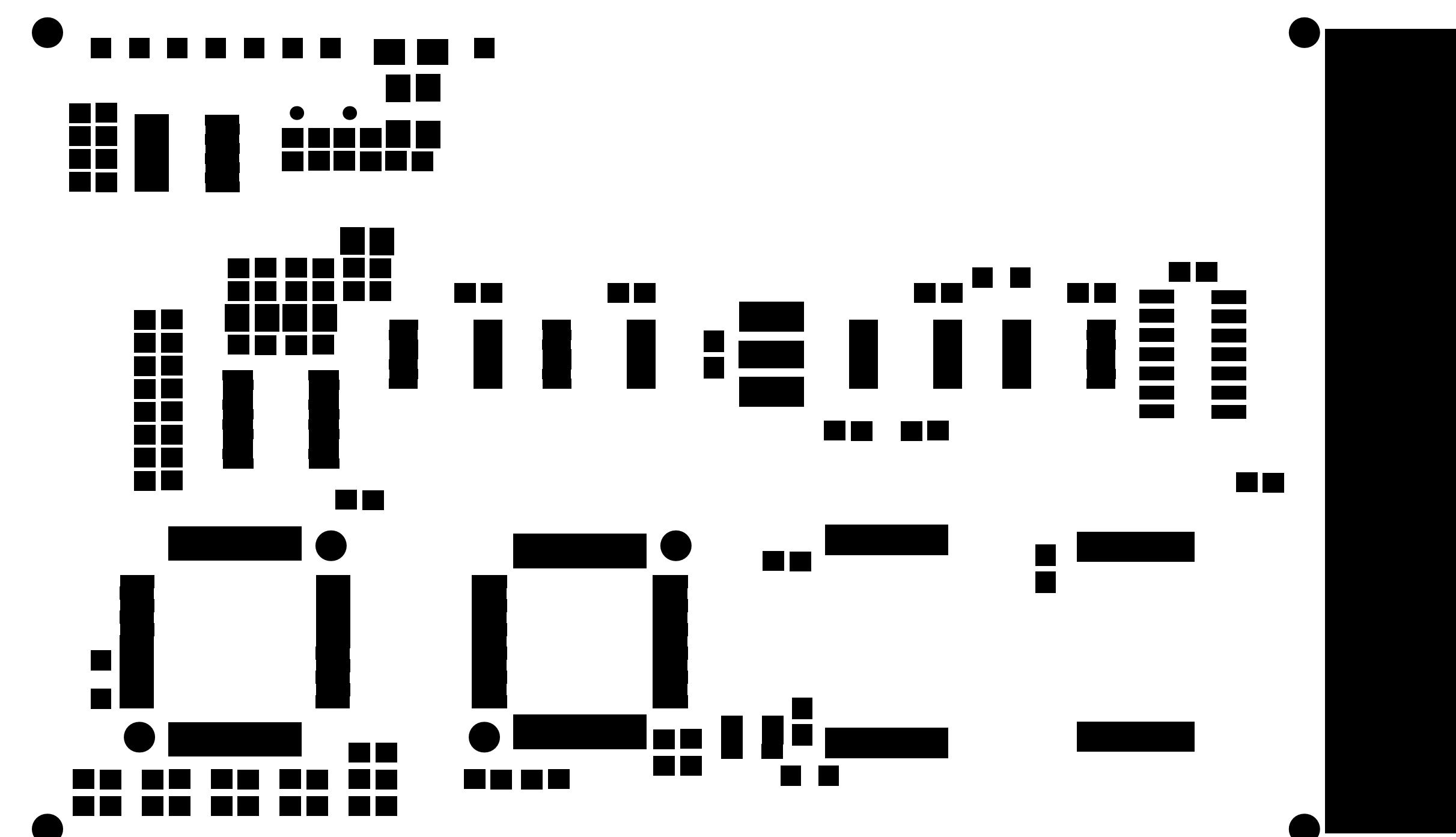
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,  
LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.  
MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.  
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.  
THIS IS A 4 LAYER BOARD.

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  
v0.4

	ENGINEER: Chris Hettrick	PROJECT TITLE:		
	PCB DESIGNER: Chris Hettrick			smari minna
	DATE: 2012-08-21		COMPANY NAME: Populate All The Resistors	
	FILE NAME: minna_pcb.PcbDoc		COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.4

## 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 ->

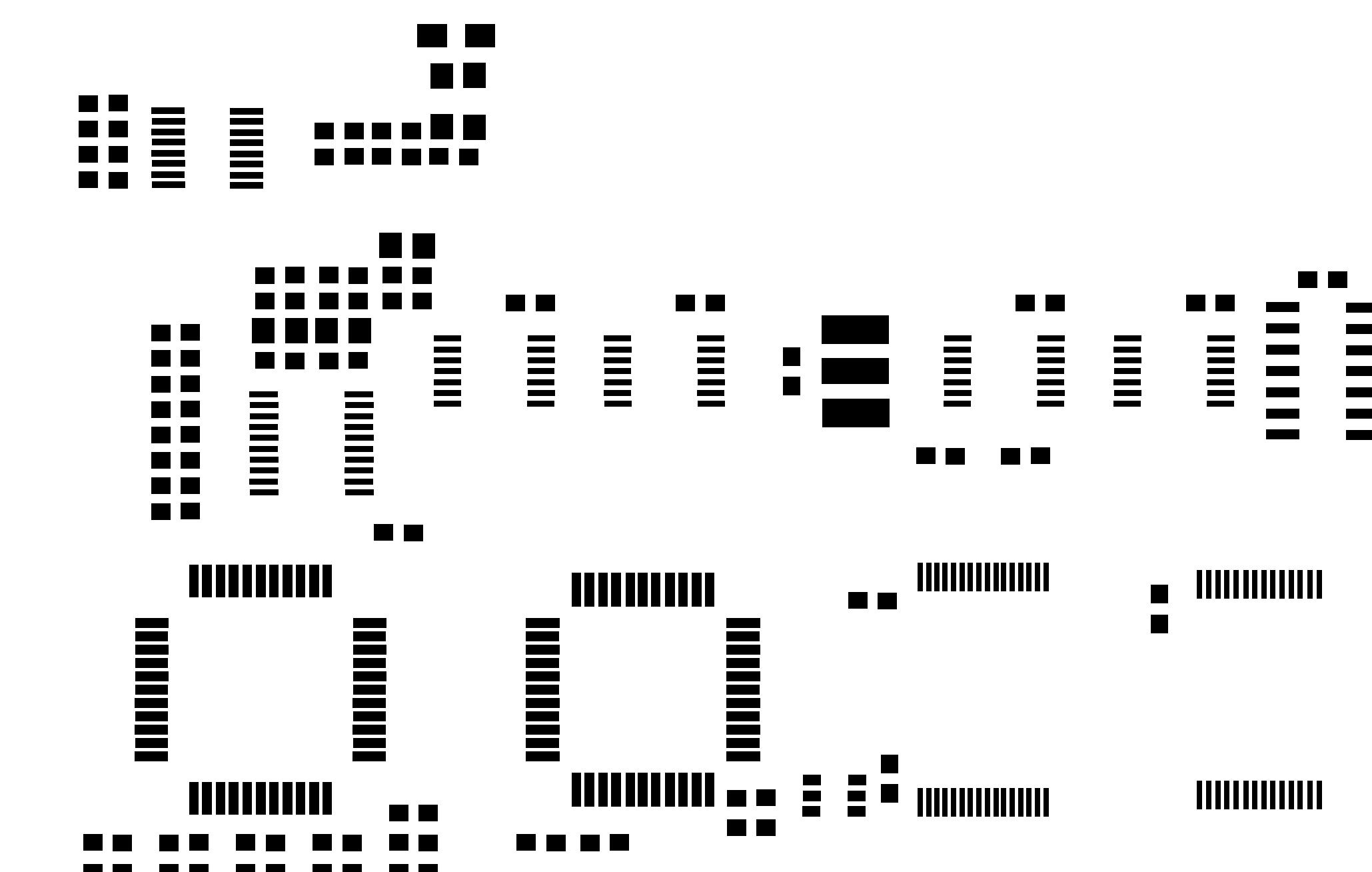
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,  
LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.  
MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.  
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.  
THIS IS A 4 LAYER BOARD.

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

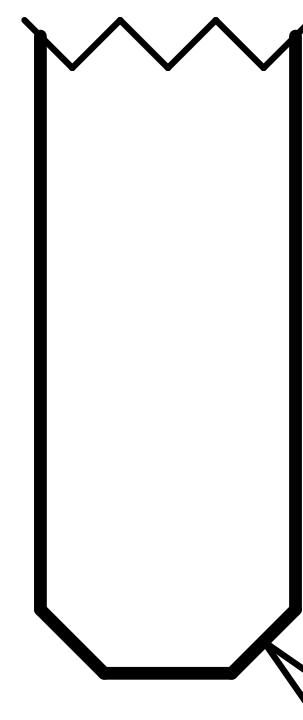
TOLERANCES ->

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.



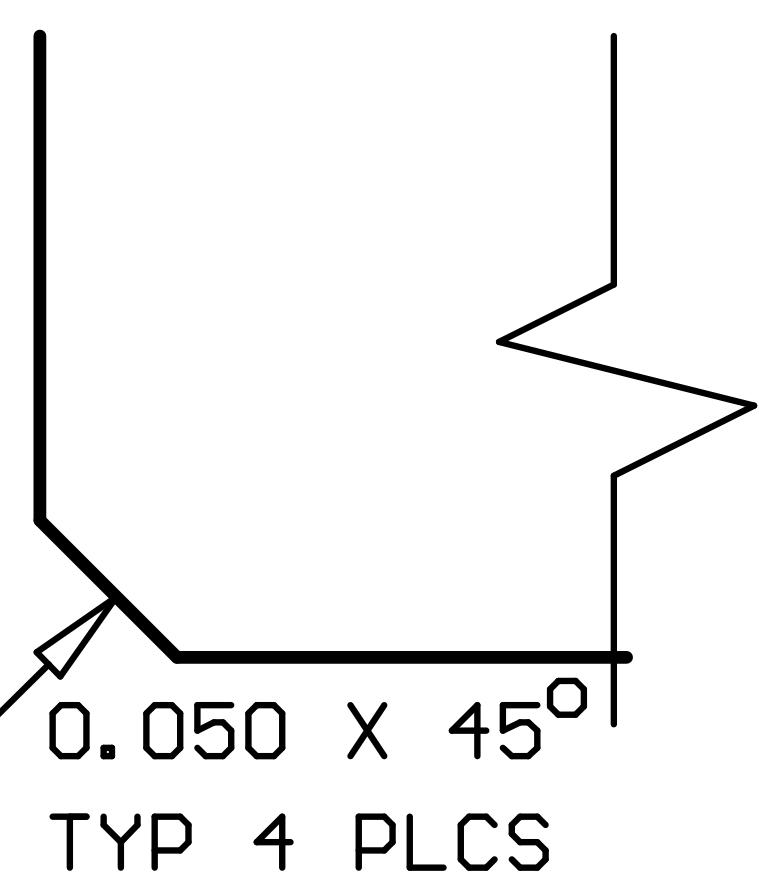
Bottom Paste  
smari minna  
v0.4

	ENGINEER: Chris Hettrick	PROJECT TITLE:		
	PCB DESIGNER: Chris Hettrick			smari minna
	DATE: 2012-08-21		COMPANY NAME: Populate All The Resistors	
	FILE NAME: minna_pcb.PcbDoc		COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.4



# DETAIL A

0.030 X 30° CHAMFER  
BOTH SIDES  
ENTIRE LENGTH OF  
EDGE CONNECTOR



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  
v0.4

ENGINEER: Chris Hettrick	PROJECT TITLE:  smari minna
PCB DESIGNER: Chris Hettrick	
DATE: 2012-08-21	COMPANY NAME: Populate All The Resistors
FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com
	REVISION: v0.4

## 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  
 v0.4

	ENGINEER: Chris Hettrick	PROJECT TITLE:  smari minna		
	PCB DESIGNER: Chris Hettrick			
	DATE: 2012-08-21	COMPANY NAME: Populate All The Resistors		
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.4	

## 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 ->

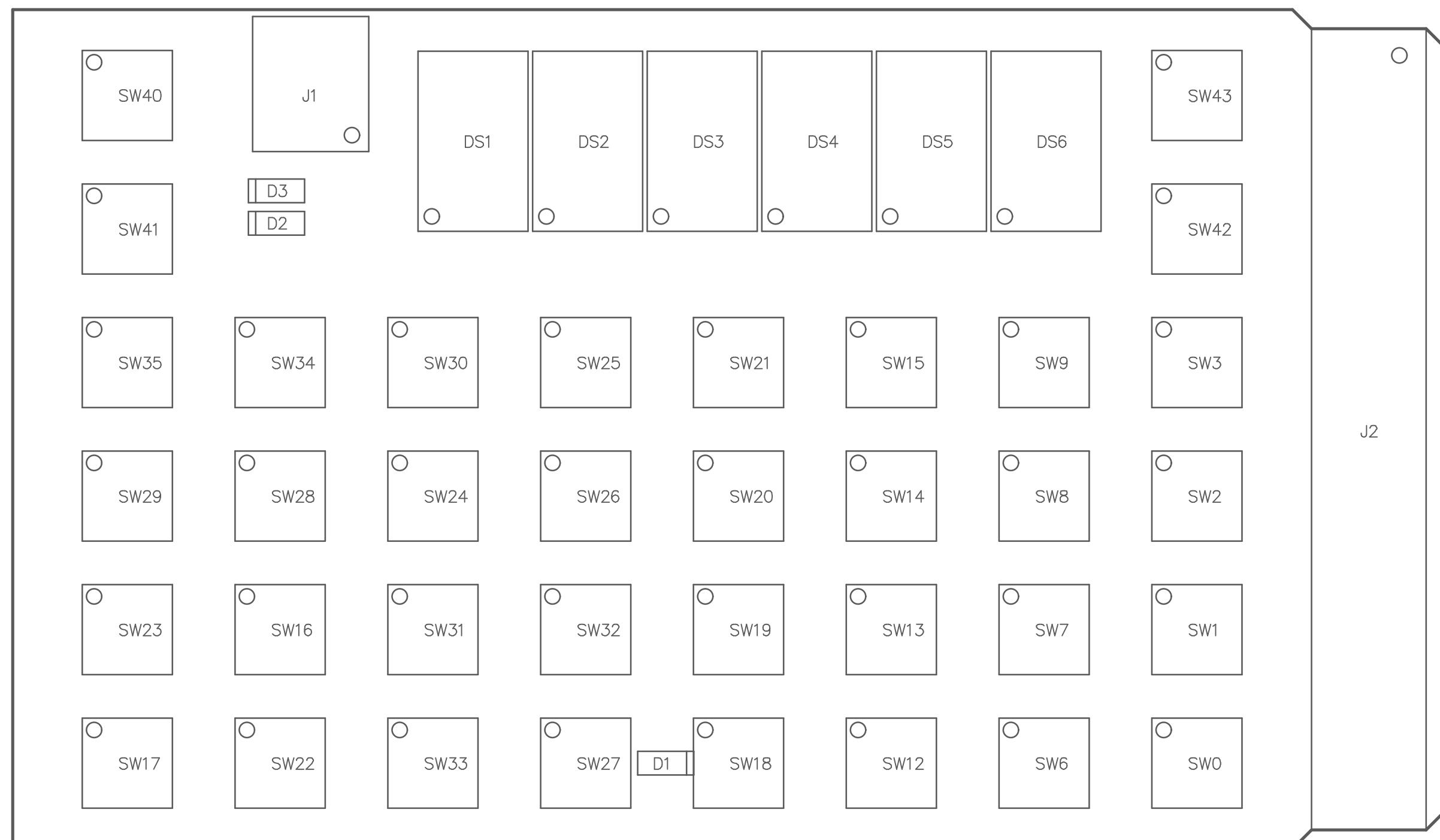
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,  
LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.  
MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.  
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.  
THIS IS A 4 LAYER BOARD.

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  
v0.4  
2012-08-21  
Not To Scale

	ENGINEER: Chris Hettrick	PROJECT TITLE:		
	PCB DESIGNER: Chris Hettrick	smari minna		
	DATE: 2012-08-21	COMPANY NAME: Populate All The Resistors		
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com	REVISION: v0.4	

## 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 ->

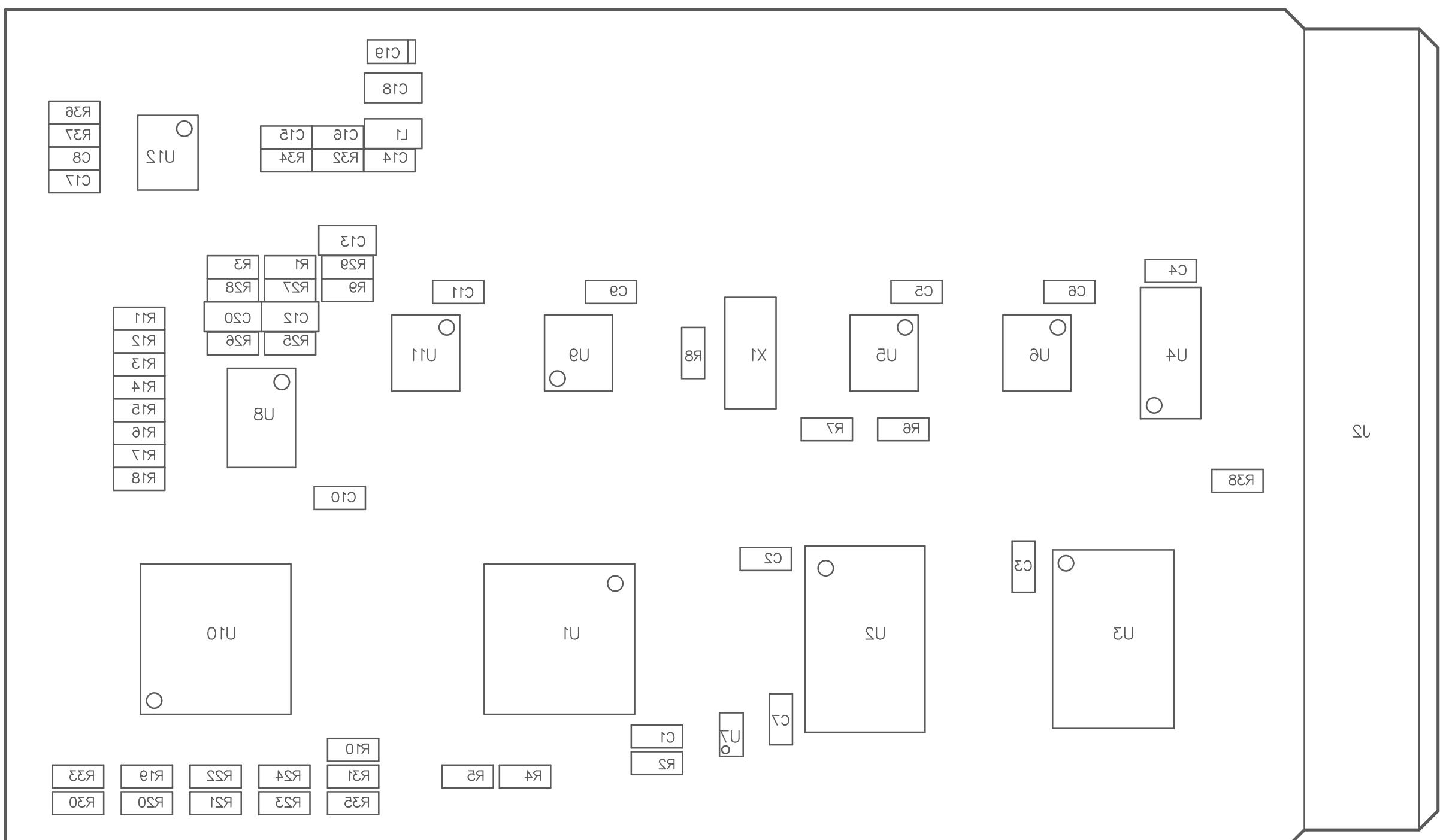
MATERIAL: EPOXY FIBERGLASS FR4 TG170 OR EQUIVALENT,  
LAMINATE AND PREPREG (B-STAGE) TO BE IN ACCORDANCE WITH IPC-4101/24 OR IPC-4101/26.  
MATERIAL MUST MEET UL 94V-0 FLAMABILITY RATING.  
ACCEPTABILITY REQUIREMENTS PER IPC-A-600E. 0.062" +/- 0.007" FINISHED THICKNESS.  
THIS IS A 4 LAYER BOARD.

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  
Not To Scale  
2012-08-21  
0.4

	ENGINEER: Chris Hettrick	PROJECT TITLE:  <b>smari minna</b>
	PCB DESIGNER: Chris Hettrick	
	DATE: 2012-08-21	COMPANY NAME: Populate All The Resistors
	FILE NAME: minna_pcb.PcbDoc	COMPANY EMAIL: info@populatealltheresistors.com

REVISION:  
v0.4

