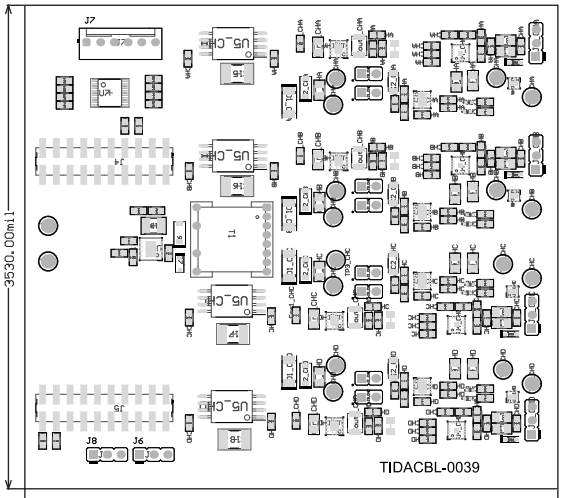
Layer	Name	Material	Thickness	Constant	Board Layer Stack
1	Top Overlay				
2	Top Solder	Solder Resist	0.40mil	3.5	
3	Top Layer	Copper	1.40mil		
4	Dielectric 1		8.00mil	4.2	
5	V-	Copper	1.40mil		
6	Dielectric 2	FR-4	40.00mil	4.2	
7	V+	Copper	1.40mil		
8	Dielectric 3		8.00mil	4.2	
9	Bottom Layer	Copper	1.40mil		
10	Bottom Solder	Solder Resist	0.40mil	3.5	
11	Bottom Overlay				



COMPONENTS MARKED 'DNP' SHOULD NOT BE POPULATED
ASSEMBLY VARIANT: 001

DESIGN INFORMATION

MIN. TRACK WIDTH: 8 MIL
MIN. CLEARANCE: 0.2 mm
MIN. VIA PAD SIZE: 24 MIL
MINIMUM ANNULAR RING 0.05mm (2MIL) EXTERNAL
PER IPC-D-275 CLASS 2 LEVEL C
REGISTRATION TOLERANCES: METAL +/- 5 MIL, HOLES +/- 3 MIL
HOLE SIZE TOLERANCE (UNLESS OTHERWISE SPECIFIED): +/- 3 MIL

MATERIAL:
☐ FR-408 ☒ FR-4 High Tg ☐ OTHER
THICKNESS: ☒ 62 MIL (1.6mm) +/-10% ☐ OTHER
TOLERANCE: ☒ ANSI IPC-6012 TYPE 3 CLASS 2
☐ OTHER +/-
BOW & TWIST: ☒ ANSI IPC-6012 TYPE 3 CLASS 2
☐ OTHER +/-

DRILLING:
REFERENCE: ☒ AS SHOWN ☒ NC_DRILL FILES
PTH COPPER THICKNESS: ☐ 20-30 um ☐ OTHER

BOARD FINISH:
SILKSCREEN: ☐ TOP ☐ BOTTOM
SILKSCREEN COLOR: ☐ WHITE ☐ OTHER
SOLDER RESIST COLOR: ☐ GREEN ☐ OTHER
☐ MATTE ☐ SEMI-GLOSS

SURFACE FINISH: ☐ IMMERSION GOLD (ENIG) ☐ ENEPIG
☐ IMM. TIN/SILVER OR EQUIV ☐ OTHER

ARRAY/PANEL: ☐ CUT AND TRIM PER M1 BOARD OUTLINE
☐ N.C. ROUTE ☐ V. SCORE

CERTIFICATION: MATERIALS AND WORKMANSHIP FOR ALL PCBS TO MEET OR EXCEED THE REQUIREMENTS OF:
☐ ANSI IPC-A-600F CLASS -> ☐ 1 ☐ 2 ☐ 3
☐ RoHS ☐ OTHER PER ORDER

ALL BOARDS MUST MEET OR EXCEED UL94-V0 REQUIREMENTS.
PCB MUST BEAR THE UL94V-0 UL REGISTERED MATERIAL ID NUMBER

ADDITIONAL REQUIREMENTS:
MICROSECTION: ☐ YES
BARE BOARD ELEC. TEST: ☐ NONE ☐ REQUIRED ☐ PER ORDER
☐ XX MIL VIAS REQUIRE NON-CONDUCTIVE FILL AND PLANARIZE
☐ XX MIL VIAS REQUIRE CONDUCTIVE FILL AND PLANARIZE
☐ OUTER XX MIL TRACES REQUIRE 50 OHM SINGLE-ENDED IMPEDANCE
☐ LAYER 2 & 3 (INNER LAYERS) XX MIL WIDE, XX MIL SPACE TRACES REQUIRE 100 OHM DIFFERENTIAL IMPEDANCE

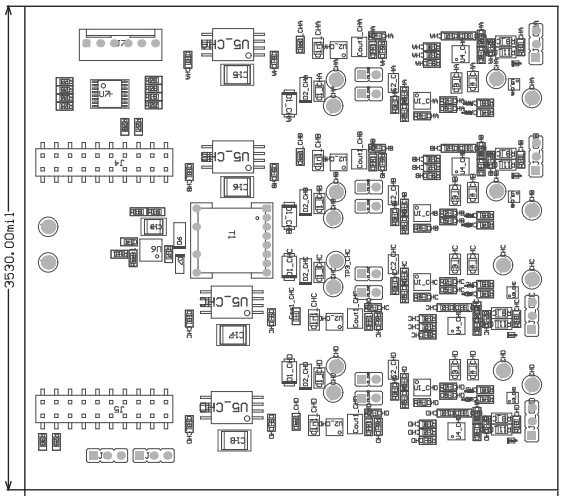
TEXAS INSTRUMENTS

PROJECT TITLE: Change in menu Project|Project Options|Parameters
DESIGNED FOR: Public Release
FILE NAME: TIDA-010048_PCB.PcbDoc

TID #: N/A	TIDA-010048
Assembly Drawing:	Top Layer

ENGINEER:	LAYOUT BY:
Ben Su	Who did the Layout?
SCALE: 0.71	ALTIUM DESIGNER VERSION: 17.1.5.472

Layer	Name	Material	Thickness	Constant	Board Layer Stack
1	Top Overlay				
2	Top Solder	Solder Resist	0.40mil	3.5	
3	Top Layer	Copper	1.40mil		
4	Dielectric 1		8.00mil	4.2	
5	V-	Copper	1.40mil		
6	Dielectric 2	FR-4	40.00mil	4.2	
7	V+	Copper	1.40mil		
8	Dielectric 3		8.00mil	4.2	
9	Bottom Layer	Copper	1.40mil		
10	Bottom Solder	Solder Resist	0.40mil	3.5	
11	Bottom Overlay				



COMPONENTS MARKED 'DNP' SHOULD NOT BE POPULATED
ASSEMBLY VARIANT: 001

DESIGN INFORMATION

MIN. TRACK WIDTH: 8 MIL
MIN. CLEARANCE: 0.2 mm
MIN. VIA PAD SIZE: 24 MIL
MINIMUM ANNULAR RING 0.05mm (2MIL) EXTERNAL
PER IPC-D-275 CLASS 2 LEVEL C
REGISTRATION TOLERANCES: METAL +/- 5 MIL, HOLES +/- 3 MIL
HOLE SIZE TOLERANCE (UNLESS OTHERWISE SPECIFIED): +/- 3 MIL

MATERIAL:
[ ] FR-408 [X] FR-4 High Tg [ ] OTHER
THICKNESS: [X] 62 MIL (1.6mm) +/-10% [ ] OTHER
TOLERANCE: [X] ANSI IPC-6012 TYPE 3 CLASS 2
[ ] OTHER +/-
BOW & TWIST: [X] ANSI IPC-6012 TYPE 3 CLASS 2
[ ] OTHER +/-

DRILLING:
REFERENCE: [X] AS SHOWN [X] NC_DRILL FILES
PTH COPPER THICKNESS: [ ] 20-30 um [ ] OTHER

BOARD FINISH:
SILKSCREEN: [ ] TOP [ ] BOTTOM
SILKSCREEN COLOR: [ ] WHITE [ ] OTHER
SOLDER RESIST COLOR: [ ] GREEN [ ] OTHER
[ ] MATTE [ ] SEMI-GLOSS

SURFACE FINISH: [ ] IMMERSION GOLD (ENIG) [ ] ENEPIG
[ ] IMM. TIN/SILVER OR EQUIV [ ] OTHER

ARRAY/PANEL: [ ] CUT AND TRIM PER M1 BOARD OUTLINE
[ ] N.C. ROUTE [ ] V. SCORE

CERTIFICATION: MATERIALS AND WORKMANSHIP FOR ALL PCBS TO MEET OR EXCEED THE REQUIREMENTS OF:
[ ] ANSI IPC-A-600F CLASS -> [ ] 1 [ ] 2 [ ] 3
[ ] RoHS [ ] OTHER PER ORDER
ALL BOARDS MUST MEET OR EXCEED UL94-V0 REQUIREMENTS.
PCB MUST BEAR THE UL94V-0 UL REGISTERED MATERIAL ID NUMBER

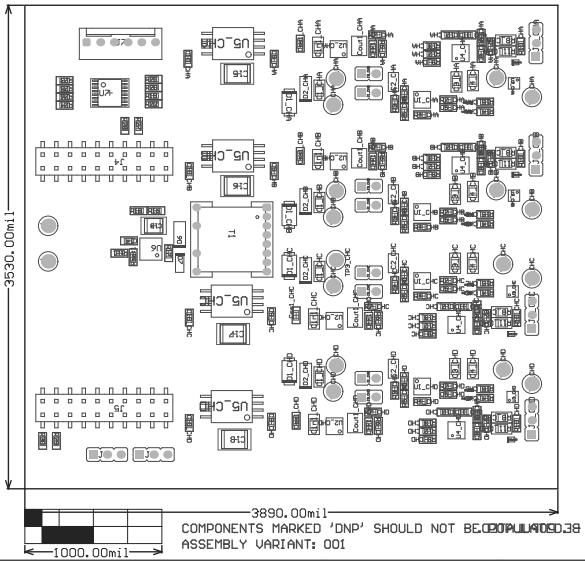
ADDITIONAL REQUIREMENTS:
MICROSECTION: [ ] YES
BARE BOARD ELEC. TEST: [ ] NONE [ ] REQUIRED [ ] PER ORDER
[ ] XX MIL VIAS REQUIRE NON-CONDUCTIVE FILL AND PLANARIZE
[ ] XX MIL VIAS REQUIRE CONDUCTIVE FILL AND PLANARIZE
[ ] OUTER XX MIL TRACES REQUIRE 50 OHM SINGLE-ENDED IMPEDANCE
[ ] LAYER 2 & 3 (INNER LAYERS) XX MIL WIDE, XX MIL SPACE TRACES REQUIRE 100 OHM DIFFERENTIAL IMPEDANCE

TEXAS INSTRUMENTS

PROJECT TITLE: Change in menu Project|Project Options|Parameters
DESIGNED FOR: Public Release
FILE NAME: TIDA-010048_PCB.PcbDoc

ENGINEER: Ben Su	LAYOUT BY: Who did the Layout?
SCALE: 0.71	ALTIUM DESIGNER VERSION: 17.1.5.472

Layer	Name	Material	Thickness	Constant	Board Layer Stack
1	Top Overlay				
2	Top Solder	Solder Resist	0.40mil	3.5	
3	Top Layer	Copper	1.40mil		
4	Dielectric 1		8.00mil	4.2	
5	V-	Copper	1.40mil		
6	Dielectric 2	FR-4	40.00mil	4.2	
7	V+	Copper	1.40mil		
8	Dielectric 3		8.00mil	4.2	
9	Bottom Layer	Copper	1.40mil		
10	Bottom Solder	Solder Resist	0.40mil	3.5	
11	Bottom Overlay				



COMPONENTS MARKED 'DNP' SHOULD NOT BE POPULATED
ASSEMBLY VARIANT: 001

DESIGN INFORMATION

MIN. TRACK WIDTH: 8 MIL
MIN. CLEARANCE: 0.2 mm
MIN. VIA PAD SIZE: 24 MIL
MINIMUM ANNULAR RING 0.05mm (2MIL) EXTERNAL
PER IPC-D-275 CLASS 2 LEVEL C
REGISTRATION TOLERANCES: METAL +/- 5 MIL, HOLES +/- 3 MIL
HOLE SIZE TOLERANCE (UNLESS OTHERWISE SPECIFIED): +/- 3 MIL

MATERIAL:
[ ] FR-408 [X] FR-4 High Tg [ ] OTHER
THICKNESS: [X] 62 MIL (1.6mm) +/-10% [ ] OTHER
TOLERANCE: [X] ANSI IPC-6012 TYPE 3 CLASS 2
[ ] OTHER +/-
BOW & TWIST: [X] ANSI IPC-6012 TYPE 3 CLASS 2
[ ] OTHER +/-

DRILLING:
REFERENCE: [X] AS SHOWN [X] NC_DRILL FILES
PTH COPPER THICKNESS: [ ] 20-30 um [ ] OTHER
BOARD FINISH:
SILKSCREEN: [ ] TOP [ ] BOTTOM
SILKSCREEN COLOR: [ ] WHITE [ ] OTHER
SOLDER RESIST COLOR: [ ] GREEN [ ] OTHER
[ ] MATTE [ ] SEMI-GLOSS
SURFACE FINISH: [ ] IMMERSION GOLD (ENIG) [ ] ENEPIG
[ ] IMM. TIN/SILVER OR EQUIV [ ] OTHER
ARRAY/PANEL: [ ] CUT AND TRIM PER M1 BOARD OUTLINE
[ ] N.C. ROUTE [ ] V. SCORE
CERTIFICATION: MATERIALS AND WORKMANSHIP FOR ALL PCBS TO MEET OR EXCEED THE REQUIREMENTS OF:
[ ] ANSI IPC-A-600F CLASS -> [ ] 1 [ ] 2 [ ] 3
[ ] RoHS [ ] OTHER PER ORDER
ALL BOARDS MUST MEET OR EXCEED UL94-V0 REQUIREMENTS.
PCB MUST BEAR THE UL94V-0 UL REGISTERED MATERIAL ID NUMBER

ADDITIONAL REQUIREMENTS:
MICROSECTION: [ ] YES
BARE BOARD ELEC. TEST: [ ] NONE [ ] REQUIRED [ ] PER ORDER
[ ] XX MIL VIAS REQUIRE NON-CONDUCTIVE FILL AND PLANARIZE
[ ] XX MIL VIAS REQUIRE CONDUCTIVE FILL AND PLANARIZE
[ ] OUTER XX MIL TRACES REQUIRE 50 OHM SINGLE-ENDED IMPEDANCE
[ ] LAYER 2 & 3 (INNER LAYERS) XX MIL WIDE, XX MIL SPACE TRACES REQUIRE 100 OHM DIFFERENTIAL IMPEDANCE

TEXAS INSTRUMENTS

PROJECT TITLE: Change in menu Project|Project Options|Parameters
DESIGNED FOR: Public Release
FILE NAME: TIDA-010048_PCB.PcbDoc

ENGINEER: Ben Su	LAYOUT BY: Who did the Layout?
SCALE: 0.71	ALTIUM DESIGNER VERSION: 17.1.5.472

Layer	Name	Material	Thickness	Constant	Board Layer Stack
1	Top Overlay				
2	Top Solder	Solder Resist	0.40mil	3.5	
3	Top Layer	Copper	1.40mil		
4	Dielectric 1		8.00mil	4.2	
5	V-	Copper	1.40mil		
6	Dielectric 2	FR-4	40.00mil	4.2	
7	V+	Copper	1.40mil		
8	Dielectric 3		8.00mil	4.2	
9	Bottom Layer	Copper	1.40mil		
10	Bottom Solder	Solder Resist	0.40mil	3.5	
11	Bottom Overlay				

DESIGN INFORMATION

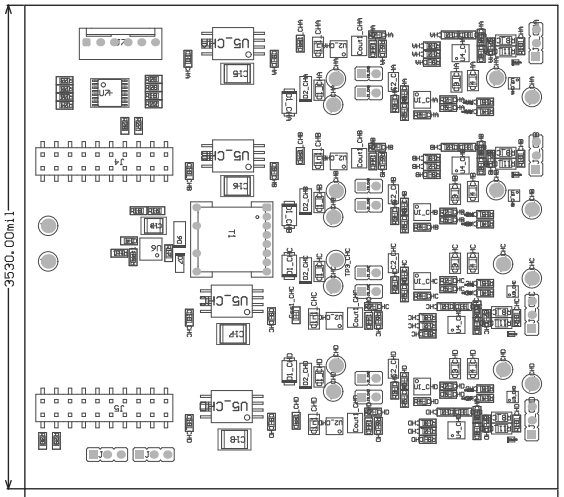
MIN. TRACK WIDTH: 8 MIL
MIN. CLEARANCE: 0.2 mm
MIN. VIA PAD SIZE: 24 MIL
MINIMUM ANNULAR RING 0.05mm (2MIL) EXTERNAL
PER IPC-D-275 CLASS 2 LEVEL C
REGISTRATION TOLERANCES: METAL +/- 5 MIL, HOLES +/- 3 MIL
HOLE SIZE TOLERANCE (UNLESS OTHERWISE SPECIFIED): +/- 3 MIL

MATERIAL:
☐ FR-408 ☒ FR-4 High Tg ☐ OTHER _____
THICKNESS: ☒ 62 MIL (1.6mm) +/-10% ☐ OTHER _____
TOLERANCE: ☒ ANSI IPC-6012 TYPE 3 CLASS 2
☐ OTHER +/- _____
BOW & TWIST: ☒ ANSI IPC-6012 TYPE 3 CLASS 2
☐ OTHER +/- _____

DRILLING:
REFERENCE: ☒ AS SHOWN ☒ NC_DRILL FILES
PTH COPPER THICKNESS: ☐ 20-30 um ☐ OTHER _____

BOARD FINISH:
SILKSCREEN: ☐ TOP ☐ BOTTOM
SILKSCREEN COLOR: ☐ WHITE ☐ OTHER _____
SOLDER RESIST COLOR: ☐ GREEN ☐ OTHER _____
☐ MATTE ☐ SEMI-GLOSS
SURFACE FINISH: ☐ IMMERSION GOLD (ENIG) ☐ ENEPIG
☐ IMM. TIN/SILVER OR EQUIV ☐ OTHER _____
ARRAY/PANEL: ☐ CUT AND TRIM PER M1 BOARD OUTLINE
☐ N.C. ROUTE ☐ V. SCORE
CERTIFICATION: MATERIALS AND WORKMANSHIP FOR ALL PCBS TO MEET OR EXCEED THE REQUIREMENTS OF:
☐ ANSI IPC-A-600F CLASS -> ☐ 1 ☐ 2 ☐ 3
☐ RoHS ☐ OTHER PER ORDER
ALL BOARDS MUST MEET OR EXCEED UL94-V0 REQUIREMENTS.
PCB MUST BEAR THE UL94V-0 UL REGISTERED MATERIAL ID NUMBER

ADDITIONAL REQUIREMENTS:
MICROSECTION: ☐ YES
BARE BOARD ELEC. TEST: ☐ NONE ☐ REQUIRED ☐ PER ORDER
☐ XX MIL VIAS REQUIRE NON-CONDUCTIVE FILL AND PLANARIZE
☐ XX MIL VIAS REQUIRE CONDUCTIVE FILL AND PLANARIZE
☐ OUTER XX MIL TRACES REQUIRE 50 OHM SINGLE-ENDED IMPEDANCE
☐ LAYER 2 & 3 (INNER LAYERS) XX MIL WIDE, XX MIL SPACE TRACES REQUIRE 100 OHM DIFFERENTIAL IMPEDANCE



COMPONENTS MARKED 'DNP' SHOULD NOT BE POPULATED
ASSEMBLY VARIANT: 001

TEXAS INSTRUMENTS

PROJECT TITLE: Change in menu Project|Project Options|Parameters

DESIGNED FOR: Public Release

FILE NAME: TIDA-010048_PCB.PcbDoc

ENGINEER: Ben Su

LAYOUT BY: Who did the Layout?

SCALE: 0.71

ALTIUM DESIGNER VERSION: 17.1.5.472

IMPORTANT NOTICE AND DISCLAIMER

TI PROVIDES TECHNICAL AND RELIABILITY DATA (INCLUDING DATASHEETS), DESIGN RESOURCES (INCLUDING REFERENCE DESIGNS), APPLICATION OR OTHER DESIGN ADVICE, WEB TOOLS, SAFETY INFORMATION, AND OTHER RESOURCES “AS IS” AND WITH ALL FAULTS, AND DISCLAIMS ALL WARRANTIES, EXPRESS AND IMPLIED, INCLUDING WITHOUT LIMITATION ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE OR NON-INFRINGEMENT OF THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

These resources are intended for skilled developers designing with TI products. You are solely responsible for (1) selecting the appropriate TI products for your application, (2) designing, validating and testing your application, and (3) ensuring your application meets applicable standards, and any other safety, security, or other requirements. These resources are subject to change without notice. TI grants you permission to use these resources only for development of an application that uses the TI products described in the resource. Other reproduction and display of these resources is prohibited. No license is granted to any other TI intellectual property right or to any third party intellectual property right. TI disclaims responsibility for, and you will fully indemnify TI and its representatives against, any claims, damages, costs, losses, and liabilities arising out of your use of these resources.

TI’s products are provided subject to TI’s Terms of Sale (www.ti.com/legal/termsofsale.html) or other applicable terms available either on ti.com or provided in conjunction with such TI products. TI’s provision of these resources does not expand or otherwise alter TI’s applicable warranties or warranty disclaimers for TI products.

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265