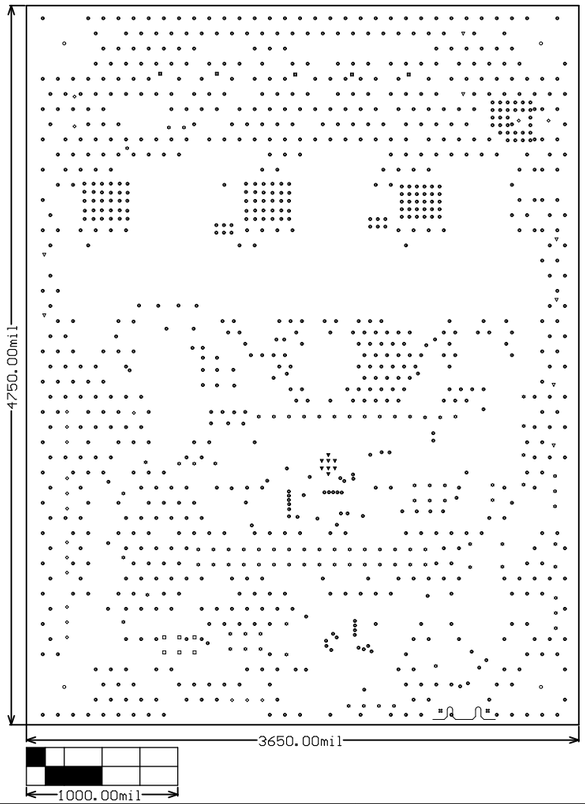
Layer	Name	Material	Thickness	Constant	Board Layer Stack
1	Top Overlay				
2	Top Solder	Solder Resist	0.40mil	3.5	
3	Top Layer	Copper	2.80mil		
4	Dielectric 2	FR-4	10.00mil	4.8	
5	GND Layer	Copper	1.40mil		
6	Dielectric 1	FR-4	30.00mil	4.2	
7	Power Layer	Copper	1.40mil		
8	Dielectric 3	FR-4	10.00mil	4.8	
9	Bottom Layer	Copper	2.80mil		
10	Solder Side	Solder Resist	0.40mil	3.5	
11	Bottom Overlay				



Symbol	Quantity	Finished Hole Size	Plated	Hole Type	Drill Layer Pair	Hole Tolerance
▽	8	7.87mil (0.200mm)	PTH	Round	Top Layer - Bottom Layer	
⊙	964	10.00mil (0.254mm)	PTH	Round	Top Layer - Bottom Layer	
□	6	23.62mil (0.600mm)	PTH	Round	Top Layer - Bottom Layer	
◇	20	33.47mil (0.850mm)	PTH	Round	Top Layer - Bottom Layer	
✡	94	40.00mil (1.016mm)	PTH	Round	Top Layer - Bottom Layer	
◘	5	51.18mil (1.300mm)	PTH	Round	Top Layer - Bottom Layer	+/-3.94mil
▽	8	51.18mil (1.300mm)	PTH	Round	Top Layer - Bottom Layer	
○	4	125.98mil (3.200mm)	PTH	Round	Top Layer - Bottom Layer	
⊠	2	23.62mil (0.600mm)	PTH	Slot	Top Layer - Bottom Layer	
	1111 Total					

Slot definitions : Routed Path Length = Calculated from tool start centre position to tool end centre position.
Hole Length = Routed Path Length + Tool Size = Slot length as defined in the PCB layout

DESIGN INFORMATION

MIN. TRACK WIDTH: 8 MIL
MIN. CLEARANCE: 0.2 mm
MIN. VIA PAD SIZE: 30 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: MCx8329AEVM
DESIGNED FOR: Public Release
FILE NAME: MD067.PcbDoc

ALL ARTWORK VIEWED FROM TOP SIDE	BOARD #: MD067	REV: A	SVN REV: 801e70aabbbc8cd27568687bf7b9a15777b2a15 (Locally Modified)
LAYER NAME = [illegible]ensions	TID #: N/A		
PLOT NAME = Fabrication Drawing	GENERATED : 9/20/2022 3:54:59 PM	TEXAS INSTRUMENTS	

Texas Instruments (TI) and/or its licensors do not warrant the accuracy or completeness of this specification or any information contained therein. TI and/or its licensors do not warrant that this design will meet the specifications, will be suitable for your application or fit for any particular purpose, or will operate in an implementation. TI and/or its licensors do not warrant that the design is production worthy. You should completely validate and test your design implementation to confirm the system functionality for your application.

ENGINEER:	LAYOUT BY:
Vishnu Balaraj	Vishnu Balaraj
SCALE: 1.00	ALTIUM DESIGNER VERSION: 18.1.9.240