



FOR REFERENCE ONLY

Layer	Name	Material	Thickness	Constant	Board Layer Stack	Board Layer Stack
1	Top Overlay					
2	Top Solder	Solder Resist	0.40mil	3.5		
3	Top Layer	Copper	2.00mil			
4	Dielectric1	FR-4	5.00mil	4.2		
5	Mid-Layer 1	Copper	2.80mil			
6	Dielectric2	FR-4	12.00mil	4.2		
7	Mid-Layer 2	Copper	2.80mil			
8	Dielectric3	FR-4	12.00mil	4.2		
9	Mid-Layer 3	Copper	2.80mil			
10	Dielectric4	FR-4	12.00mil	4.2		
11	Mid-Layer 4	Copper	2.80mil			
12	Dielectric5	FR-4	5.00mil	4.2		
13	Bottom Layer	Copper	2.00mil			
14	Bottom Solder	Solder Resist	0.40mil	3.5		
15	Bottom Overlay					

Symbol	Quantity	Finished Hole Size	Plated	Hole Type	Drill Layer Pair	Hole Tolerance
○	166	7.800mil (0.198mm)	PTH	Round	Top Layer - Bottom Layer	+3.000mil/-7.874mil
A	24	7.874mil (0.200mm)	PTH	Round	Top Layer - Bottom Layer	
B	6	29.528mil (0.750mm)	PTH	Round	Top Layer - Bottom Layer	
C	61	40.000mil (1.016mm)	PTH	Round	Top Layer - Bottom Layer	
D	12	49.213mil (1.250mm)	PTH	Round	Top Layer - Bottom Layer	
E	15	63.000mil (1.600mm)	PTH	Round	Top Layer - Bottom Layer	
F	4	126.000mil (3.200mm)	NPTH	Round	Top Layer - Bottom Layer	
	288 Total					

DESIGN INFORMATION

MIN. TRACK WIDTH: 10 MIL
 MIN. CLEARANCE: 6 MIL
 MIN. VIA PAD SIZE: 22 MIL
 MINIMUM ANNULAR RING 0.05mm (2ML) EXTERNAL
 PCB TO BE FABRICATED TO IPC 6012 class 2
 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
 7.874 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



PROJECT TITLE:
TPSM5D1806EUM

DESIGNED FOR:
Public Release

FILE NAME:
BSR105A.PcbDoc

ENGINEER:
Jason Arrigo

LAYOUT BY:
Jim McTeague

SCALE: 1.00

ALTIM DESIGNER VERSION:
17.1.9.592

ALL ARTWORK VIEWED FROM TOP SIDE

LAYER NAME = ~~00055-1806EUM~~

PLOT NAME = Fabrication Drawing

BOARD #: BSR105

TID #: N/A

REV: A

SUN REV: Not In VersionControl

GENERATED : 8/17/2020 5:46:17 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.