



Board Characteristics

1. 6-Layer Board
2. Minimum trace width: and clearance: 0.007"
3. 1 oz copper for all power layers and for Signal_1,2 (Top and Bottom)
4. Apply Solder Mask over bare copper.
5. Board Thickness: 0.062"
6. Silkscreen on Component Side.
7. FHS tolerances: +/- 0.003
8. Do Not Remove Any Thermal Ties from Artwork
9. 45 degrees chamfer
10. Interlayer spacing as specified
11. Zc=55 Ohm, Zd=1000hm for 0.007" traces on Top.
12. Immersion Gold Plating

BOARD'S DRILL SCHEDULE

DRILL SYMBOL	DRILL SIZE	COUNT	PLATED	Tolerance	COMMENT
○	.02	53	YES	---	
⊞	.035	48	YES	---	
⊕	.041	129	YES	---	
⊞	.057	4	YES	---	
⊖	.062992126	9	YES	---	
⊞	.106	2	NO	---	
⊕	.12795276	6	YES	---	
□	.15	2	NO	---	

UNLESS OTHERWISE SPECIFIED DIMENSIONS ARE IN INCHES TOLERANCES ARE: FRACTIONS DECIMALS ANGLES :XX :XXX DO NOT SCALE DRAWING		CONTRACT NO.		UNIVERSITY OF CHICAGO ELECTRONICS DEVELOPMENT GROUP		
TREATMENT		APPROVALS	DATE	TITLE PMT Distribution Module Specification Drawing		
FINISH		DRAWN M. Bogdan	8/25/2015	SIZE B FSCM NO. DWG. NO. 2854 REV. A		
SIMILAR TO		CHECKED M. Bogdan	8/25/2015	SCALE 1/2 SHEET		
ACT. WT	CALC WT	ISSUED				