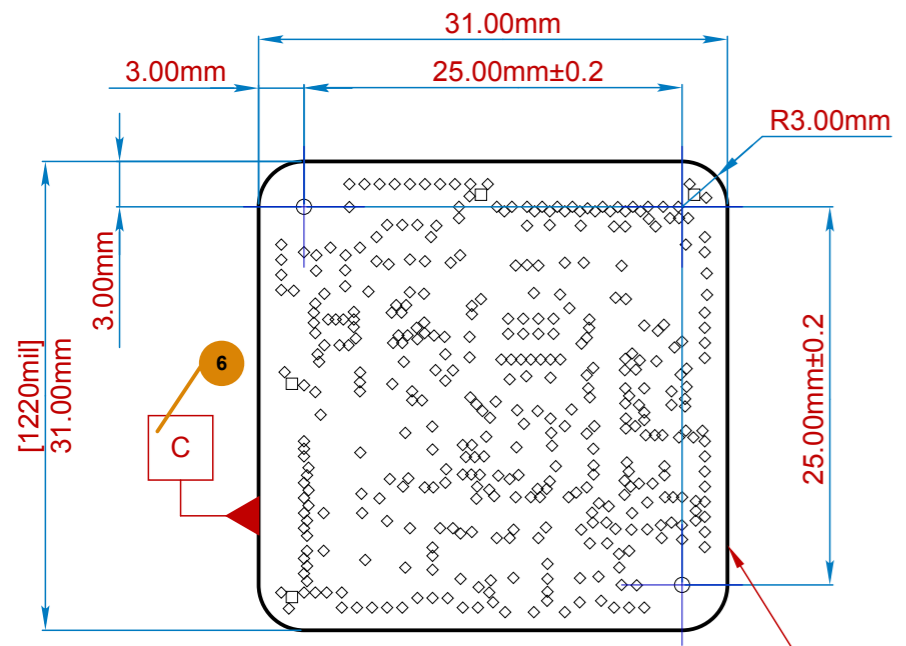


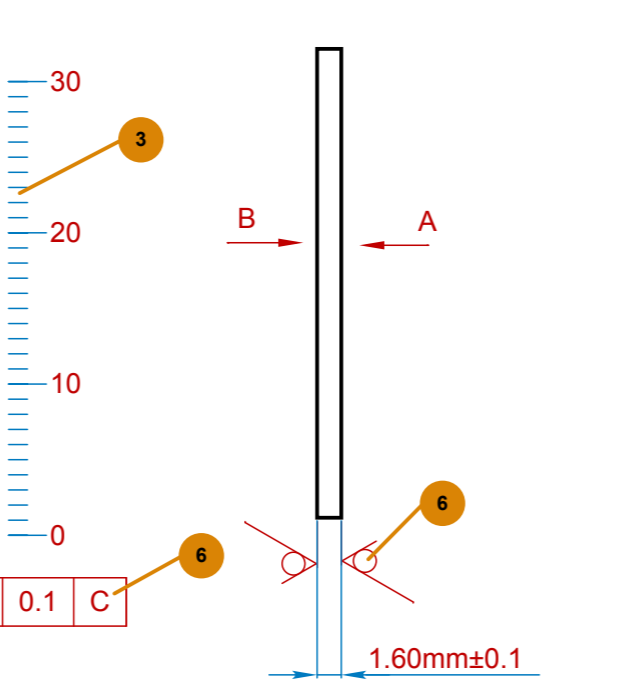
The Main Advantages when using Draftsman to Create a Fabrication Drawing

REV STATUS OF SHEETS		REVISIONS									
REV SHEET		ZONE	REV	DESCRIPTION	DATE	APPROVED					

Drill Drawing View (Scale 2:1)



View from Front side (Scale 2:1)

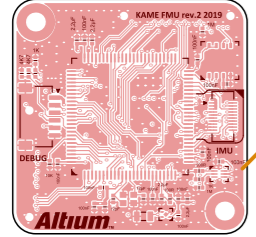


Ra 6.3

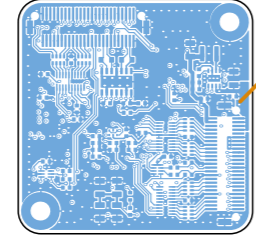
NOTES UNLESS OTHERWISE SPECIFIED:

- PCB VENDOR MUST NOTIFY RESPONSIBLE PCB CONTACT OF ANY DISCREPANCIES FOUND BETWEEN FABRICATION DATA AND FABRICATION DRAWING NOTES.
- THIS DRAWING IS VIEWED FROM THE PRIMARY OR TOP SIDE OF THE PCB.
- FABRICATION OF THIS PCB SHALL BE IN CONFORMANCE WITH THE FOLLOWING SPECIFICATIONS: IPC-6012 CLASS 2 (LATEST REVISION).
- FABRICATION OF THIS PCB TO BE ACCEPTABLE TO IPC-A-600 CLASS 2 (LATEST REVISION).
- ALL MATERIALS USED MUST BE RoHS COMPLIANT.
- ALL DIMENSIONAL LIMITS APPLY AFTER PLATING OR PROCESSING.
- TOLERANCES OF DATUM HOLE TO: BOARD EDGE LOCATIONS +/- 0.1. DRILLED HOLE LOCATIONS +/- 0.1. V-SCORE LOCATIONS +/- 0.1
- MATERIALS TO BE FR-4 .
- FLAME CLASS: UL 94V-0 & MUST MEET REQUIREMENTS OF UL796.
- FINISHED CONDUCTOR WIDTH NOT TO BE REDUCED MORE THAN 10% OF MINIMUM WIDTH FROM ARTWORK SUPPLIED. FINISHED CONDUCTOR SPACING NOT TO BE REDUCED MORE THAN 10% OF MINIMUM SPACING FROM ARTWORK SUPPLIED.
- SOLDER MASK IS LIQUID PHOTO IMAGEABLE AND IN ACCORDANCE WITH IPC-SM-840C CLASS T. FINISH MUST BE BLUE AND GLOSSY
- PCB FINISH BOTH SIDES TO BE ELECTROLESS NICKEL (Ni) FOLLOWED BY IMMERSION GOLD (Au) (ENIG) ACCEPTABLE TO IPC-4552.

A (Scale 1:1) Top Layer and Top Overlay



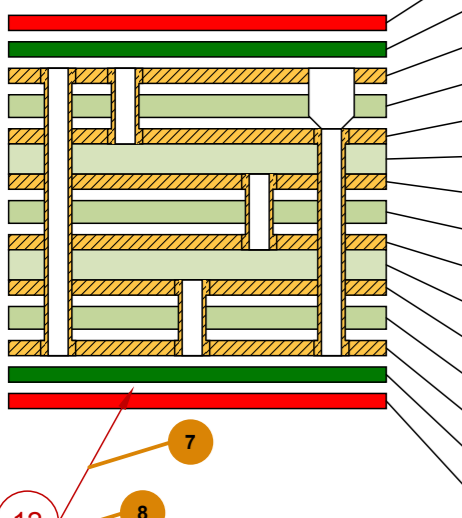
B (Scale 1:1) Bottom Layer



Transmission Line Structure Table

Impedance Id	Transmission Line	Target Impedance	Calculated Impedance	Trace layer	Wide Trace Width	Narrow Trace Width	Reference layers
1	Coated Microstrip	50	50.01	Top	0.27mm	0.27mm	GND2
2	Offset Stripline	50	49.98	GND2	0.18mm	0.18mm	Top,L3
3	Offset Stripline	50	49.98	L3	0.18mm	0.18mm	GND2,L4

Layer Stack Legend



Material	Layer	Thickness	Df	Dk	Process	Manufacturer	Frequency
Surface Material	Top Overlay						
Top Solder	Top Solder	0.01mm		3,5			
Copper	Top	0.02mm					
Prepreg		0.15mm	0,02	4,1	Altium Designer	1GHz	
CF-004	GND2	0.02mm			Altium Designer		
Core		0.51mm	0,02	4,5	Altium Designer	1GHz	
CF-004	L3	0.02mm			Altium Designer		
Prepreg		0.15mm	0,02	4,1	Altium Designer	1GHz	
CF-004	L4	0.02mm			Altium Designer		
Core		0.51mm	0,02	4,5	Altium Designer	1GHz	
CF-004	GND5	0.02mm			Altium Designer		
Prepreg		0.15mm	0,02	4,1	Altium Designer	1GHz	
Copper	Bottom	0.02mm					
Surface Material	Bottom Solder	0.01mm		3,5			
Bottom Overlay	Bottom Overlay						

Total thickness: 1.60mm

Drill Table

Symbol	Count	Hole Size	Plated	Hole Tolerance
◇	386	0.20mm	Plated	-0.13mm
□	4	0.60mm	Non-Plated	+/-0.05mm
○	2	2.50mm	Plated	+0.05mm/-0.18mm
392 Total				

- Template of the document sheet. You only need to fill in the fields that are important to you.
- The board view of any layer or any combination of layers.
- The X, Y Axis tool.
- Table with the main parameters of transmission lines
- Detailed display of layer stackup
- The annotation tools that are typical for professional MCADs.
- Callout
- Technical requirements with reference to the callout.

THRU HOLE PROJECTION

PART NO: =PCB_PART_NUMBER	APPROVALS	DATE
ENGINEER: =PCB_ENGINEER	=PCB_ENGINEER	
DESIGNER: =PCB_DESIGNER	=PCB_DESIGNER	
CHECKER: =PCB_CHECKER	=PCB_CHECKER	
BOM DOC: =DOC_NO_BOM	TITLE: SRC-Kame FMU	
ASSY DOC: =DOC_NO_FAB_DWG	DESIGN ITEM REVISION: .ItemRevision	
SCH DOC: =DOC_NO_SCH_DWG	SIZE: A3	CAGE CODE: =CAGE_CODE
PCB DOC: =PCB_DWG_NO	FILE NAME: Kame_FM_U_FAB.PCBDwf	SHEET: 1 OF 1

.It =DOC_NO_ASSY_DWG Client