THIS DOCUMENT AND THE DATA DISCLOSED HEREIN OR HEREWITH IS THE PROPERTY OF ALTIUM LIMITED AND MAY BE FREELY DISTRIBUTED IN WHOLE. NO RIGHTS ARE RESERVED OR EXPRESS OR IMPLIED WARANTEE GIVEN.

The Main Advantages when using Draftsman to Create an Assembly Drawing

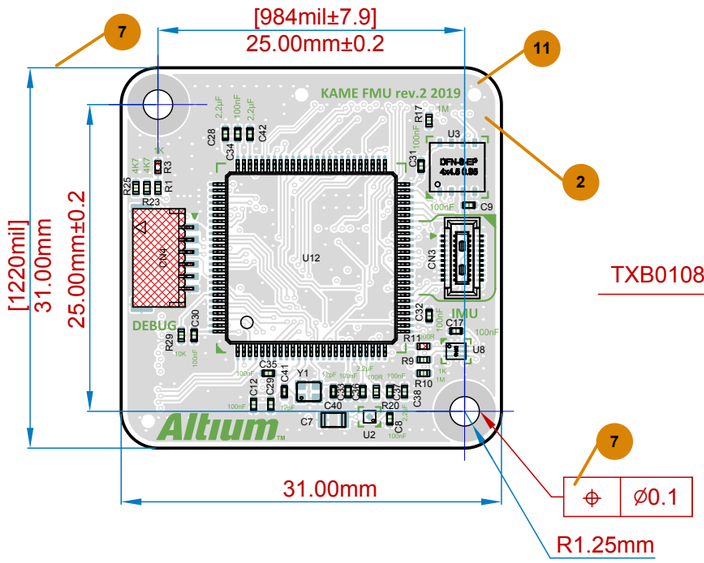
REV STATUS OF SHEETS											
REV											
SHEET											

DWG NO: =DOC_NO_ASSY_DWG REV: .Ite

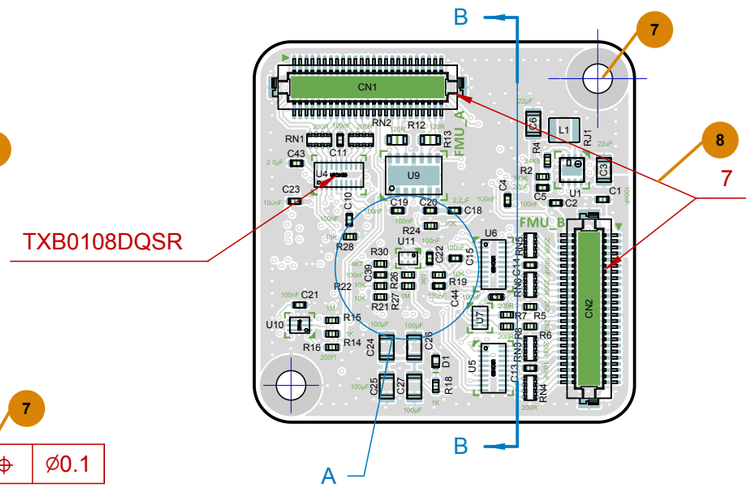
REVISIONS

ZONE	REV	DESCRIPTION	DATE	APPROVED

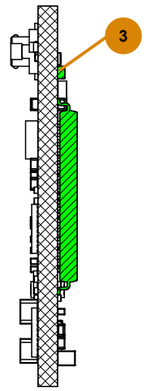
View from Top side (Scale 2:1)



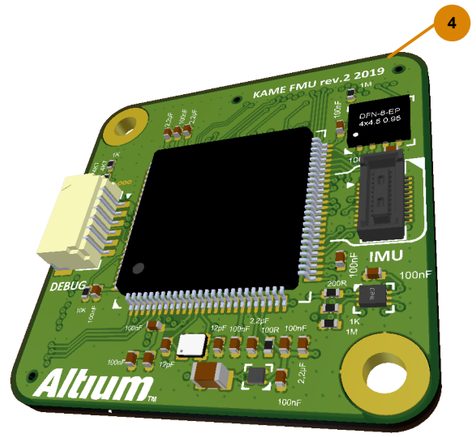
View from Bottom side (Scale 2:1)



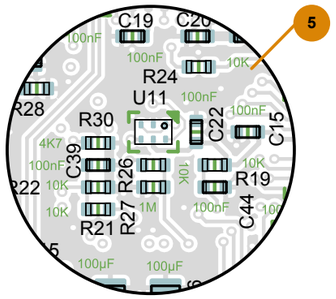
SECTION B-B (Scale 2:1)



Realistic View



DETAIL A (Scale 4:1)

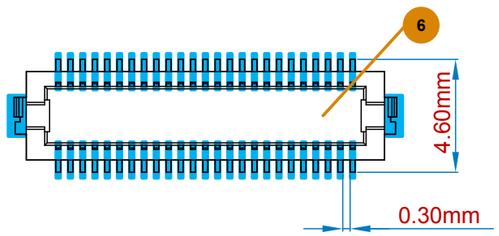


Bill Of Materials

9

Line #	Designator	Comment	Quantity
1	C1, C2, C4, C8, C9, C10, C11, C12, C13, C14, C15, C16, C17, C19, C20, C21, C22, C23, C30, C31, C32, C33, C34, C35, C37, C39, C44	Capacitor 100nF +/-20% 10V 0402, 100nF, 100nF , 100nF, 100nF	27
2	C3, C6, C7	Capacitor 22µF +/-40% 16V 0805	3
3	C5	Capacitor 22pF +/-10% 50V 0402	1

Component View DF12(3.0)-50DS-0.5V(86)(HIRO-DF12(3.0)-50DS-0.5V_V) (Scale 4:1)

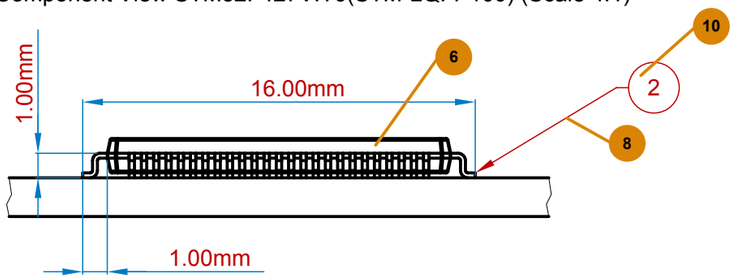


Notes:

10

1. Dimensions for reference.
2. Soldering with solder paste ULF-208-98

Component View STM32F427VIT6(STM-LQFP100) (Scale 4:1)



1. Template of the document sheet. You only need to fill in the fields that are important to you.
2. The board view with components and layout. Component graphics are sourced from the 3D model projections. No need to create additional graphics and layers.
3. The board section view. Create the section where the board should be shown with more details.
4. The board realistic view. Can be used as a reference view of the assembled board.
5. The detail view. Allows you to increase the detail level of a particular PCB area.
6. The component view. You can place a view of a particular component, measure it, and show the details of its mounting on the board.
7. The annotation tools that are typical for professional MCADs.
8. The callout. Using this multifunctional tool, you can enter component positions from the BOM, component parameters, or just text." (component positions with a BOM table and a specification item - aren't these the same).
9. BOM table. BOM is located directly in your drawing.
10. Technical requirements with reference to the callout.
11. Display the board view with components for a specific variant.

.It

DWG NO: =DOC_NO_ASSY_DWG REV: .Item

PART NO: =PCB_PART_NUMBER

APPROVALS	DATE
ENGINEER: =PCB_ENGINEER	=PCB_ENGINEER
DESIGNER: =PCB_DESIGNER	=PCB_DESIGNER
CHECKER: =PCB_CHECKER	=PCB_CHECKER

Reference Documents

BOM DOC: =DOC_NO_BOM
ASSY DOC: =DOC_NO_FAB_DWG
SCH DOC: =DOC_NO_SCH_DWG
PCB DOC: =PCB_DWG_NO

NEXT ASSY | USED ON
APPLICATION

1 THIRD ANGLE PROJECTION

Altium

DESIGN ITEM: .Item
DESIGN ITEM REVISION: .ItemRevision
TITLE: =PCB_TITLE_1 =PCB_TITLE_2
SIZE: A3 CAGE CODE: =CAGE_CODE DWG NO: REV:
SCALE: FILE NAME: Kame_FMU_AS.PCBDwf SHEET: 1 OF 1