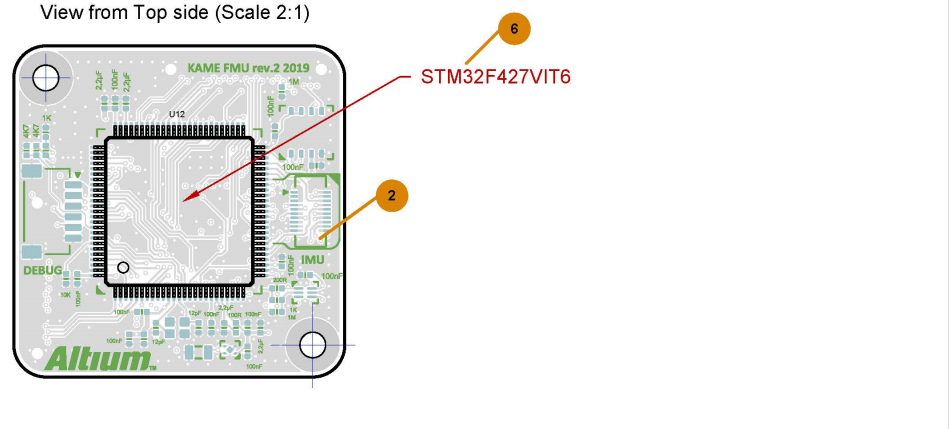
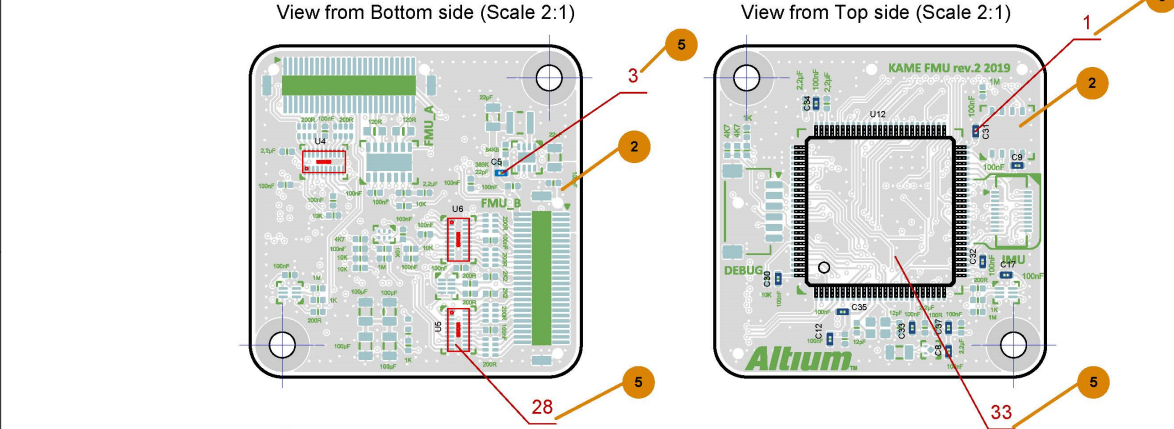


REV STATUS		REV	REVISIONS									
OF SHEETS	SHEET		ZONE	REV	DESCRIPTION		DATE		APPROVED			



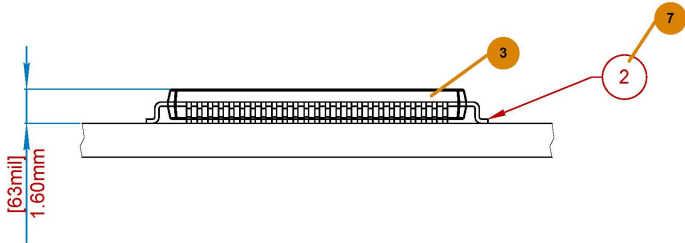
Bill Of Materials

Line #	Designator	Comment	Quantity
27	U3	FM25V02A-DGTR	1
28	U4, U5, U6	TXB0108DQSR	3
29	U7	TXS0102DQMR	1
30	U8, U10	TSV991AIQ1T	2

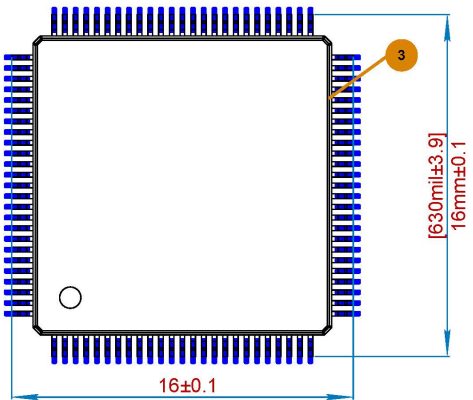
Bill Of Materials

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

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

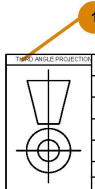


Component View STM32F427VIT6(STM-LQFP100) (Scale 4:1)



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

- 1 Document sheet template. You only need to fill in the important fields, others can be left blank.
- 2 Board view with a specific component or group of components
- 3 Component view. If needed, you can place a view of a specific component, include measurements, and show assembly details.
- 4 BOM table. In some drawings with smaller number of placements or components, it is appropriate to put the BOM directly into the drawing.
- 5 Component location with a callout line referring to a specific line in the BOM table.
- 6 Callout line with a component part number.
- 7 Callout line referencing an assembly note.



PART NO: =PCB_PART_NUMBER		DATE		<b>Altium</b>	
ENGINEER: =PCB_ENGINEER		=PCB_ENGINEER			
DESIGNER: =PCB_DESIGNER		=PCB_DESIGNER		.Item	
CHECKER: =PCB_CHECKER		=PCB_CHECKER			
BOM DOC: =DOC_NO_BOM		TITLE: =PCB_TITLE_1		.ItemRevision	
ASSY DOC: =DOC_NO_FAB_DWG		=PCB_TITLE_2			
SCH DOC: =DOC_NO_SCH_DWG		SIZE: CAGE CODE: DWG NO:			
PCB DOC: =PCB_DWG_NO		SCALE: A3=CAGE_CODE			
APPLICATION		FILE NAME: Kame_FMU_AS1.PCBDwf		SHEET 1 OF 1	