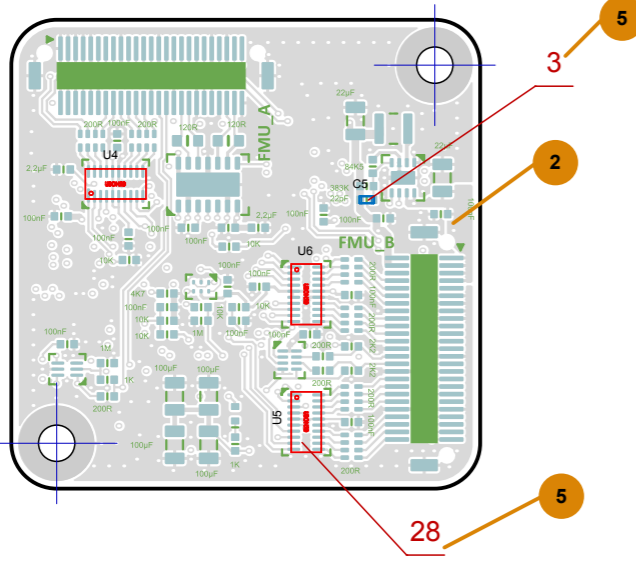


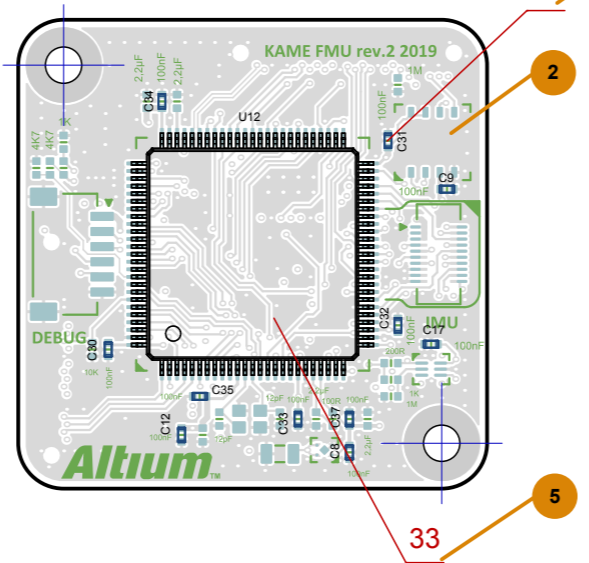
# The Main Advantages when using Draftsman to Create a Component Mounting Drawing

REV STATUS OF SHEETS		REV	DATE	DESCRIPTION	DATE	APPROVED

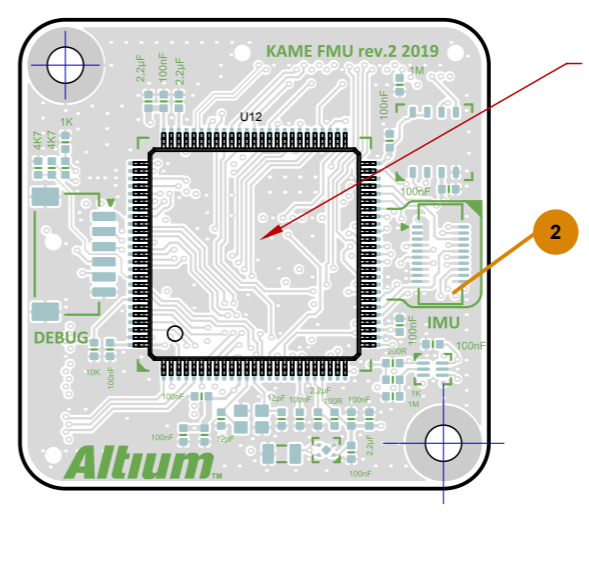
View from Bottom side (Scale 2:1)



View from Top side (Scale 2:1)



View from Top side (Scale 2:1)



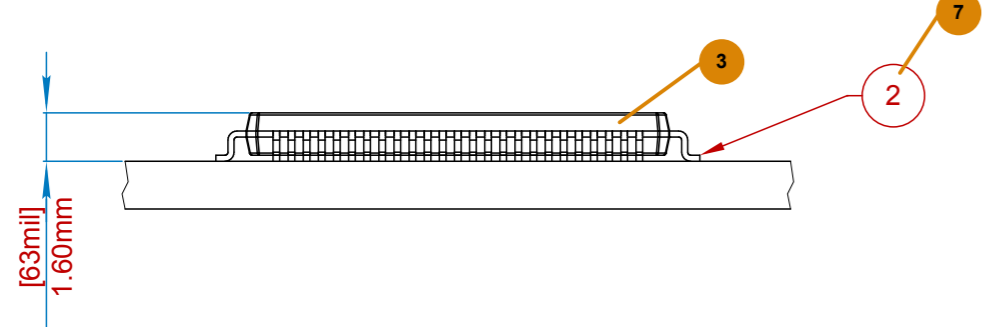
Bill Of Materials

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

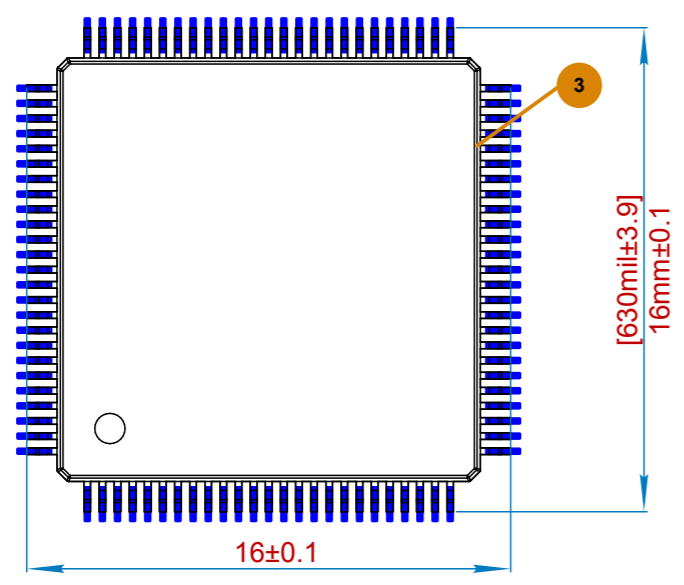
Bill Of Materials

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)



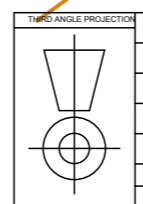
Component View STM32F427VIT6(STM-LQFP100) (Scale 4:1)



Notes:

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

- 1 Template of the document sheet. You only need to fill in the fields that are important to
- 2 The board view with a specific component or group of components.
- 3 The component view. You can place a view of a particular component, measure it, and show the details of its mounting on the board.
- 4 BOM table. BOM is located directly in your drawing.
- 5 Component position with reference to a BOM table
- 6 A callout line with a component parameter
- 7 Technical requirements with reference to the callout.



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