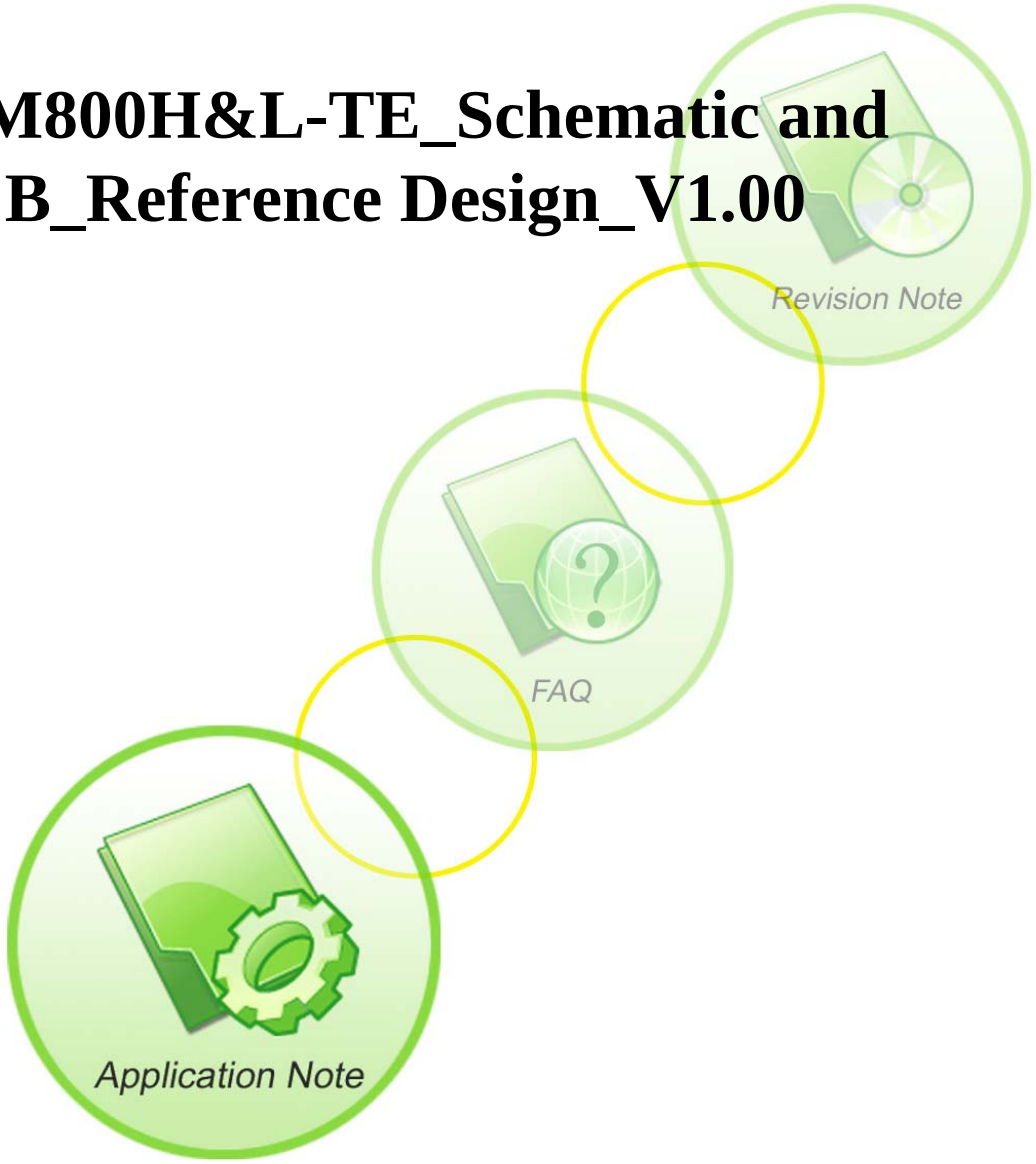
SIM800H&L-TE_Schematic and PCB_Reference Design_V1.00



Document Title	SIM800H&L-TE_Schematic and PCB_ Reference Design
Version	1.00
Date	2013-10-14
Status	Release
Document Control ID	SIM800H&L-TE_Schematic and PCB_ Reference Design_V1.00

General Notes

SIMCom offers this information as a service to its customers, to support application and engineering efforts that use the products designed by SIMCom. The information provided is based upon requirements specifically provided to SIMCom by the customers. SIMCom has not undertaken any independent search for additional relevant information, including any information that may be in the customer's possession. Furthermore, system validation of this product designed by SIMCom within a larger electronic system remains the responsibility of the customer or the customer's system integrator. All specifications supplied herein are subject to change.

Copyright

Contents

1 Introduce 6

2 Schematic 6

2.1 Power Supply 6
2.2 Audio 7
2.3 USB 7
2.4 Antenna of GSM and Bluetooth 8
2.5 Antenna of FM 9

3 PCB Introduce 10

3.1 PCB Stack-up 13
3.2 PCB Layout 14
3.3 Traces Routing 17
3.4 Power Supply Part 18
3.5 GND 20
3.6 Audio Part 22
3.7 USB 25
3.8 RF Trace to Antenna 26

Figure Index

Figure 1：Schematic of the power supply part 6
Figure 2：Schematic of the audio part 7
Figure 3：Schematic of the USB part 7
Figure 4：Schematic of the GSM and Bluetooth antenna 8
Figure 5：Schematic of the FM antenna 9
Figure 6：Layer 1 10
Figure 7：Layer 2 11
Figure 8：Layer 3 12
Figure 9：Layer 4 13
Figure 10: PCB stack-up 14
Figure 11: PCB via 14
Figure 12：Pin assignment 15
Figure 13：Top side placement 16
Figure 14：Bottom side placement 17
Figure 15：LGA module traces routing 18
Figure 16：Power supply trace on layer 3 19
Figure 17：Power supply trace on layer 1 20
Figure 18：GND on layer 1 21
Figure 19：GND on layer 4 22
Figure 20: Audio trace on layer 1 23
Figure 21: Audio trace on layer 2 23
Figure 22：Audio trace on layer 3 24
Figure 23：Audio trace on layer 4 24
Figure 24：USB trace on layer 1 25
Figure 25：USB trace on layer 2 25
Figure 26：USB trace on layer 3 26
Figure 27：USB trace on layer 4 26
Figure 28：Calculating the RF trace impendence 27
Figure 29：RF trace width and the separation from GND 27
Figure 30：GSM RF trace and the antenna connectors 28
Figure 31：Bluetooth RF trace and the antenna connectors 29
Figure 32：FM traces and the antenna connectors 29

Version History:

Data	Version	Description of change	Author
2013-10-14	1.00		Jialin.song, Ya.li

1 Introduce

This document introduces the SIM800H&L-TE_V1.02 schematic and PCB layout, also describes the key points about circuit design, component placement, PCB layout and RF trace design. Customer can refer to these key points when they design the products with SIM800H or SIM800L module.

2 Schematic

2.1 Power Supply

Power input range is 3.4V~4.4V of the SIM800H&L-TE, following figure is the schematic of power supply part：

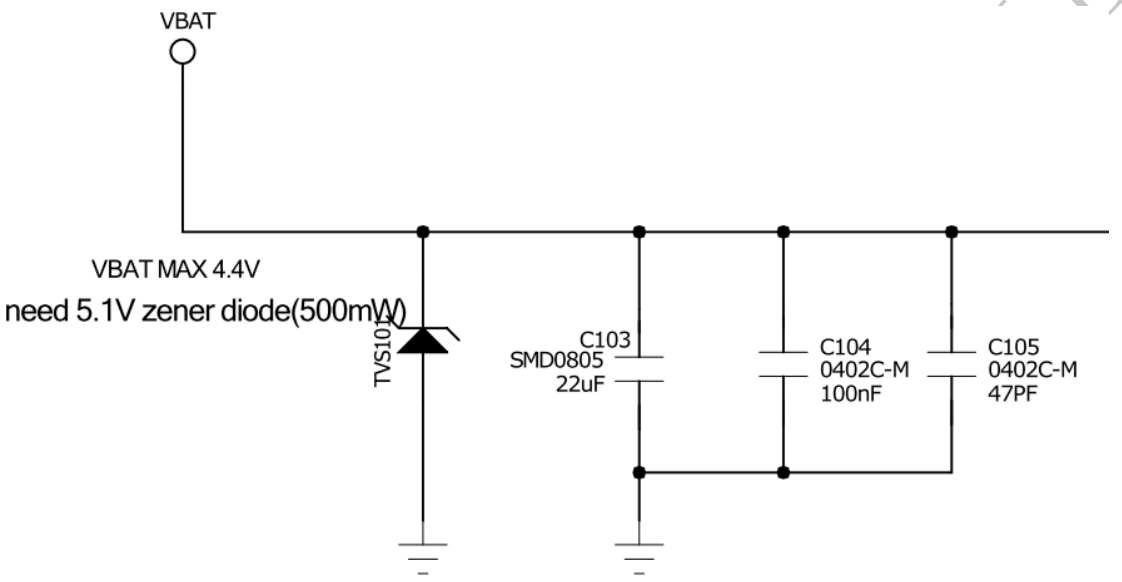


Figure 1：Schematic of the power supply part

2.2 Audio

Following figure is schematic of the audio part：

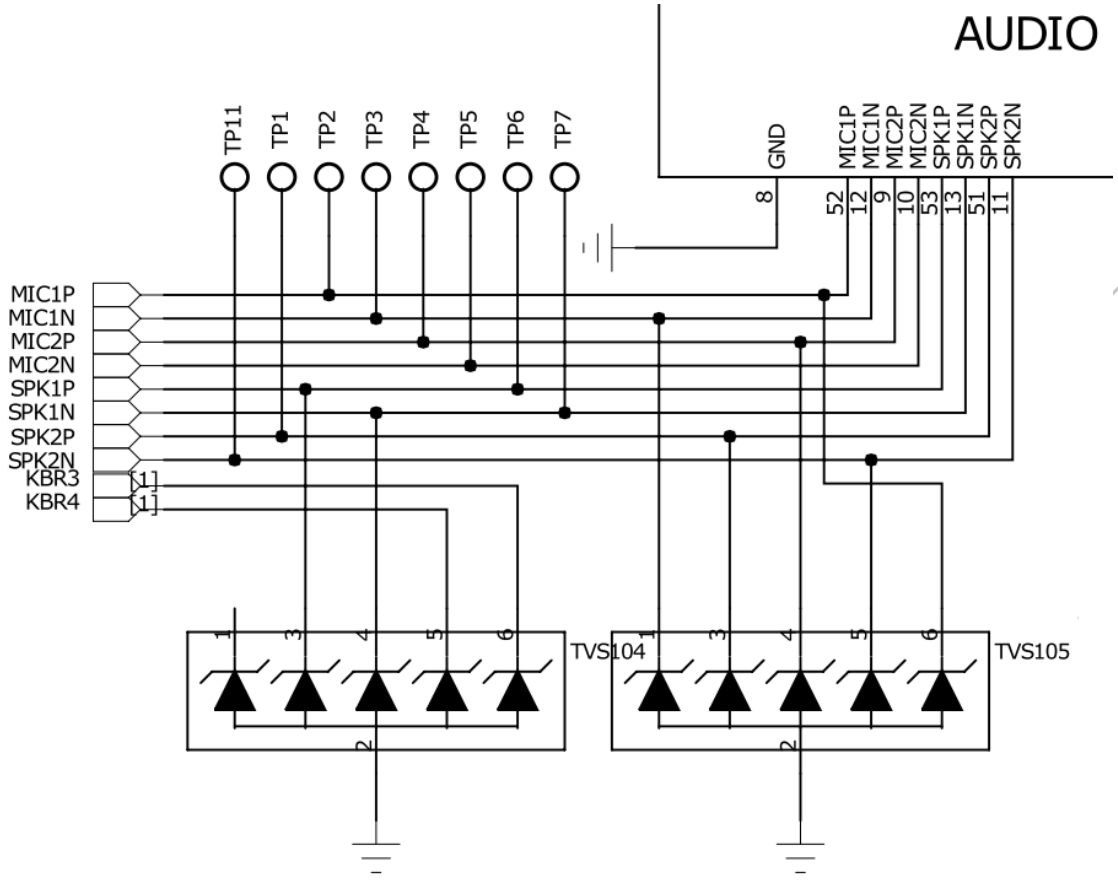


Figure 2：Schematic of the audio part

2.3 USB

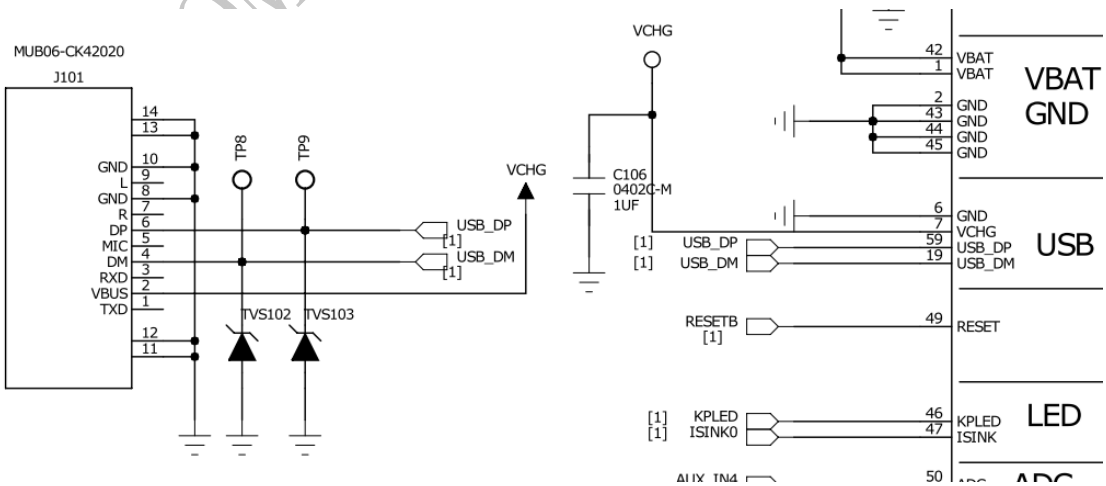


Figure 3：Schematic of the USB part

2.4 Antenna of GSM and Bluetooth

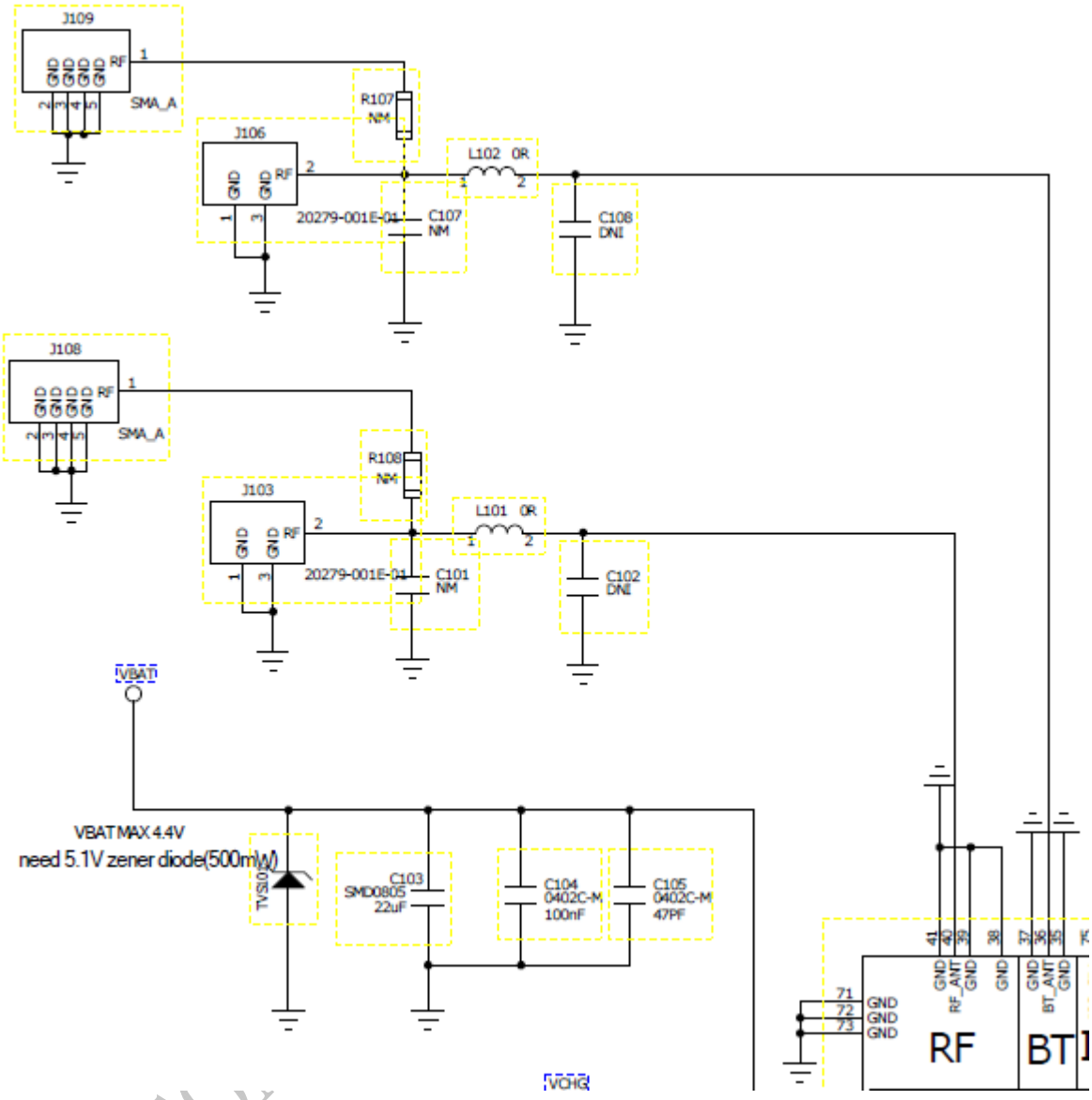


Figure 4：Schematic of the GSM and Bluetooth antenna

2.5 Antenna of FM

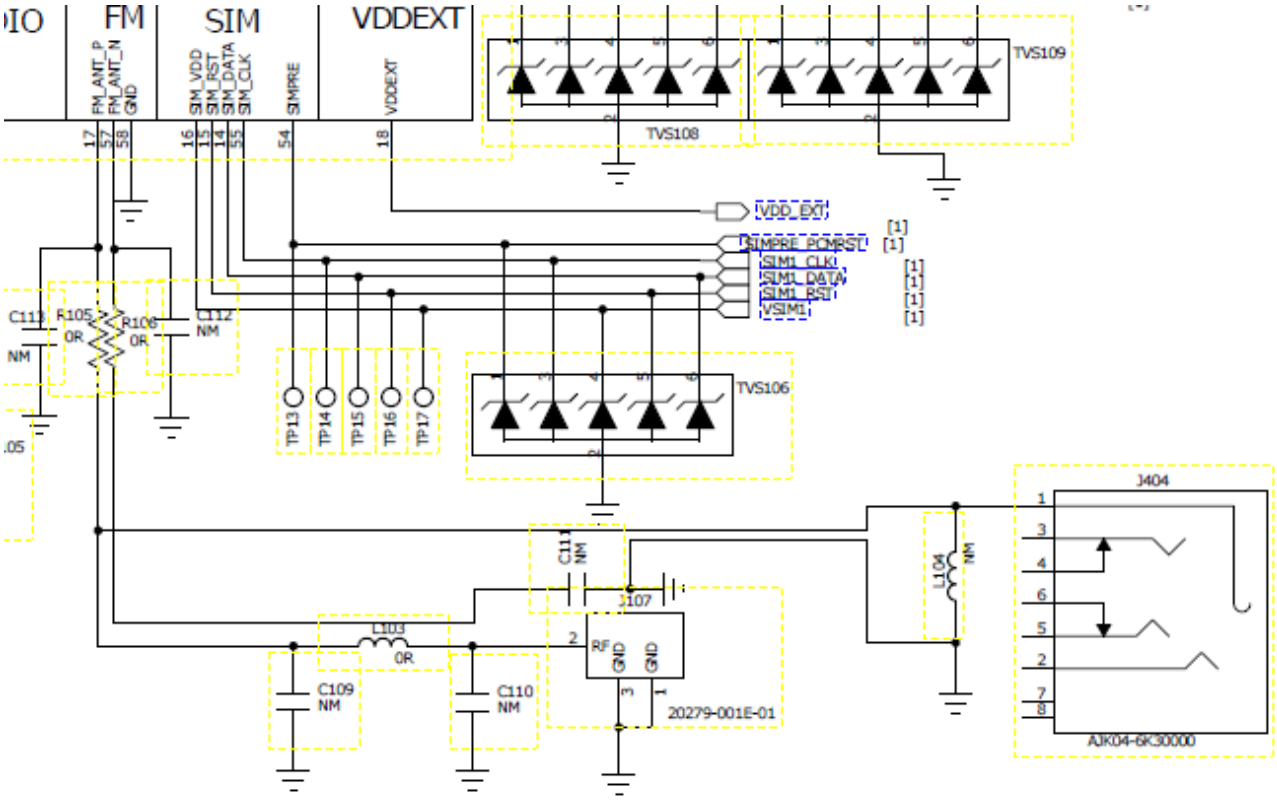


Figure 5：Schematic of the FM antenna

3 PCB Introduce

SIM800H&L-TE_V1.02 is four-layer PCB, each layer is shown as below figures:

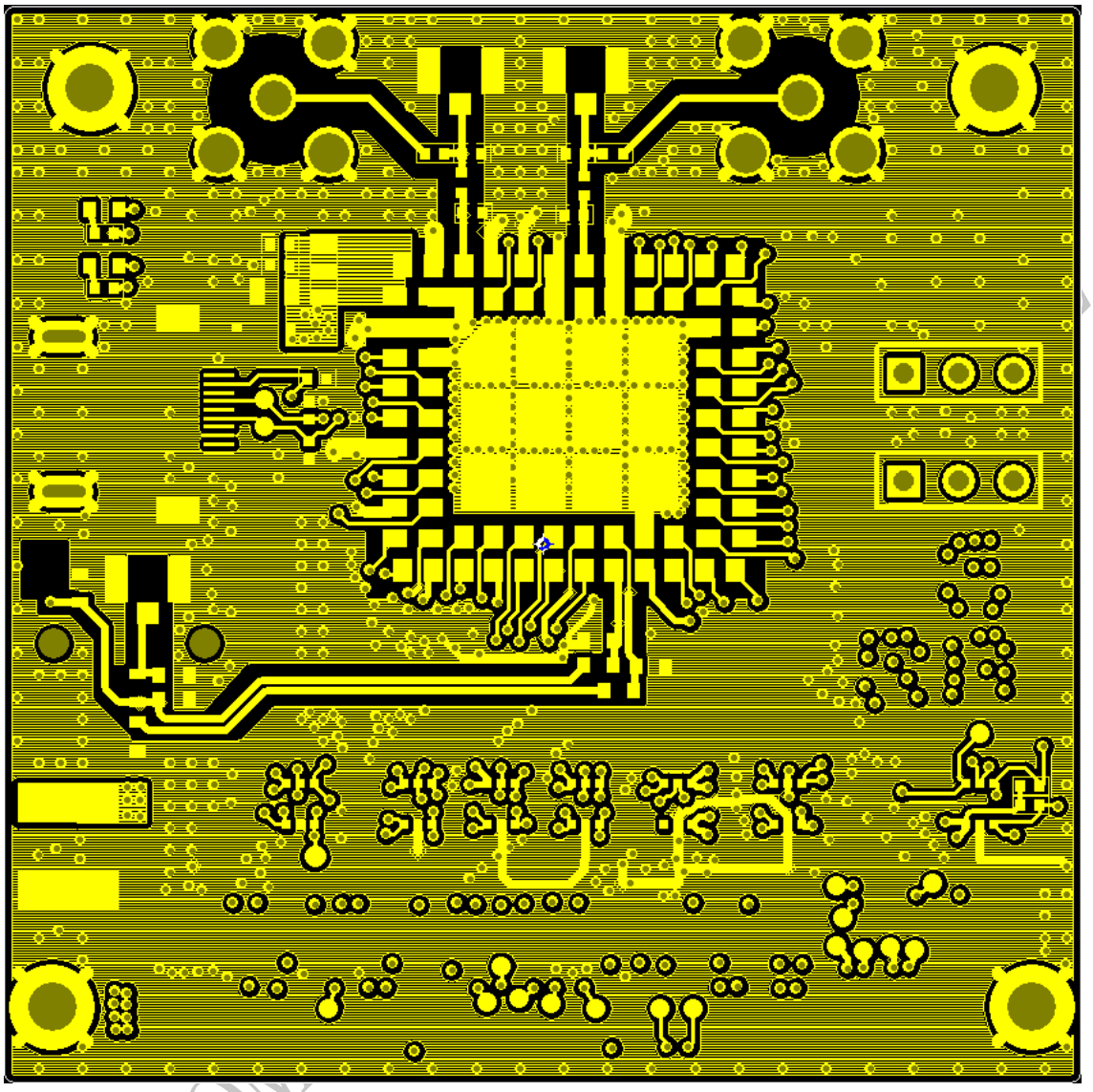


Figure 6：Layer 1

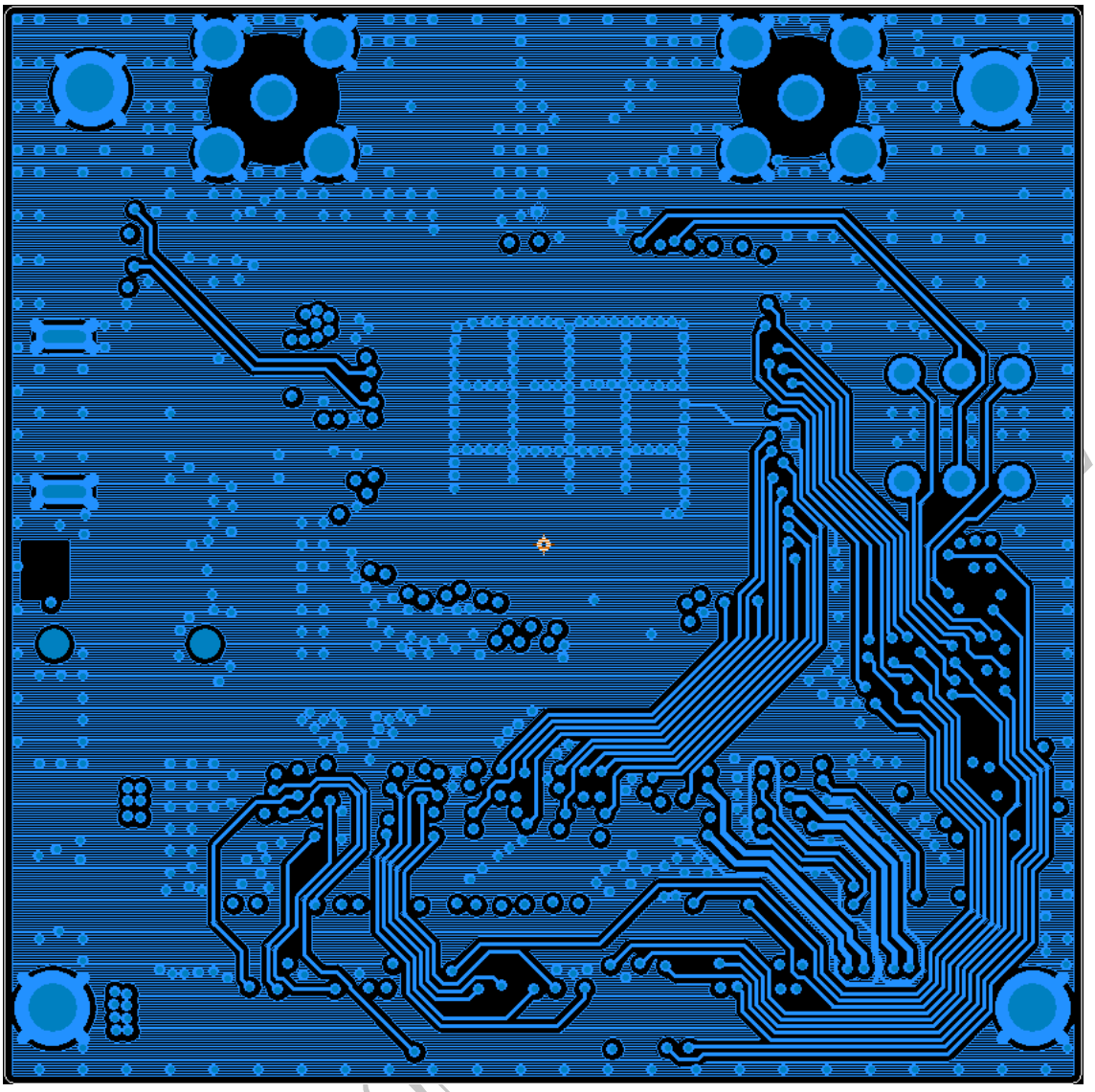


Figure 7：Layer 2

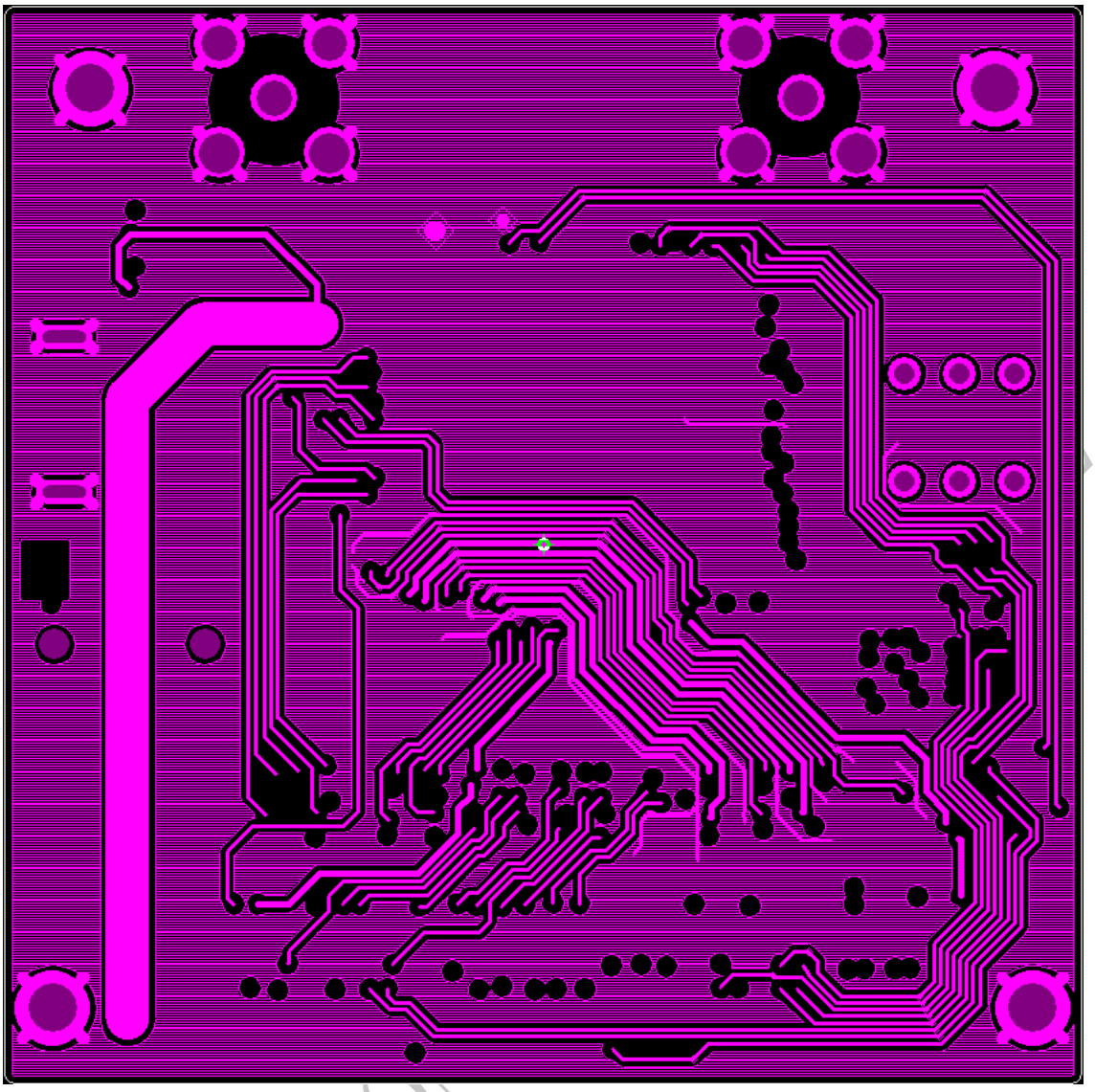


Figure 8：Layer 3

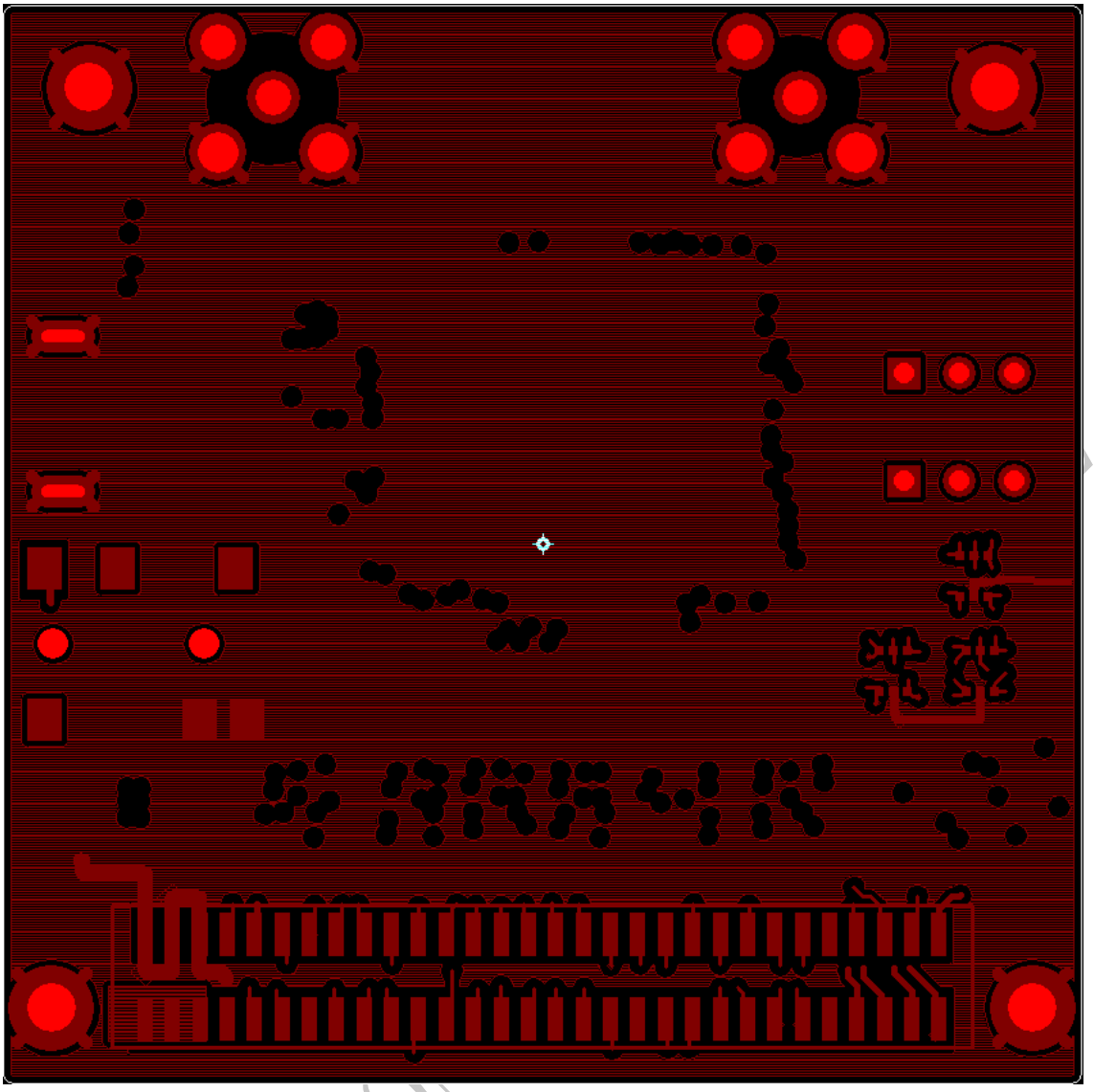


Figure 9：Layer 4

3.1 PCB Stack-up

The SIM800H&L-TE is a four layer PCB and the PCB's total thickness is 1.0mm, PCB stack-up is shown as figure 10.

All vias of the PCB are through-hole vias and the diameter of the vias is 0.55mm, shown as figure 11.

	Solder Mask	18.0 µm.
	Chemical Gold	0.05 µm.
	Electroless Nickel	2.54 µm.
	Copper Plating	25 µm.
Layer 1	Copper	18.0 µm.
	1080LDP	200.0 µm
Layer 2	Copper	18.0 µm.
	Core	485.0 µm
Layer 3	Copper	18.0µm.
	1080LDP	200.0 µm
Layer 4	Copper	18.0 µm.
	Copper plating	25 µm.
	Electroless Nickel	2.54 µm.
	Chemical Gold	0.05 µm.
	Solder Mask	18.0 µm.

Figure 10: PCB stack-up

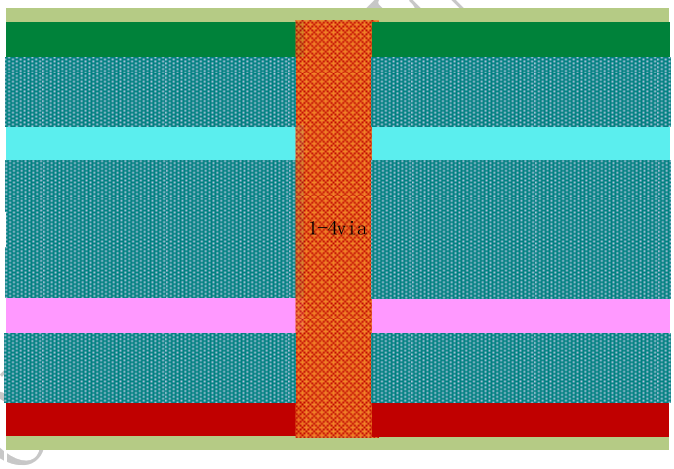


Figure 11: PCB via

3.2 PCB Layout

Before PCB layout, we should learn well about pin assignment in order to get reasonable layout with so many external components. Following figure is the overview of pin assignment of the module.

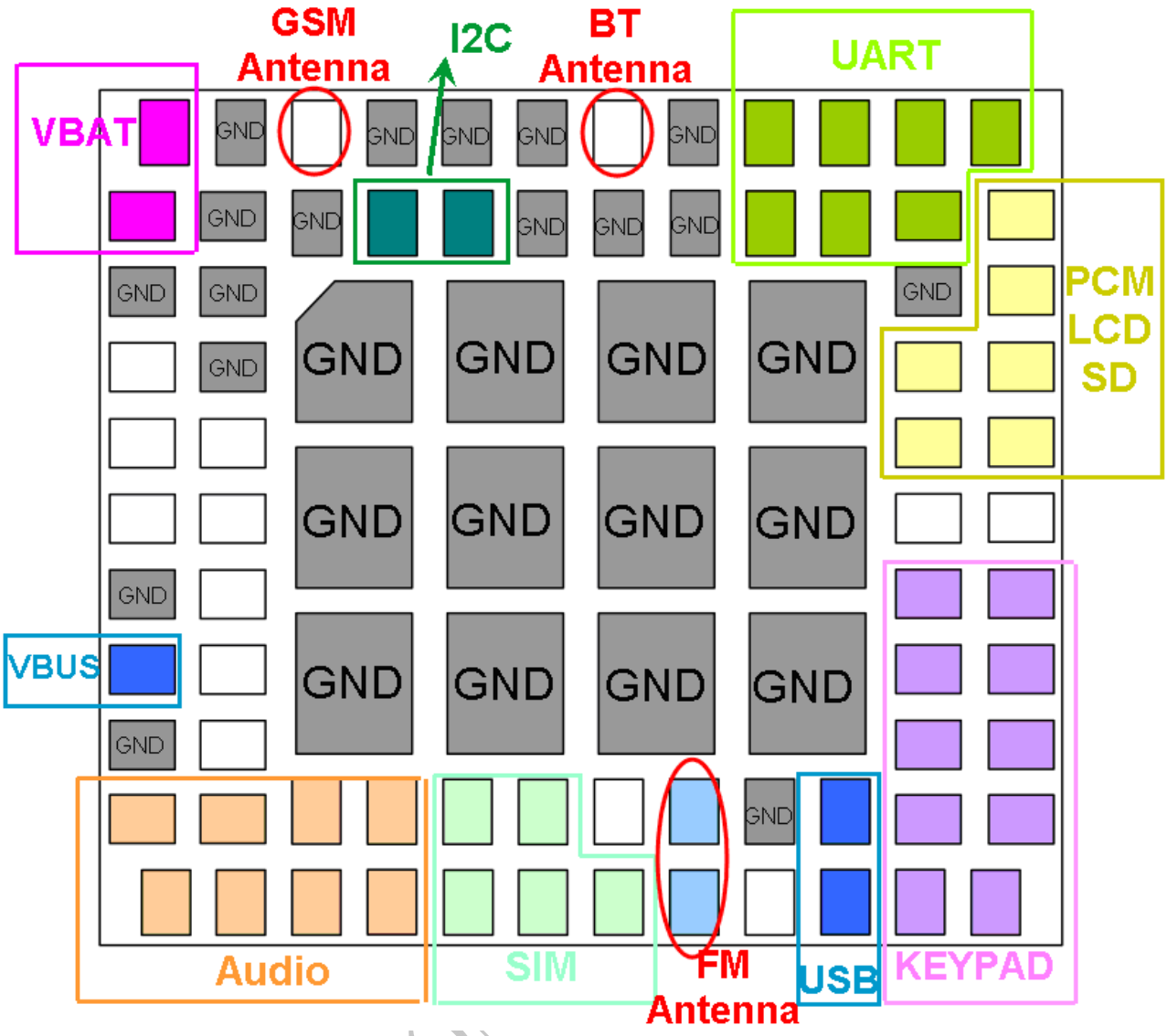


Figure 12：Pin assignment

Following figure is the placement of SIM800H&L-TE_V1.02

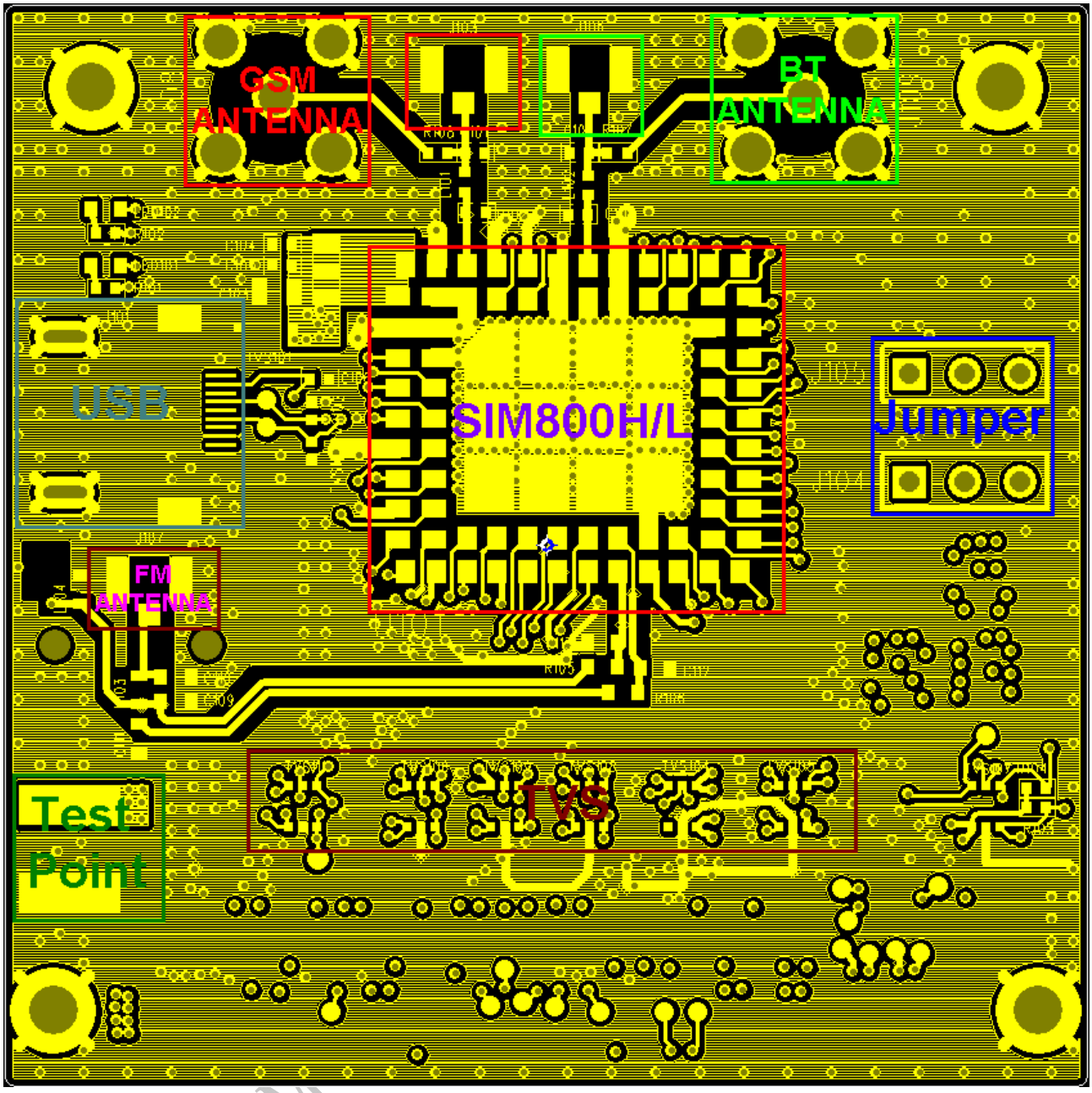


Figure 13：Top side placement

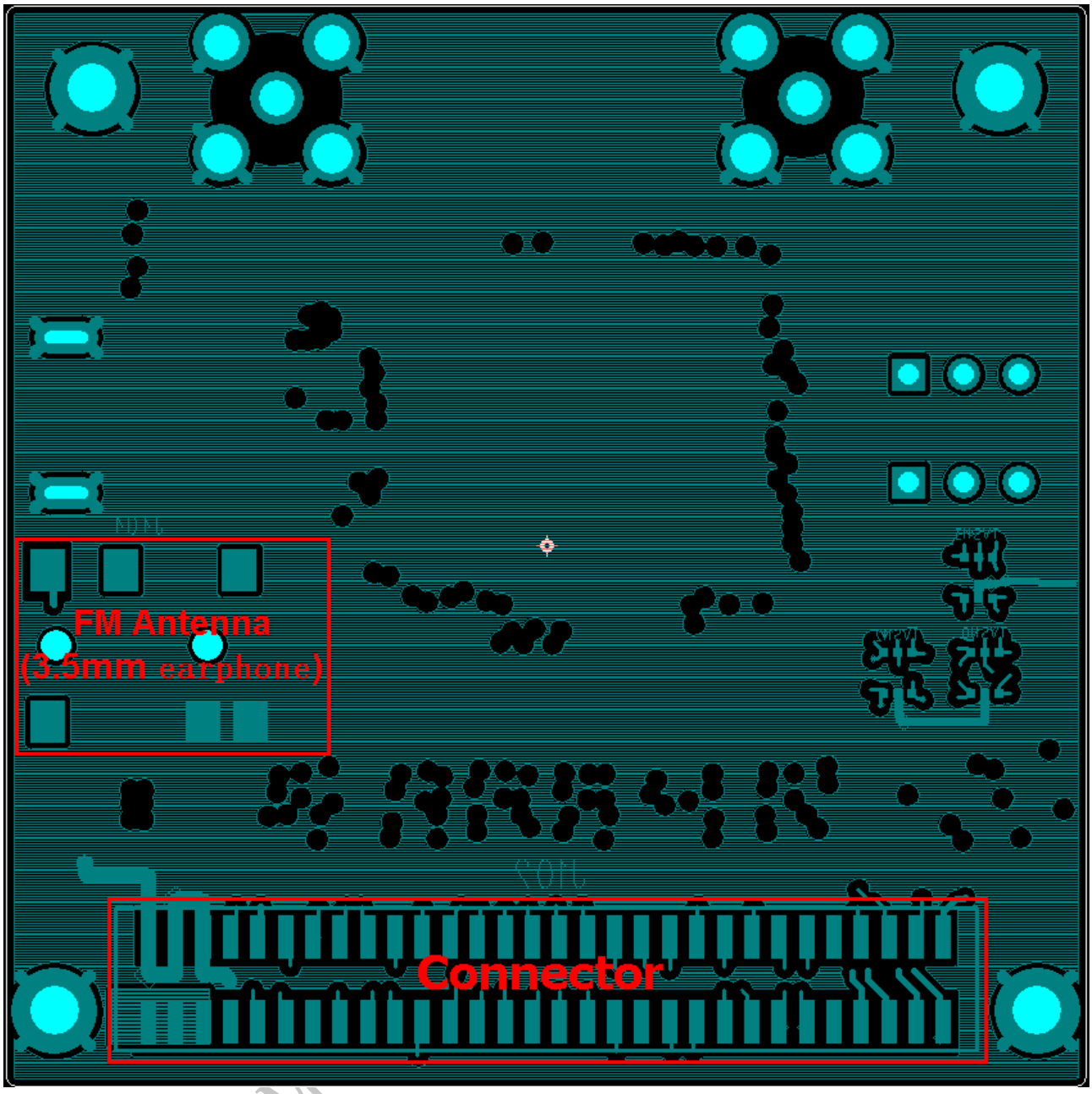


Figure 14：Bottom side placement

3.3 Traces Routing

The form factor of SIM800H and SIM800L module is LGA package, there are two rows of PINs on the board, the overall arrangement of the LGA pads is shown in figure 12. If customer PCB is through-hole design, it's recommended that the signal trace of the inner pads is routed to outside on top side and then drill vias to bottom side.

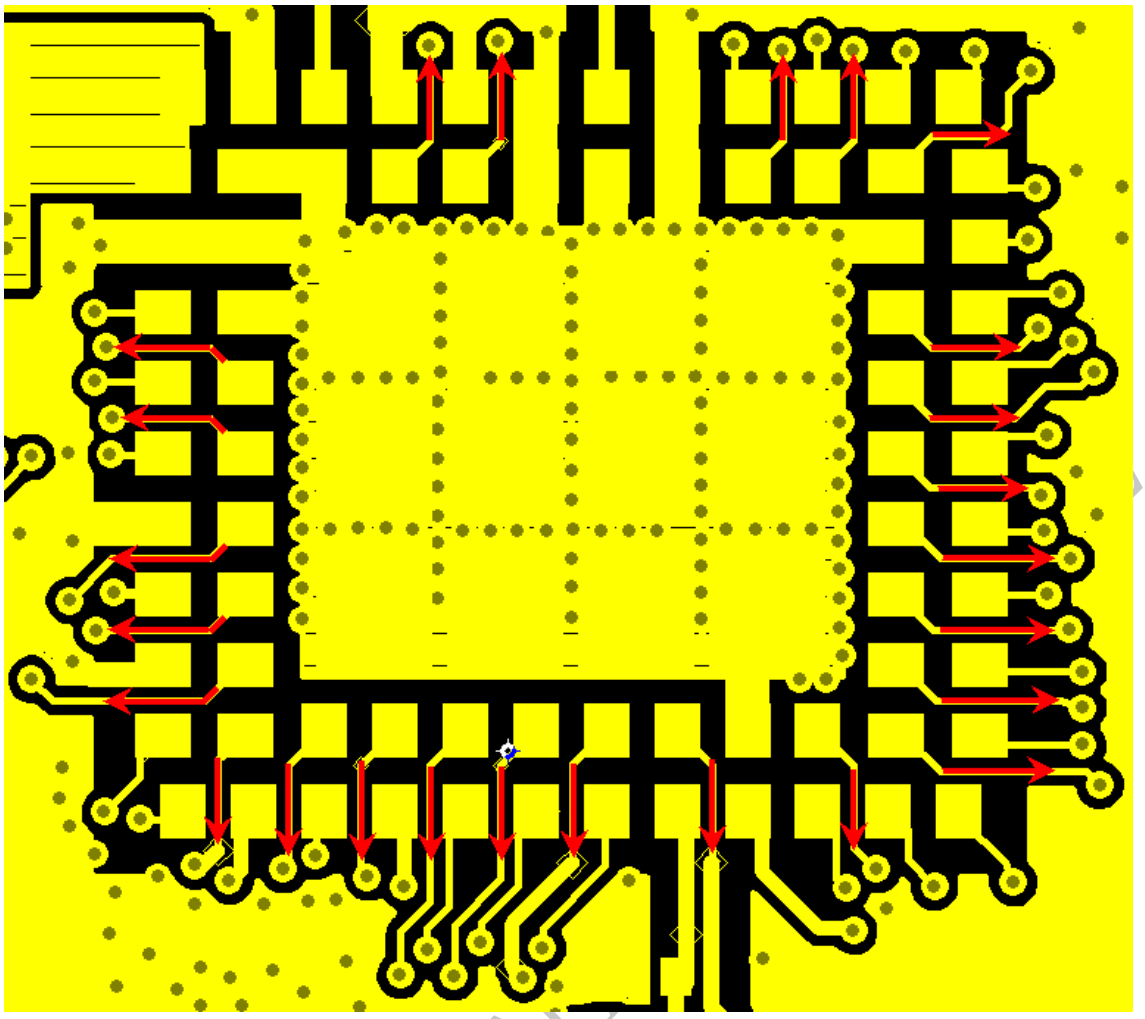


Figure 15：LGA module traces routing

3.4 Power Supply Part

The power supply trace is shown as figure 15 and figure 16.

Zener diode (TVS101) and decoupling capacitors (C103, C104, C105) are mounted on the top layer (layer 1), near to the module's VBAT pads. The VBAT trace is routed from the inner layer (layer 3) to the top layer (layer 1), and the VBAT trace should be more than 2mm, keeping away from RF area. The number of VBAT vias should be as more as possible.

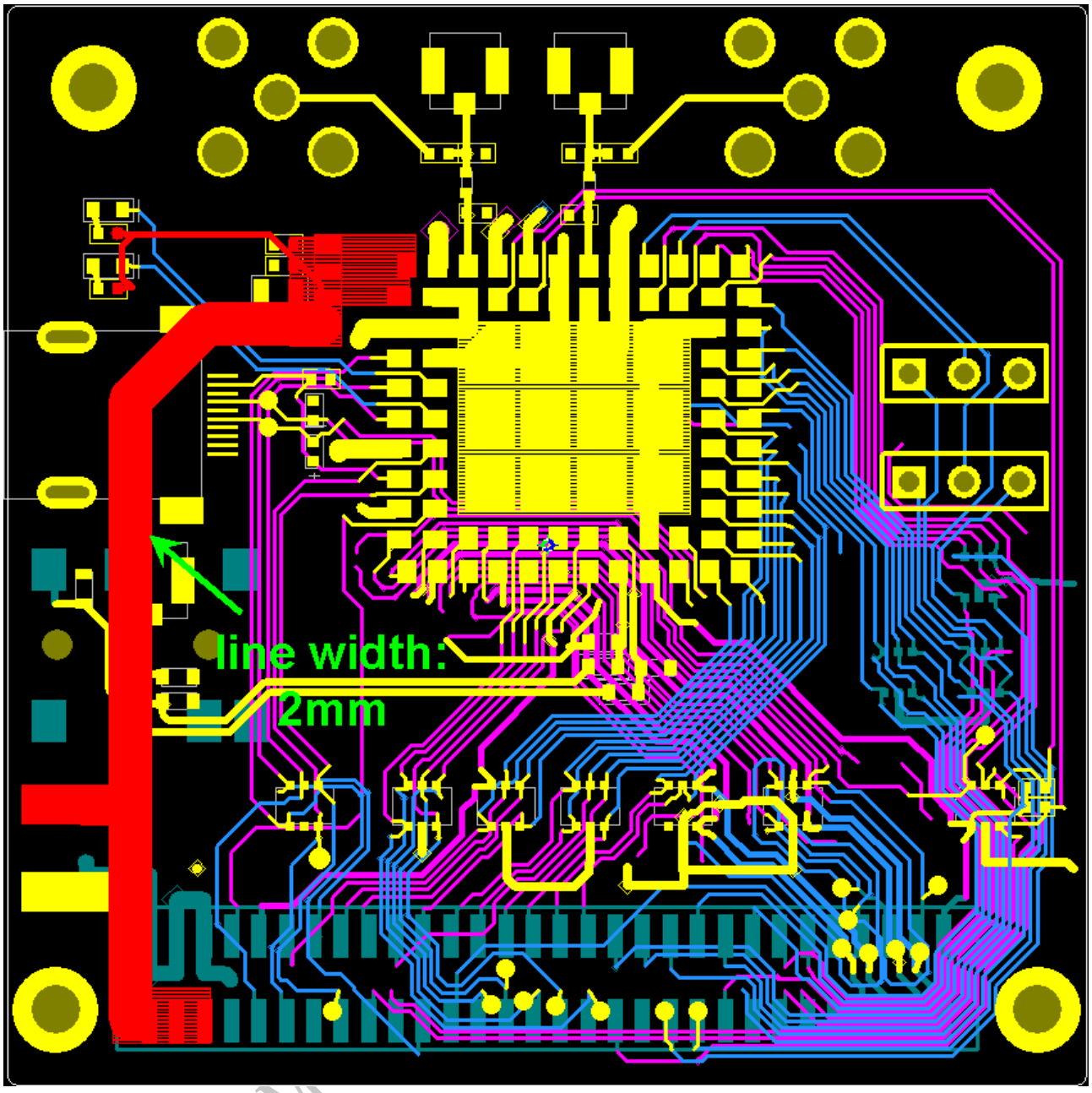


Figure 16：Power supply trace on layer 3

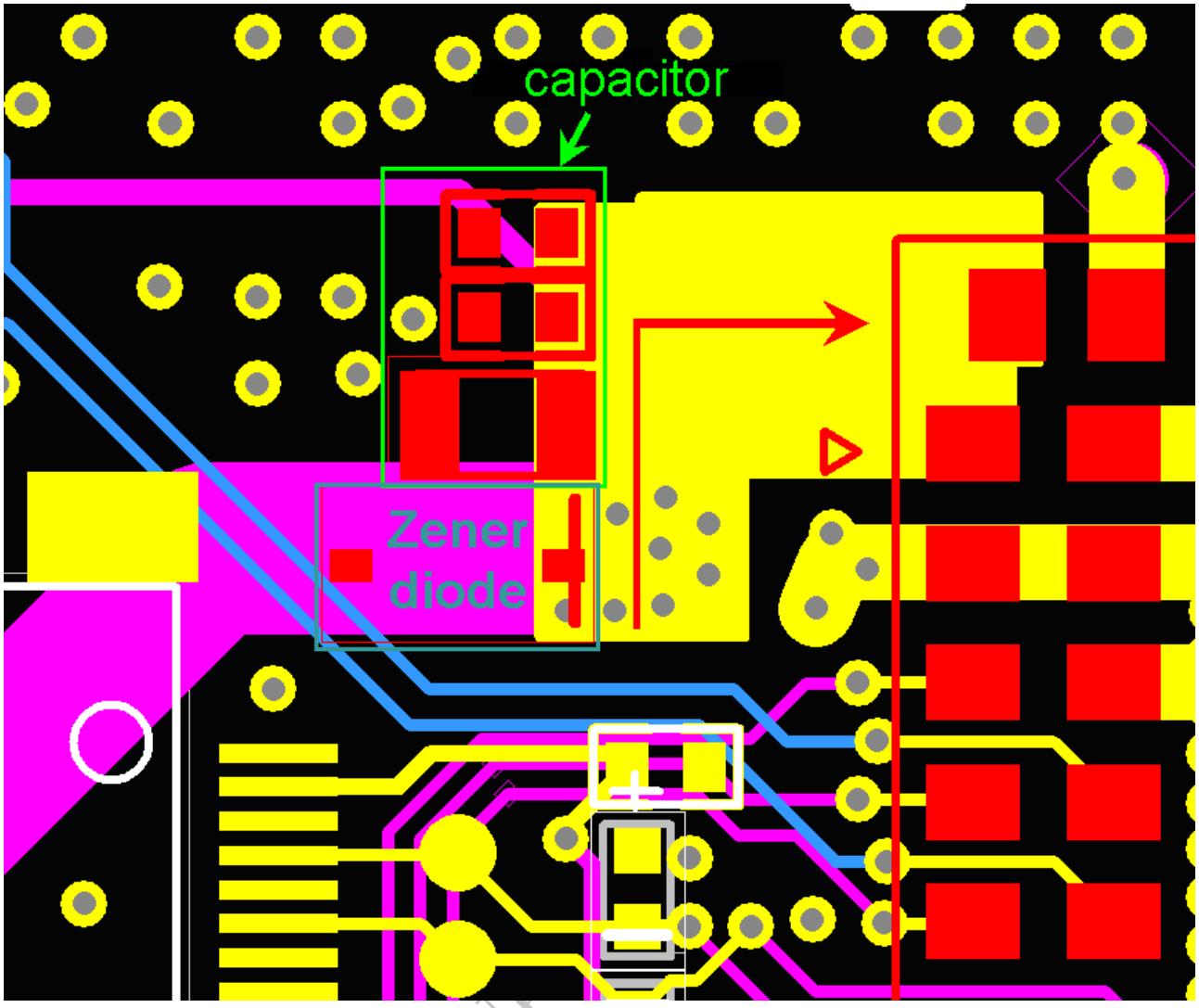


Figure 17：Power supply trace on layer 1

3.5 GND

Pin2, Pin43, Pin44 and Pin45 are main GND signals for VBAT. The traces between these 4pin (Pin2, Pin43, Pin44 and Pin45) and pin77~pin88 should be as short as possible.

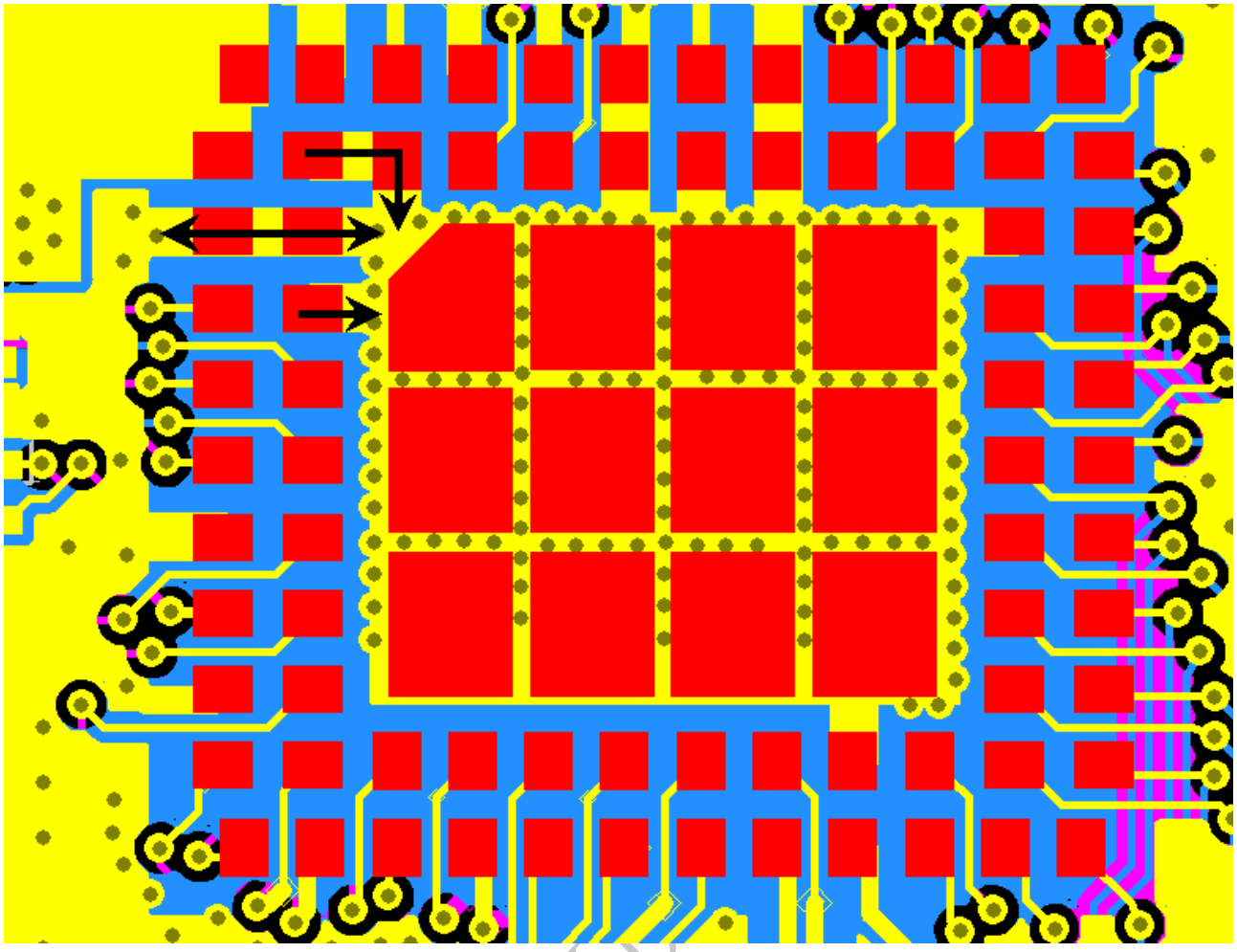


Figure 18：GND on layer 1

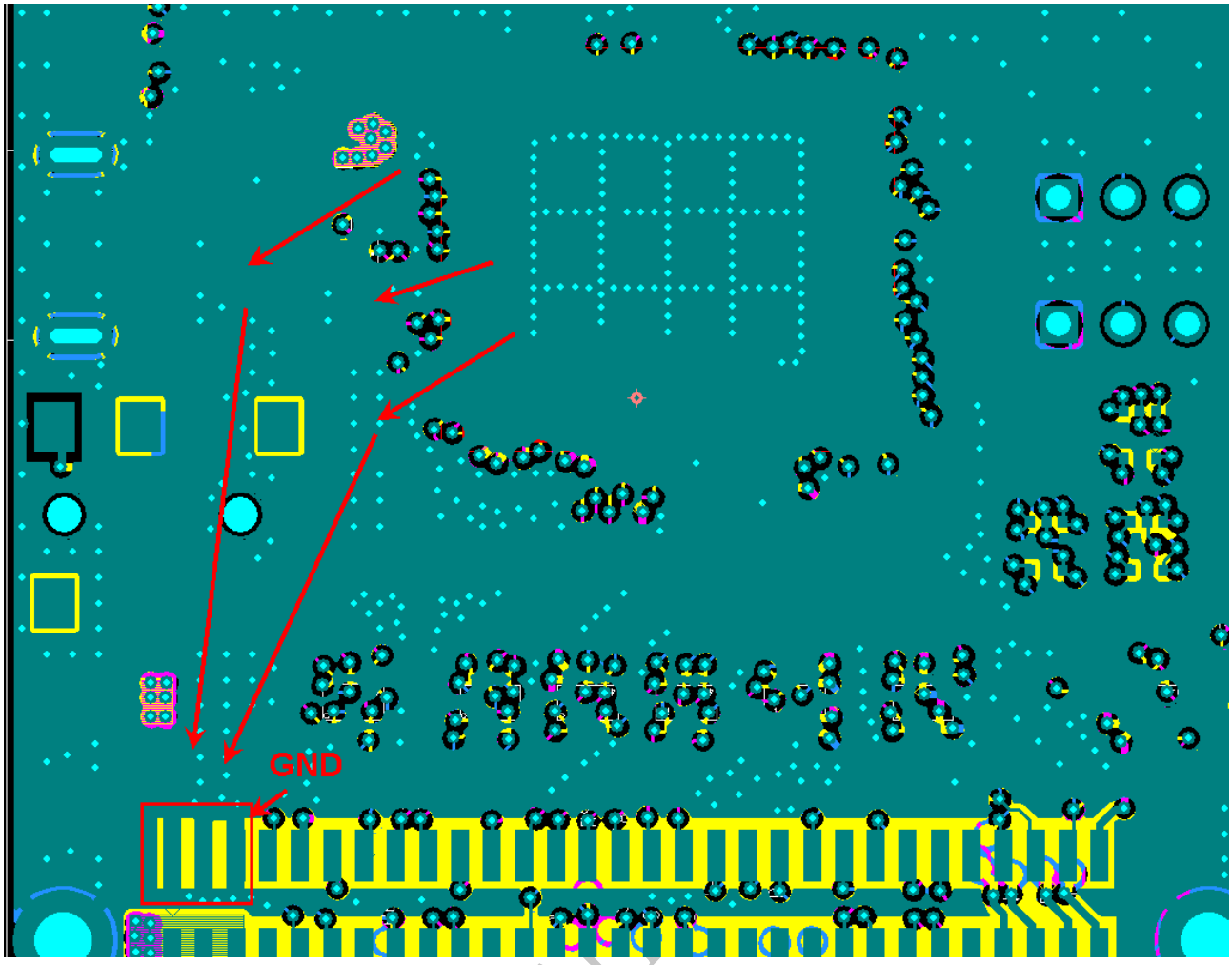


Figure 19：GND on layer 4

3.6 Audio Part

The audio trace is shown as figure 20 to figure 23.

The MIC/SPK traces should be surrounded by ground to get better noise decoupling, and the traces should be more than 0.3mm. The MIC/SPK traces should be routed through TVS firstly, then to the audio pin of module; and the TVS should be placement as close as possible to the connector.

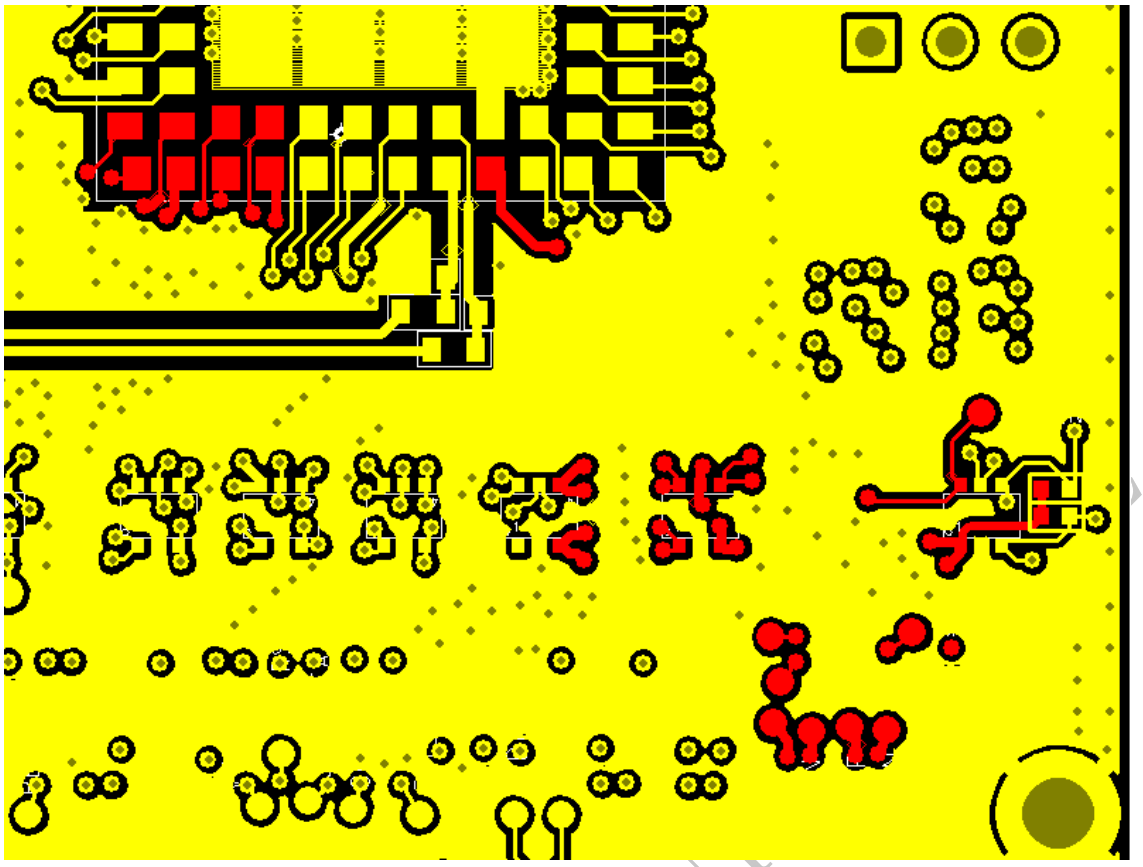


Figure 20: Audio trace on layer 1

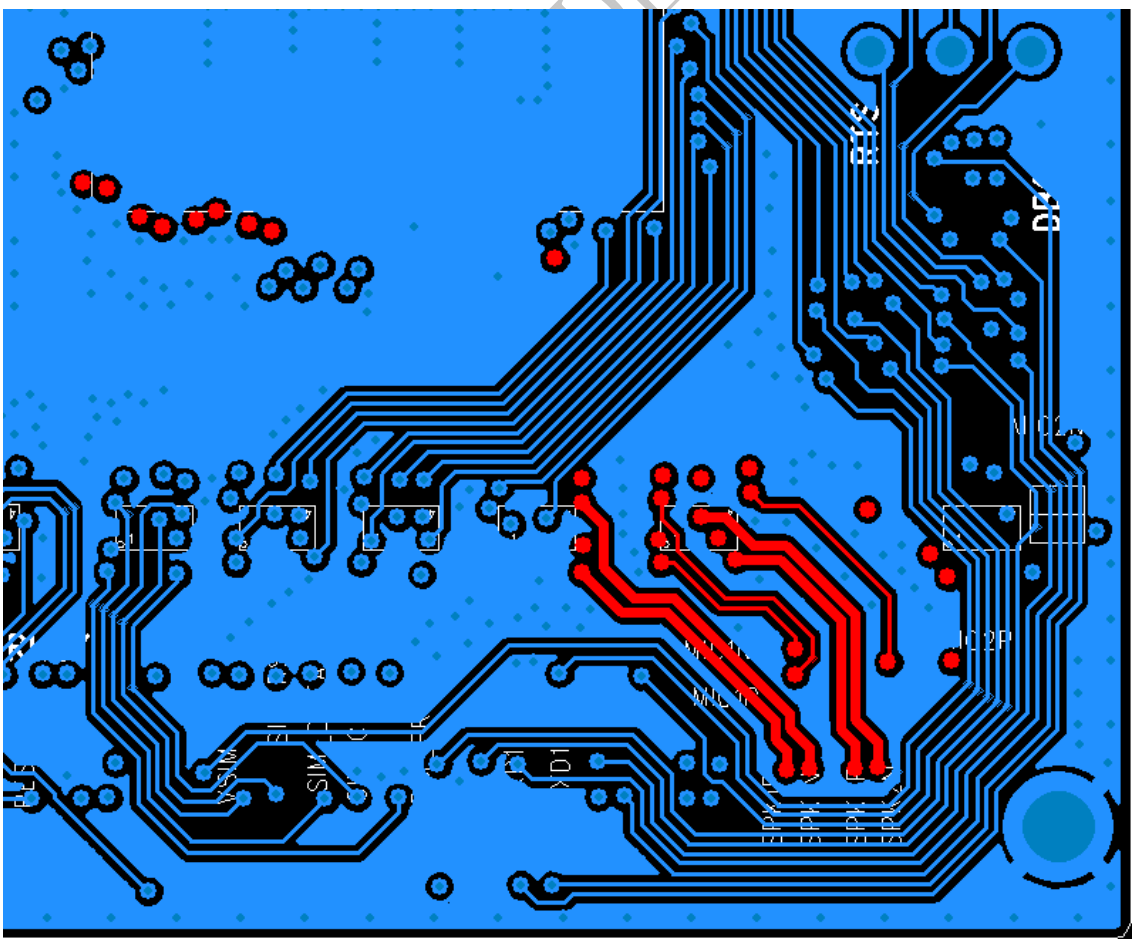


Figure 21: Audio trace on layer 2

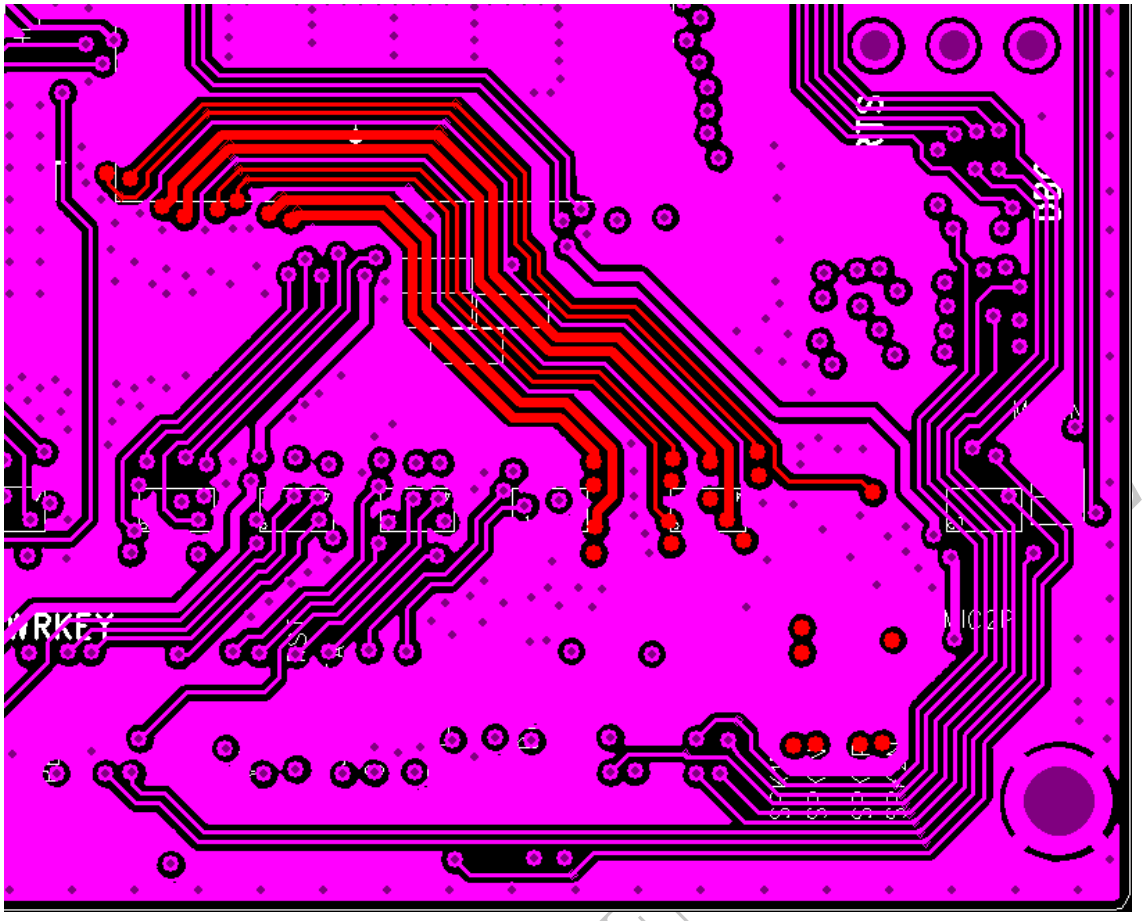


Figure 22：Audio trace on layer 3

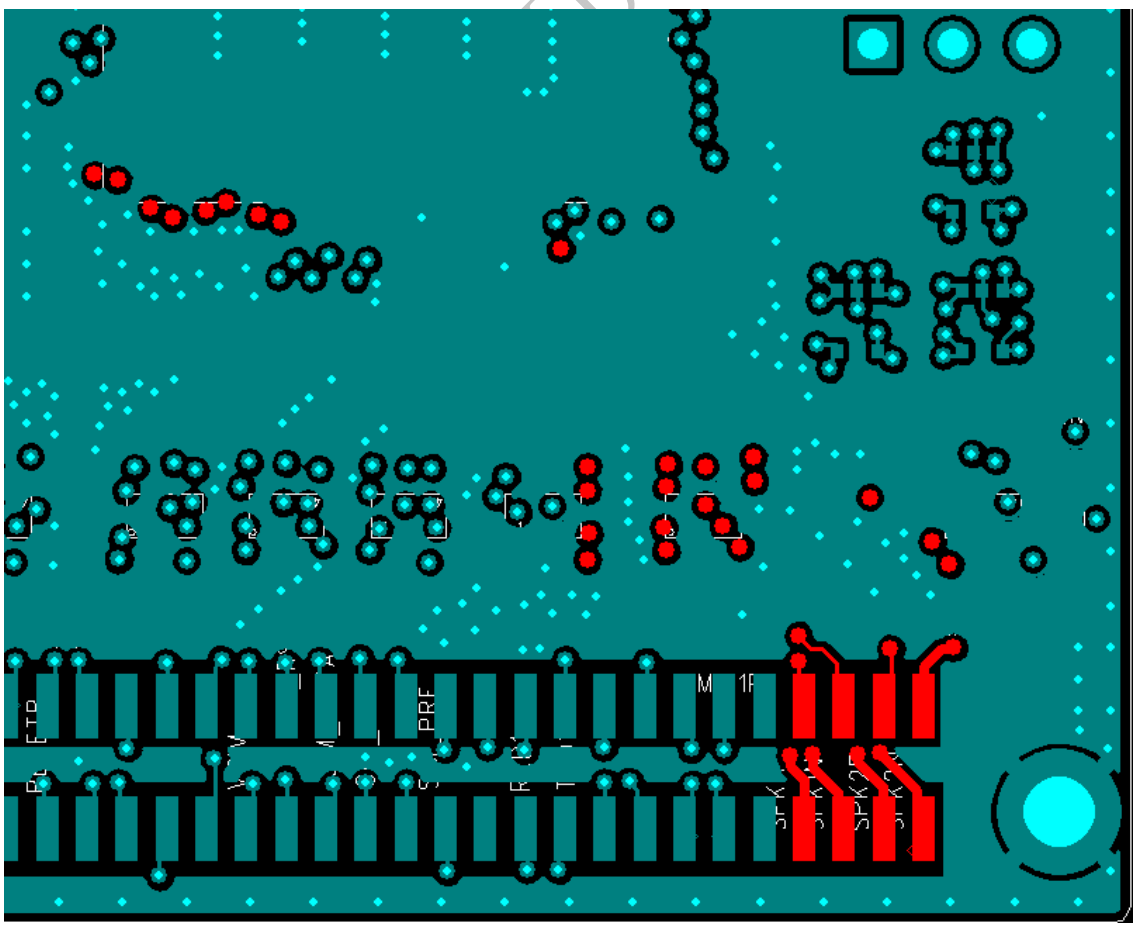


Figure 23：Audio trace on layer 4

3.7 USB

The USB trace is shown as below figure.

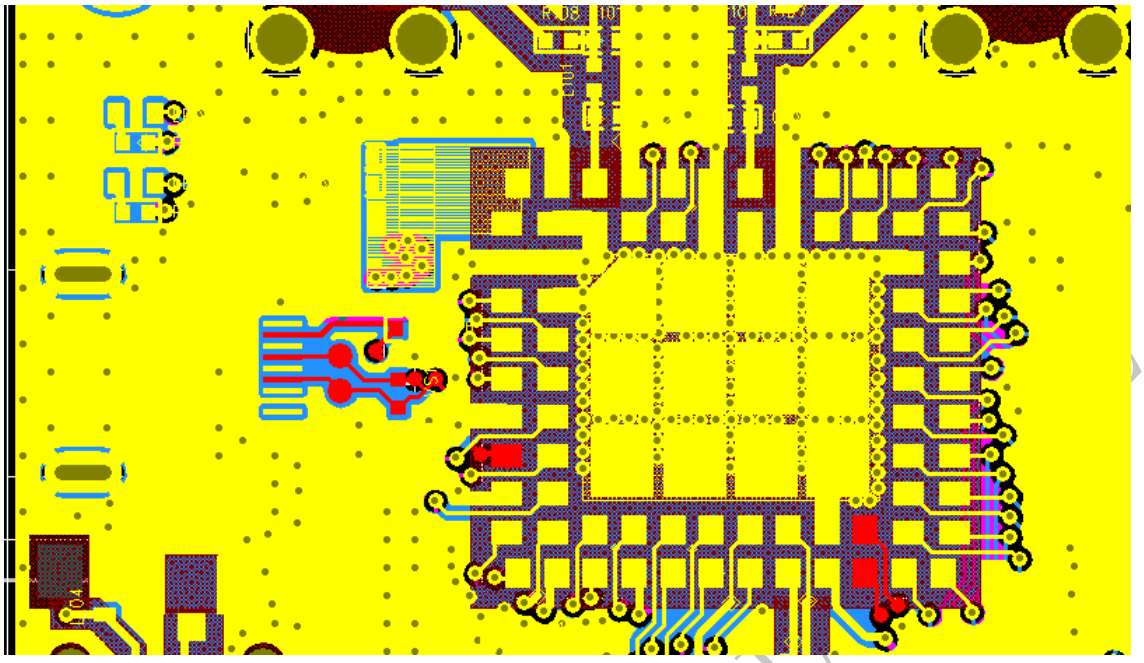


Figure 24：USB trace on layer 1

Figure 25：USB trace on layer 2

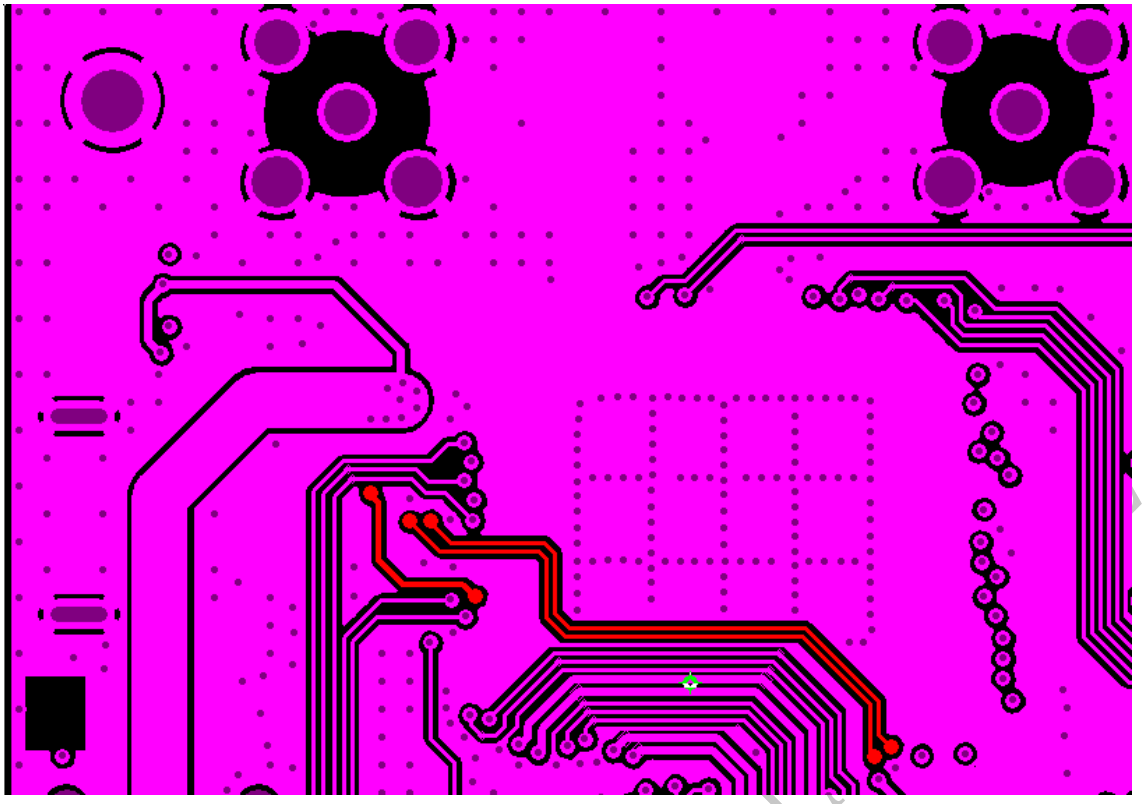


Figure 26：USB trace on layer 3

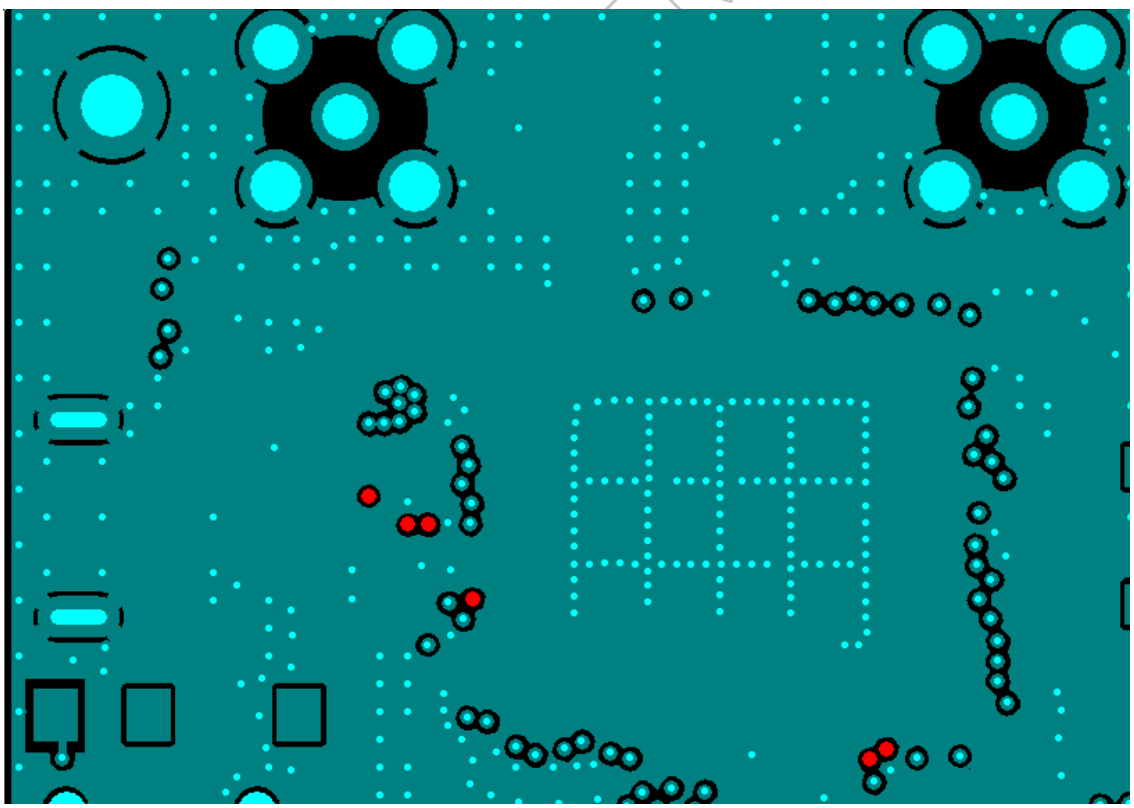


Figure 27：USB trace on layer 4

3.8 RF Trace to Antenna

The RF interface has an impedance of 50Ω. The trace impendence between RF PAD and antenna should be also 50Ω, for the maximum RF power transmitting. To avoid the mismatching and losses by the via, the trace is

recommended to be micro strip line on the top layer instead of the strip line in the inner layers.

SIM800H&L-TE is four-layer PCB, the thickness between top layer and second layer is 200um, the PCB material is FR4. The RF trace is layout on the top layer, with width 0.33mm, and the separation from the GND is 0.6mm. The detail is shown in the following figure:

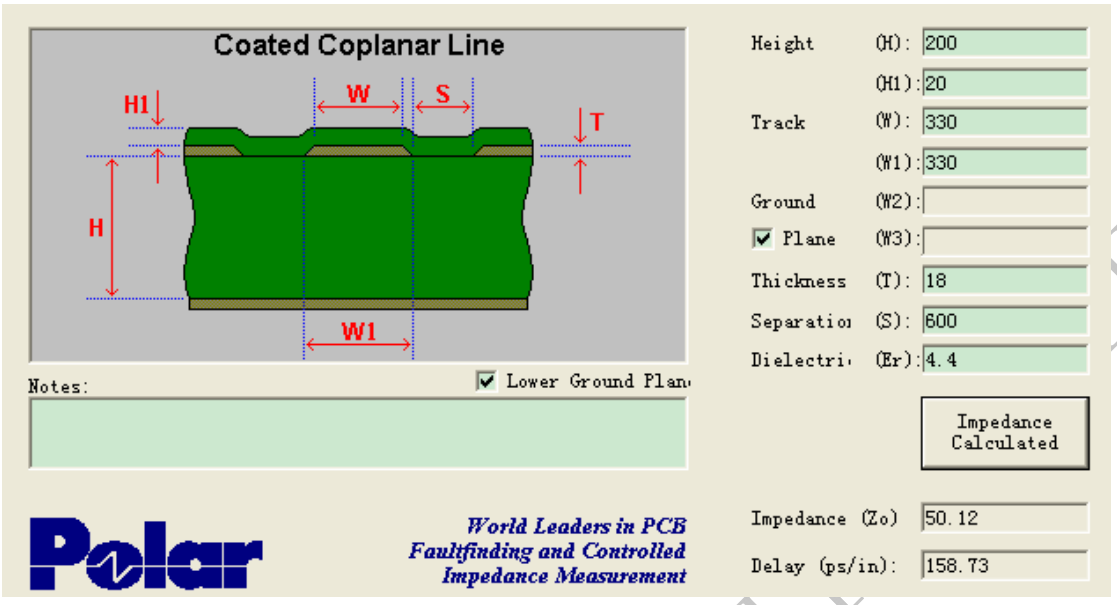


Figure 28：Calculating the RF trace impendence

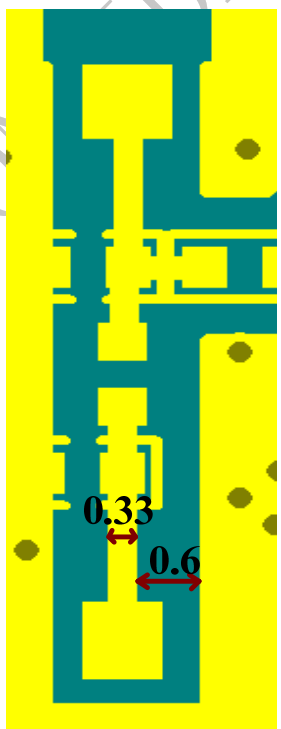


Figure 29：RF trace width and the separation from GND

To suit the physical design of individual applications SIM800H&L-TE offers two alternative approaches to connect the antenna:

- Mini RF coaxial connector. This antenna connector approaches is recommended, and the recommended

connector is shown as below table.

- SMA connectors. The specification was shown in the following figure:

Recommended mini RF coaxial connector:

Vendor	Part Number	Web Site
HRS	U.FL-R-SMT(10)	http://www.hirose-connectors.com
I-PEX	20279-001E-01	http://www.i-pex.com/cn

NOTE: Both connectors can only be applied alternatively. This means, if an antenna is plugged to the mini RF connector, the SMA connector must not be used; and if the antenna is connected to the SMA connector, then the mini RF connector must not be used.

GSM RF trace and the antenna connectors are shown as below:

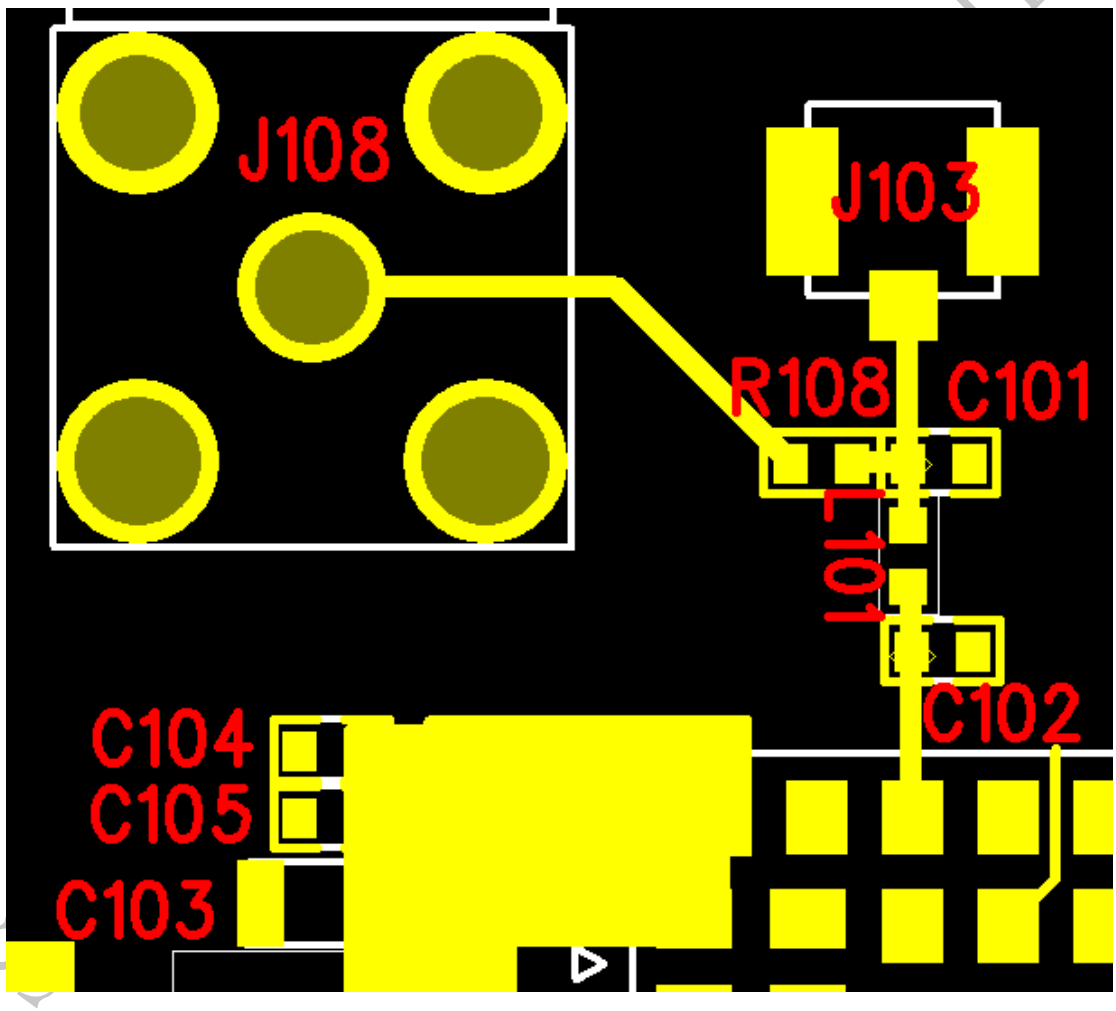


Figure 30：GSM RF trace and the antenna connectors

If the J103 is used to be GSM antenna connector, L101 should be mounted a 0Ω resistor and R108\C101\C102 not mounted. And if the J108 is used, R108 and L101 should be mounted 0Ω resistors and C101\C102 not mounted.

Bluetooth RF trace and the antenna connectors are shown as below:

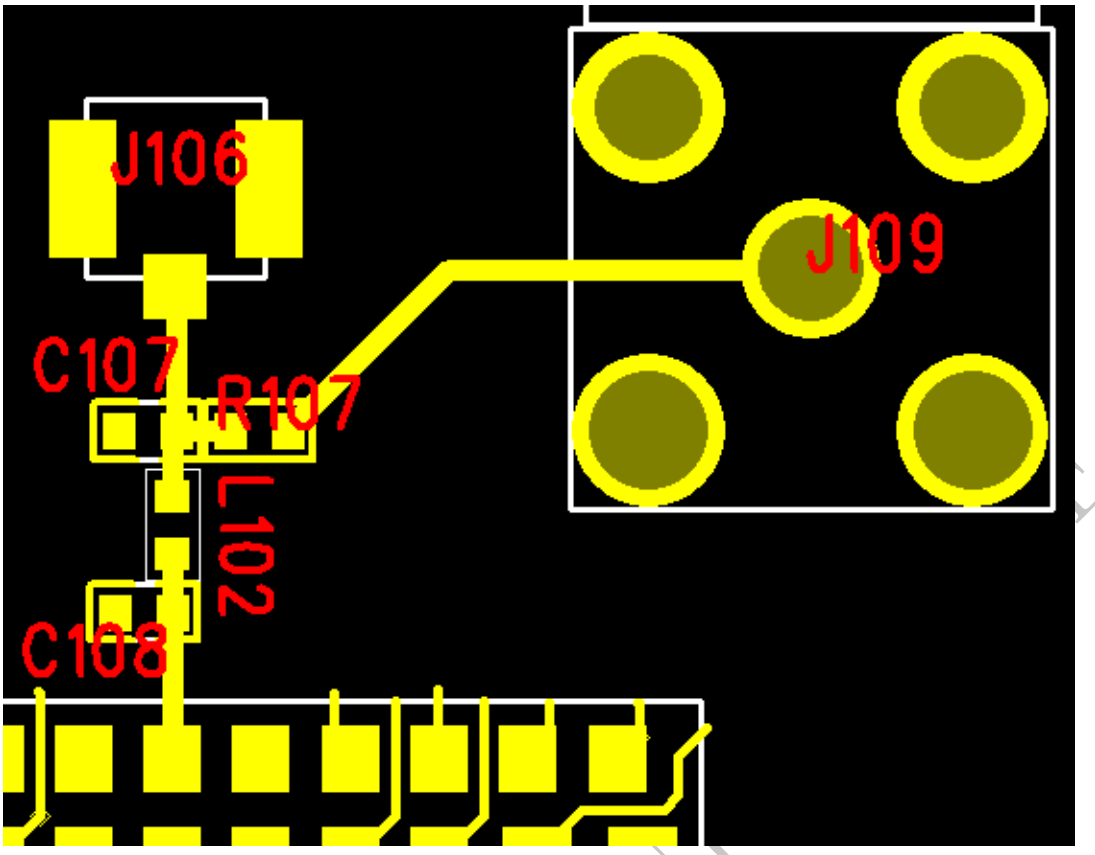


Figure 31：Bluetooth RF trace and the antenna connectors

If the J106 is used to be Bluetooth antenna connector, L102 should be mounted a 0Ω resistor and R107\C107\C108 not mounted. And if the J109 is used, R107 and L102 should be mounted 0Ω resistors and C107\C108 not mounted.

FM traces and the antenna connectors are shown as below:

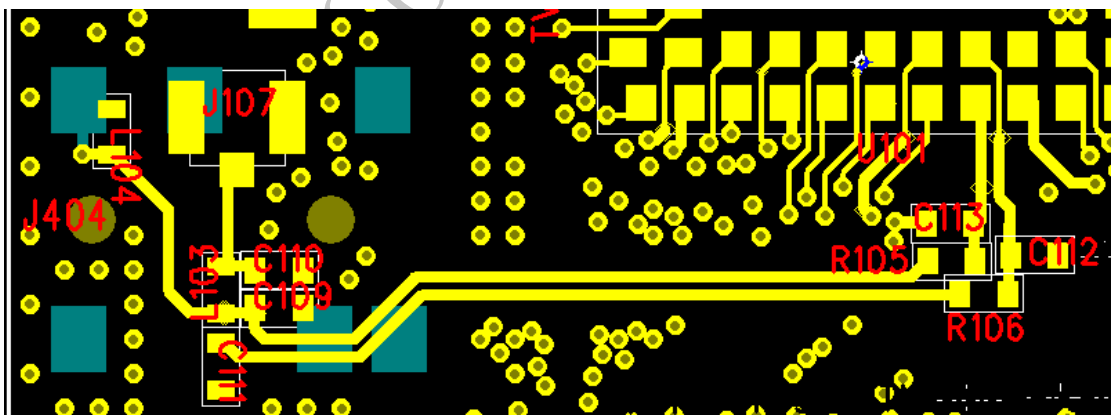


Figure 32：FM traces and the antenna connectors

J107 is the mini RF connector for FM antenna; J404 is the 3.5mm earphone jack. The GND pin of the J404 is used to be the FM antenna.

If the J107 is used to be FM antenna connector, L103\R105\R106 should be mounted 0Ω resistors and C113\C112\C110\C109\C111\L104 not mounted. And if the J404 is used, R105\R106 should be mounted 0Ω resistors and C113\C112\C110\C109\C111\L104\L103 not amounted.

Contact us:

Shanghai SIMCom Wireless Solutions Co.,Ltd.
Address: Building A, SIM Technology Building, No. 633, Jinzhong Road, Shanghai, P. R. China 200335
Tel: +86 21 3252 3300
Fax: +86 21 3252 3020
URL: www.sim.com/wm