M10 User Manual

Contents

Table Index

Figure Index

0. Revision history

Revision	Date	Author	Description of change
1.00		Ken Ji/ Samuel Hong	Origin

1. Introduction

This document gives recommendation for M10 module integration in a wireless application, such as vehicle tracking system, smart metering and PDA. It gives some recommendations for design notes, reference circuit and PCB layout.

The M10 is a Quad-band GSM/GPRS engine that works at frequencies of GSM850, and PCS1900. M10 features GPRS and supports the GPRS coding schemes CS-1, CS-2, CS-3 and CS-4.

With a tiny profile of 29mm ×29mm ×3.6 mm, the module can meet almost all the requirements for M2M applications, including Tracking and Tracing, Intelligent Instrument, Wireless POS, Security, Telematics, Remote Controlling, etc.

M10 is an SMD type module, which can be embedded in customer application through its 64-pin pads. It provides all hardware interfaces between the module and customer's host board.

1.1. Reference

2. Product Concept

The M10 is a Quad-band GSM/GPRS engine that works at frequency bands of GSM850 and PCS 1900. The M10 features GPRS multi-slot class 12 and supports the GPRS coding schemes CS-1, CS-2, CS-3 and CS-4.

The M10 is an SMD type module with 64-pin pads and a tiny profile of 29mm x 29mm x 3.6 mm (the thickness of PCB is 1.6mm), which can fit into almost all customers' applications. It provides all hardware interfaces between the module and customer' host board.

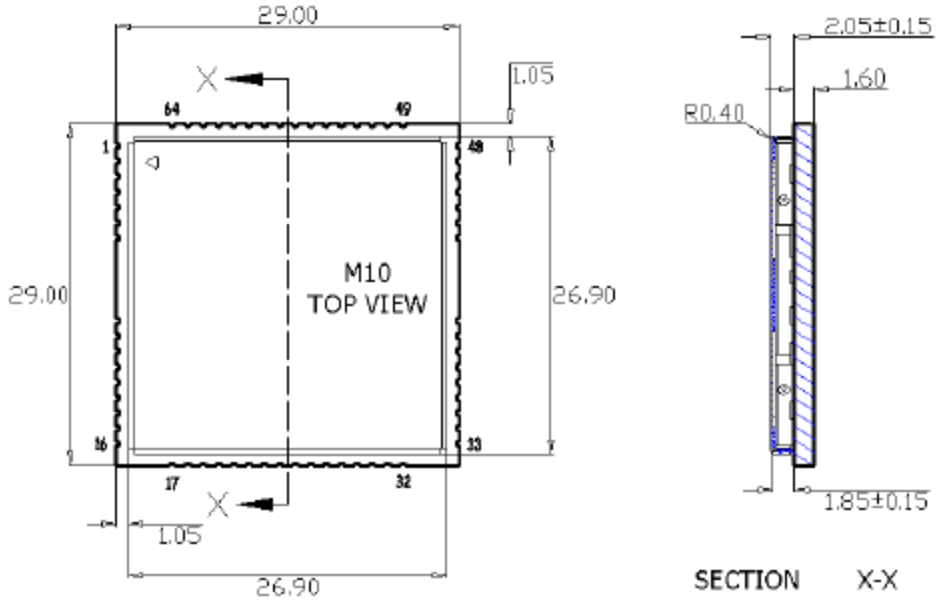
- External controller can communicate with M10 through its main UART port.
- Two audio channels include two microphone inputs and two speaker outputs, which can be easily configured by AT command.

The module is designed with power saving technique so that the current consumption could be very low.

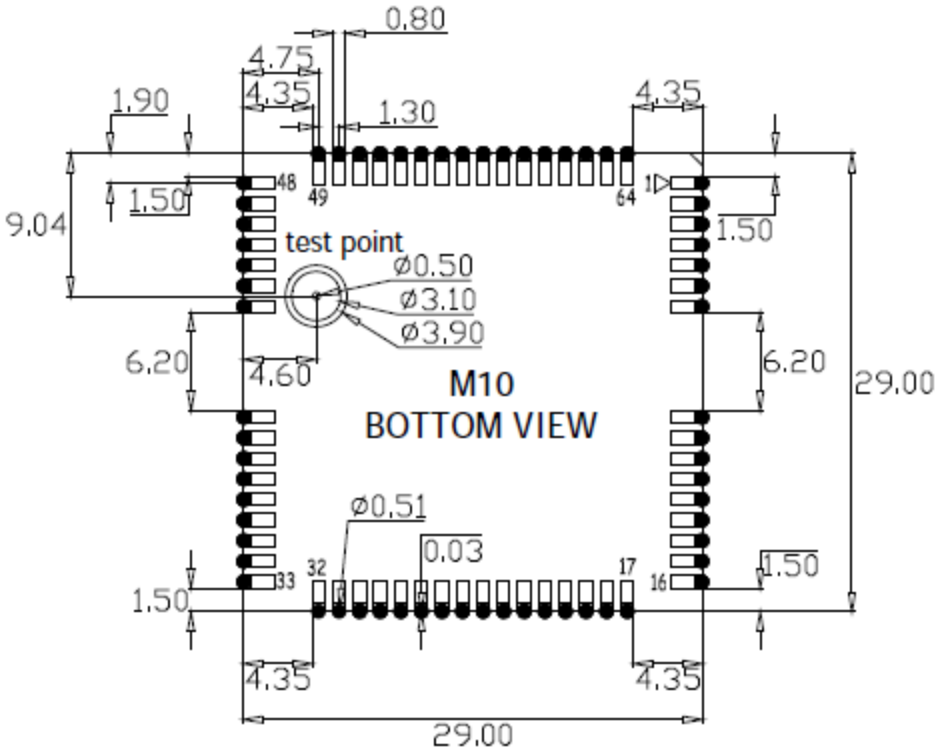
TCP/IP protocol stack has been integrated in the module. Moreover, extended TCP/IP AT commands have been developed for customer to use the internal TCP/IP protocol easily, which is very useful for data transfer application.

The module is fully RoHS compliant to EU regulation.

Mechanical dimensions of module



M10 top and side dimensions（Unit: mm）



M10 bottom dimensions（Unit: mm）

3. Placement

Please pay attention to the placement and the PCB layout in your application design.

3.1. Pin Assignment

The pin assignment of the M10 module is shown in Figure 1. Placement of module should be carefully considered to make the RF_IN pad as close as possible to antenna so as to reduce overall RF trace length. The longer the RF trace to antenna, the larger the RF insertion loss. In addition, please keep RF part and antenna from the system crystal and the audio part in host board as far as possible to reduce possible RF interference due to GSM transmission bursts from antenna and RF trace.

M10 module include the following major functional parts:

 The GSM baseband part

 Flash and SRAM

 The GSM radio frequency part

 The SMT pads interface

—LCD interface

—SIM card interface

—Audio interface

—Key-board interface

—UART interface

—Power supply

—RF interface

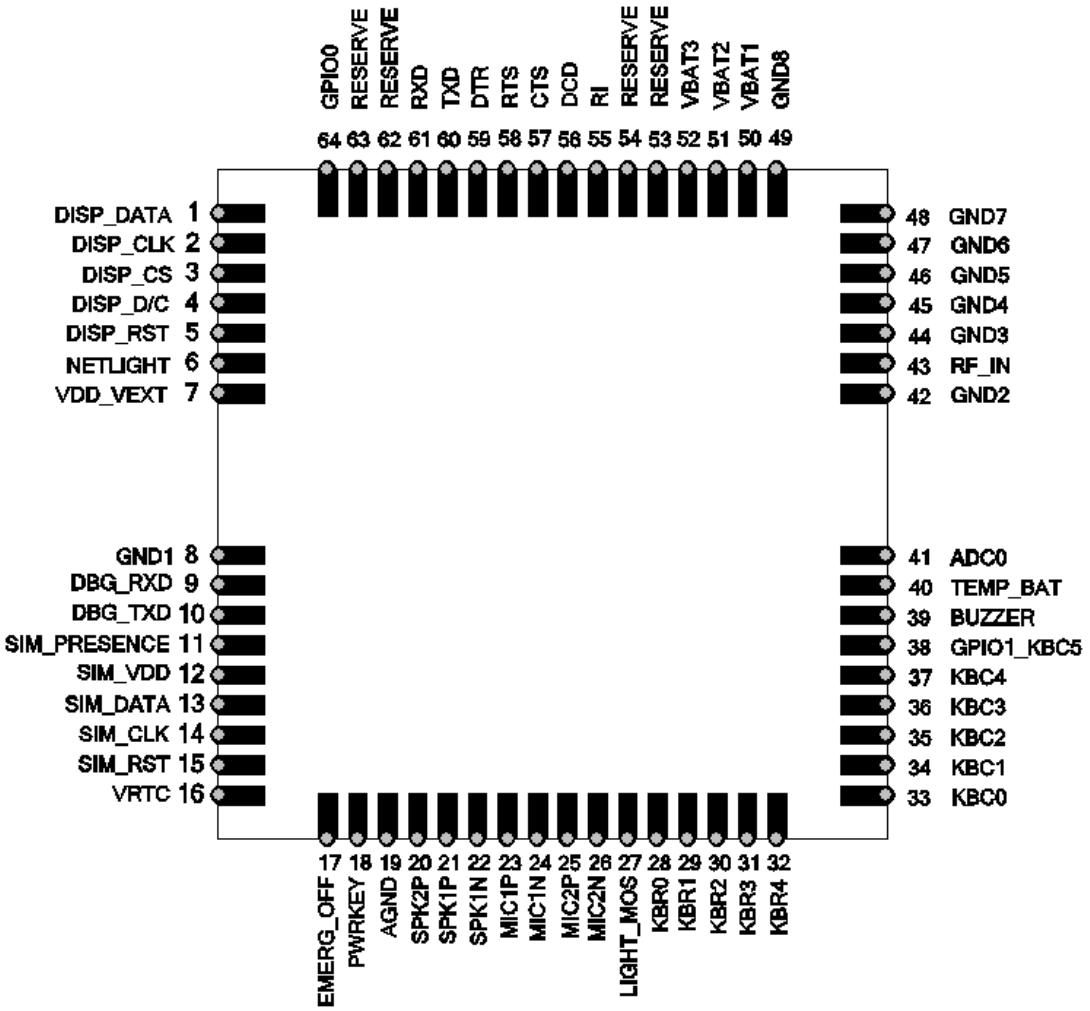


Figure 1: Pin assignment

PIN NO.	PIN NAME	I/O	PIN NO.	PIN NAME	I/O
1	DISP_DATA	I/O	2	DISP_CLK	O
3	DISP_CS	O	4	DISP_D/C	O
5	DISP_RST	O	6	NETLIGHT	O
7	VDD_EXT	O	8	GND1	
9	DBG_RXD	I	10	DBG_TXD	O
11	SIM_PRESENCE	I	12	SIM_VDD	O
13	SIM_DATA	I/O	14	SIM_CLK	O
15	SIM_RST	O	16	VRTC	I/O
17	EMERG_OFF	I	18	PWRKEY	I
19	AGND	O	20	SPK2P	O
21	SPK1N	O	22	SPK1P	O
23	MIC1P	I	24	MIC1N	I
25	MIC2P	I	26	MIC2N	I
27	LIGHT_MOS	O	28	KBR0	O
29	KBR1	O	30	KBR2	O
31	KBR3	O	32	KBR4	O
33	KBC0	I	34	KBC1	I
35	KBC2	I	36	KBC3	I
37	KBC4	I	38	GPIO1_KBC5	I/O
39	BUZZER	O	40	TEMP_BAT	I
41	ADC0	I	42	GND2	
43	RF_ANT	I/O	44	GND3	
45	GND4		46	GND5	
47	GND6		48	GND7	
49	GND8		50	VBAT1	I
51	VBAT2	I	52	VBAT3	I
53	VCHG(1)	I	54	STATUS	O
55	RI	O	56	DCD	O
57	CTS	O	58	RTS	I
59	DTR	I	60	TXD	O
61	RXD	I	62	TXD3	O
63	RXD3	I	64	GPIO0	I/O

Note: Please keep all reserved pins open.

(1): This function is not supported in the default hardware configuration.

Power supply				
PIN NAME	I/O	DESCRIPTION	DC CHARACTERISTICS	COMMENT
VBAT	I	Module main power supply. The power supply of module has to be a single voltage source of VBAT=3.4V~4.5V. It must be able to provide sufficient current in a transmitting burst which typically rises to 2A.	Vmax= 4.5V Vmin=3.4V Vnorm=4.0V	
VCHG	I	Voltage input for the charging circuit	Vmax=6.5V Vmin=1.1 * VBAT Vnorm=5.0V	If unused, keep this pin open. Charging function is not supported in default.
VRTC	I/O	Power supply for RTC when VBAT is not supplied for the system. Charging for backup battery or golden capacitor when the VBAT is supplied.	VImax=VBAT VImin=2.6V VInorm=2.75V VOmax=2.85V VOmin=2.6V VOnorm=2.75V Iout(max)= 730uA Iin=2.6~5 uA	Recommend to connect to a backup battery or a golden capacitor.
VDD_EXT	O	Supply 2.8V voltage for external circuit.	Vmax=2.9V Vmin=2.7V Vnorm=2.8V Imax=20mA	1. If unused, keep this pin open. 2. Recommend to add a 2.2~4.7uF bypass capacitor, when using this pin for power supply.
GND		Digital ground		
Power on or power off				
PIN NAME	I/O	DESCRIPTION	DC CHARACTERISTICS	COMMENT
PWRKEY	I	Power on/off key. PWRKEY should be pulled down for a moment to turn on or turn off the system.	VILmax=0.3*VBAT VIHmin=0.7*VBAT VImax=VBAT	Pull up to VBAT Internally.
Emergency shutdown				
PIN NAME	I/O	DESCRIPTION	DC CHARACTERISTICS	COMMENT
EMERG_ OFF	I	Emergency off. Pulled down for at least 20ms will turn off the module in case of emergency. Use it only when normal shutdown through PWRKEY or AT command can't perform well.	VILmax=0.4V VIHmin=2.2V V_{open}max=2.8V	Open drain/collector driver required in cellular device application. If unused, keep this pin open.

Module status indication				
PIN NAME	I/O	DESCRIPTION	DC CHARACTERISTICS	COMMENT
STATUS	O	Used to indicate module operating status. High level indicates module power-on and low level indicates power-down.	VOLmin=GND VOLmax=0.34V VOHmin=2.0V VOHmax= VDD_EXT	If unused, keep this pin open.
Audio interfaces				
PIN NAME	I/O	DESCRIPTION	DC CHARACTERISTICS	COMMENT
MIC1P MIC1N	I	Positive and negative voice-band input	Audio DC Characteristics refer to Chapter 3.10.	If unused, keep these pins open
MIC2P MIC2N	I	Auxiliary positive and negative voice-band input		
SPK1P SPK1N	O	Positive and negative voice-band output		If unused, keep these pins open
SPK2P	O	Auxiliary positive voice-band output		If unused, keep this pin open.
AGND		AGND is separate ground connection for external audio circuits.		If unused, keep this pin open.
BUZZER	O	Buzzer output		If unused, keep this pin open
General purpose input/output				
PIN NAME	I/O	DESCRIPTION	DC CHARACTERISTICS	COMMENT
KBC0~ KBC4	I	Keypad interface	VILmin=0V VILmax=0.67V VIHmin=1.7V VIHmax= VDD_EXT+0.3 VOLmin=GND VOLmax=0.34V VOHmin=2.0V VOHmax= VDD_EXT	If unused, keep these pins open
KBR0~ KBR4	O			Pull up to VDD_EXT, if unused, keep these pins open
DISP_DATA	I/O	LCD display interface		If unused, keep these pins open
DISP_CLK	O			
DISP_CS	O			
DISP_D/C	O			
DISP_RST	O			
NETLIGHT	O	Network status indication		If unused, keep these pins open
GPIO0	I/O	Normal input/output port		
GPIO1_ KBC5	I/O	Normal input/output port/Keypad interface		
LIGHT_ MOS	O	Open drain output port	Imax=100mA	If unused, keep this pin open

Serial port				
PIN NAME	I/O	DESCRIPTION	DC CHARACTERISTICS	COMMENT
DTR	I	Data terminal ready	VILmin=0V VILmax=0.67V VIHmin=1.7V VIHmax= VDD_EXT+0.3 VOLmin=GND VOLmax=0.34V VOHmin=2.0V VOHmax= VDD_EXT	If only use TXD, RXD and GND to communicate, recommend to keep other pins open.
RXD	I	Receiving data		
TXD	O	Transmitting data		
RTS	I	Request to send		
CTS	O	Clear to send		
RI	O	Ring indicator		
DCD	O	Data carrier detection		
Debug port				
DBG_TXD	O	Serial interface for debugging only.		If unused, keep these pins open
DBG_RXD	I			

Serial port 3				
TXD3	O	Transmitting data	VILmin=0V VILmax=0.67V VIHmin=1.7V VIHmax= VDD_EXT+0.3 VOLmin=GND VOLmax=0.34V VOHmin=2.0V VOHmax= VDD_EXT	If unused, keep these pins open.
RXD3	I	Receiving data		

SIM interface				
PIN NAME	I/O	DESCRIPTION	DC CHARACTERISTICS	COMMENT
SIM_VDD	O	Voltage supply for SIM card	The voltage can be selected by software automatically. Either 1.8V or 3V.	All signals of SIM interface should be protected against ESD with a TVS diode array. Maximum cable length 200mm from the module pad to SIM card holder.
SIM_DATA	I/O	SIM data	VIHmin=0.7*SIM_VDD VOHmin=0.8*SIM_VDD VOLmax=0.4V When SIM_VDD=3V VILmax=0.4V When SIM_VDD=1.8V VILmax=0.2* SIM_VDD VOHmin=0.9*SIM_VDD When SIM_VDD=3V VOLmax=0.4V When SIM_VDD=1.8V VOLmax=0.2* SIM_VDD	
SIM_CLK	O	SIM clock		
SIM_RST	O	SIM reset		
SIM_PRESENCE	I	SIM card detection	VILmax=0.67V VIHmin=1.7V	If unused, keep this pin open.

AUXADC				
PIN NAME	I/O	DESCRIPTION	DC CHARACTERISTICS	COMMENT
ADC0	I	General purpose analog to digital converter.	voltage range: 0V to 2.8V	If unused, keep this pin open
TEMP_BAT	I	ADC input for battery temperature over NTC resistor.	voltage range: 0V to 2.8V	NTC should be installed inside or near battery pack to deliver temperature values. If unused keep this pin open.

RF interface				
PIN NAME	I/O	DESCRIPTION	DC CHARACTERISTICS	COMMENT
RF_ANT	I/O	RF antenna pad	impedance of 50Ω	

3.2. Placement recommendation

The analog part components such as microphone should be placed far away from antenna and power supply. General placement recommendation is shown in Figure 2.

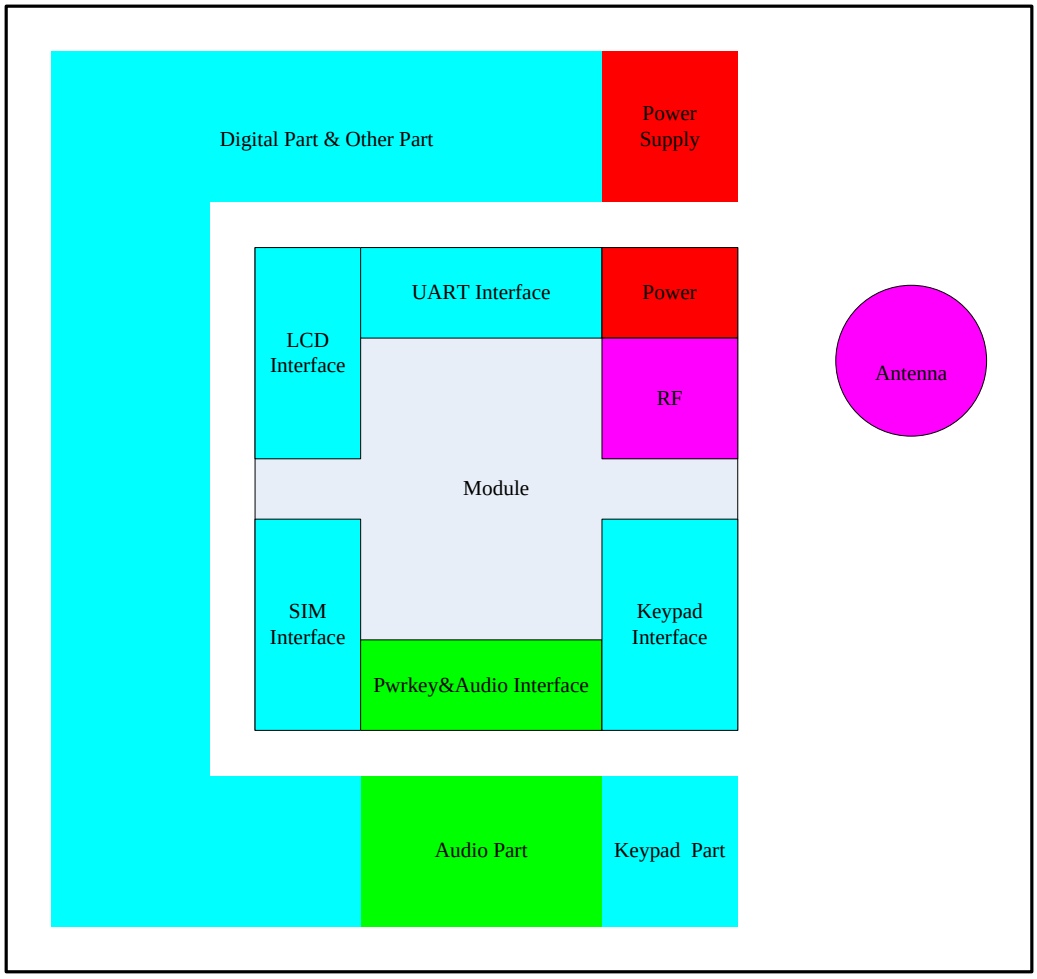


Figure 2: Recommendation of placement

3.3. Placement clearance

The module mounts with 64 SMT pads. For easy maintenance of this module and accessing to these pads, please keep a distance no less than 3mm between M10 and other components.

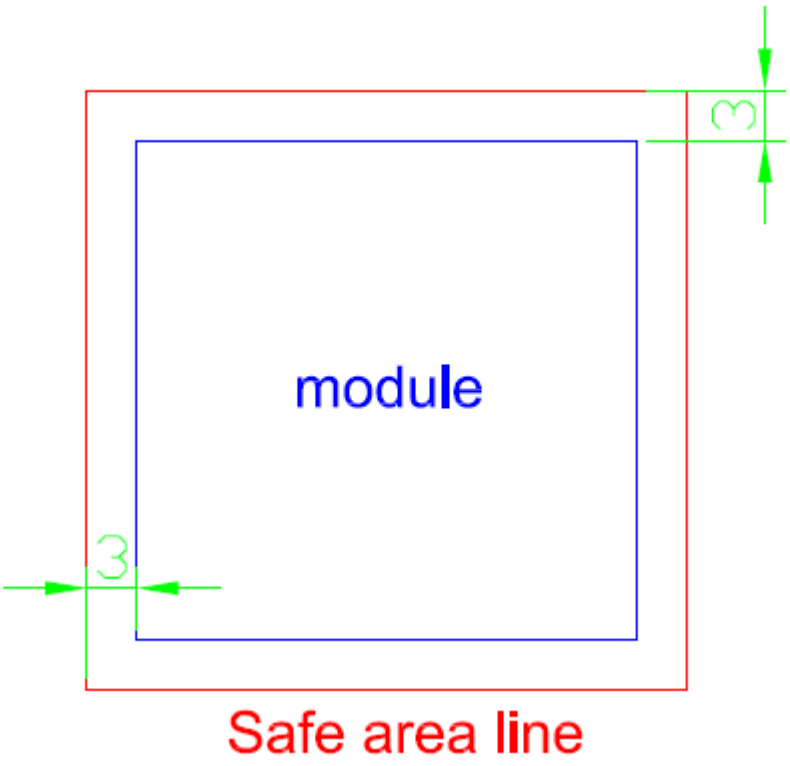


Figure 3: Placement clearance

4. Digital I/O Connection

If the voltage level of peripheral interface circuit does not match module interface, the power consumption of the system could increase, and could even cause the module damaged.

- Each digital I/O of the module operates in a 2.8V logic level inside the module. The voltage level of those digital interfaces connected to the module should match the electrical characteristics of the module listed in Table 2. Otherwise, a level shifter circuit must be inserted between the host and the module.

Table 1: Digital I/O electrical characteristics

SYMBOL	MIN	MAX	UNITS
V_{IL}	0	0.67	V
V_{IH}	1.7	3.1	V
V_{OL}	0	0.34	V
V_{OH}	2.0	2.8	V

- For direct connection between I/Os, please pay attention to I/Os' input or output configuration. If the I/O direction configuration conflicts with each other, the power consumption could increase, and the module could be very hot, and even be damaged. For example, it is forbidden that user's I/O outputs a low level while module's connected I/O outputs a high level.

5. VDD_EXT Pin

This pin is a power supply from a regulator inside the module which can supply current of about 20mA. Customer can also use this pin to judge whether the module is off or not. When the module is turned off, the VDD_EXT pin will change from high level to low level.

6. Serial Interface and Debug Interface

The TXD and RXD pins should be connected to host MCU. The DTR pin should be controlled to trigger SLEEP mode or wakeup the module. The RTS and CTS pins should be connected to the host MCU if hardware flow control is required.

The TXD, RXD, PWRKEY and GND pins can also be used for software upgrade and high-level acoustic parameters configuration. The DBG_TXD and DBG_RXD pins are only used for software debug. Please note that the PWRKEY pin should be pulled to low level when the M10 is being upgraded. For more detailed information on serial port design, please refer to document [2]

Notes: It's recommended to connect the pins necessary for firmware upgrade to external interface.

7. SIM Card

As shown in Figure 4, connecting a large volume capacitor such as 10uF in the SIM_VDD line could lead to failure of detecting the SIM card. A capacitor between 100nF and 1uF is recommended.

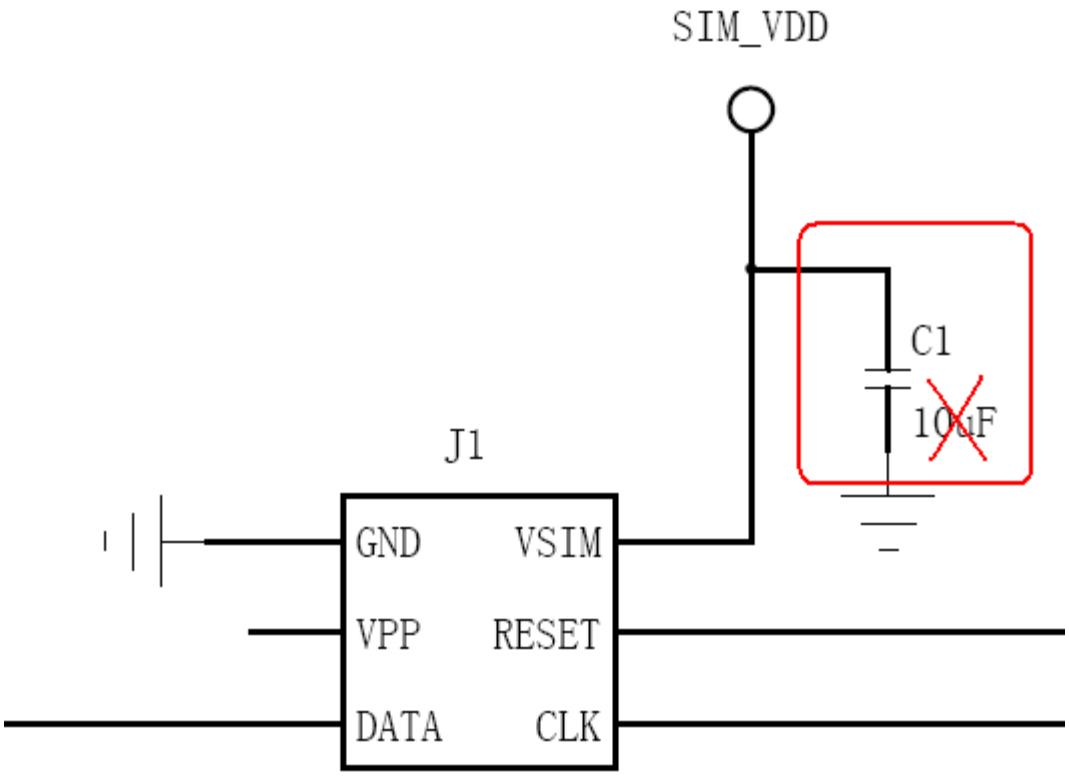


Figure 4: Circuit of the SIM card

8. SLEEP Mode

The command AT+CSCLK can enable or disable SLEEP mode. When the SLEEP mode is enabled, pulling the DTR pin to high level would drive the module into SLEEP mode; and pulling the DTR pin to low level the module would exit from SLEEP mode.

9. Audio Trace

If possible, the audio trace should be placed in inner layer, and shielded by ground in the same layer and the upper and lower adjacent layers to prevent from RF interference. In addition, it is recommended to add as many via as possible between ground layers so as to reduce RF noise.

The AGND signal is usually used for AOUT2 channel to establish a single-end output with SPK2P. Do not pair GND with SPK2P to establish an audio output, otherwise TDD (Time Division Duplex) noise from power supply could occur in AOUT2.

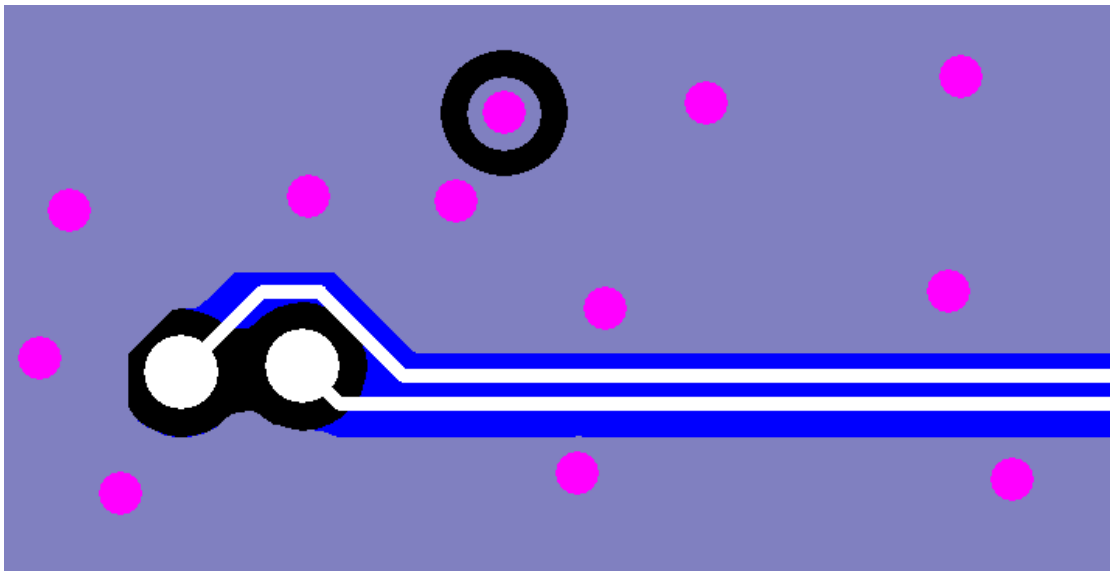


Figure 5: Audio trace routing example

10. RF Design Guide

Correct RF design is essential for RF performance such as transmitting power, receiving sensitivity and harmonics. Following this RF design guide could benefit to improve the RF performance of customer's product.

10.1. Recommended Impedance Matching Circuit

The impedance of M10's RF_ANT port is 50Ω. If the impedance of antenna is close to 50Ω in all working frequency bands, the antenna could be connected to the RF_ANT port directly via 50Ω transmission line. But if the impedance of antenna is not close to 50Ω, a T-type or π-type matching circuit should be inserted between transmission line and antenna. The matching components should be placed as close as possible to the antenna's feed point.

Figure 6 and Figure 7 show the reference designs of T-type and π-type matching circuits.

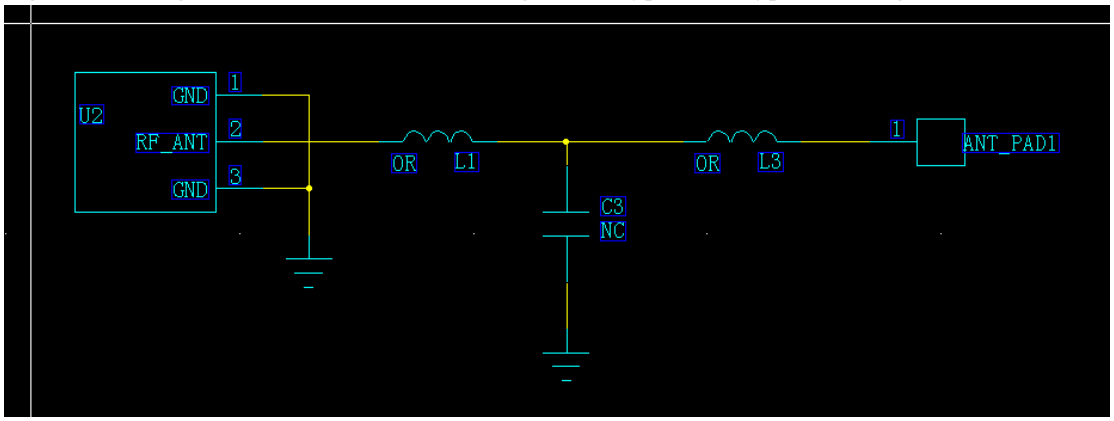


Figure 6: T-type matching circuit

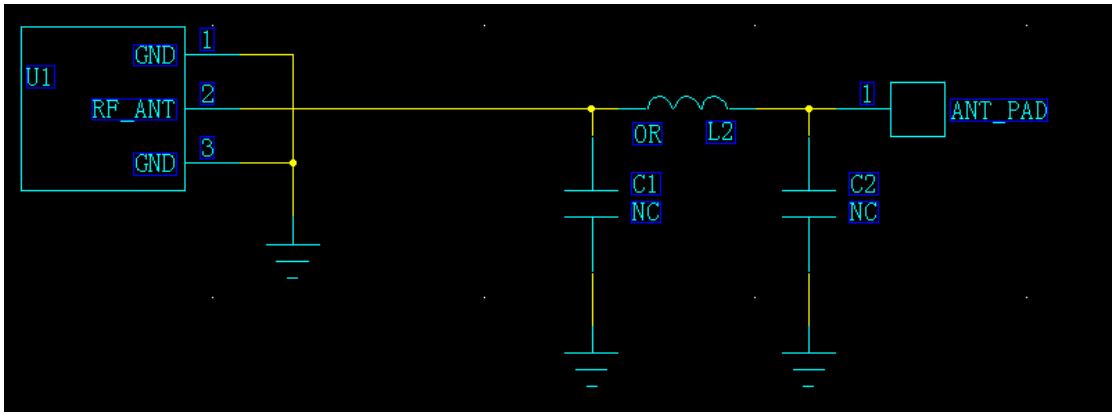


Figure 7: π-type matching circuit

NOTE: The impedance of traces in Bold type must be 50Ω.

10.2. Matched RF Transmission Line Design

In PCB layout, a matched RF transmission line has a fixed characteristic impedance, which is called Z_0, from its source to its load. The source should have an internal resistance of Z_0 and the resistance of matching load should close to Z_0.

Since the impedance of M10's RF_ANT port is 50Ω, the impedance of the RF transmission line from this port to the antenna or the matching circuit should also be made to 50Ω.

More than twelve different types of transmission line can be created on a PCB simply by controlling trace geometry, and some of them are shown in Figure 8.

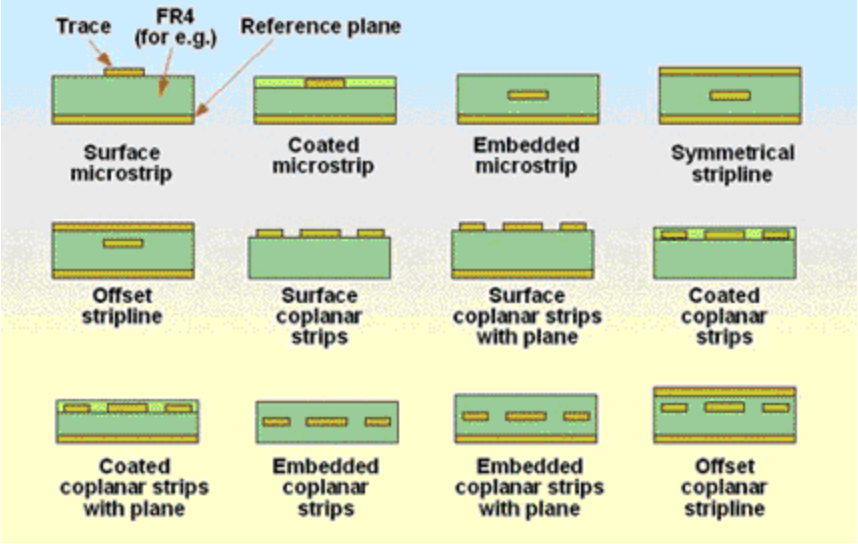


Figure 8: M10 RF_ANT PCB layout

Customer may adopt one or certain types of them to design RF trace. Upon the demand of application design, the number of PCB layer can be different such as two and four. Each type of PCB has corresponding "stack-up". The "stack-up" is the name given to the order of the various etched copper foil and dielectric layers that are laminated together under pressure and heat to make a PCB.

10.3. PCB Layout Consideration

PCB Layout is essential to the performance of customer's product. Here are some rules that should be followed:

- Impedance control

Control the impedance of RF trace as close as possible to 50Ω. If the thickness between RF_ANT pad and the ground layer is less than 0.4mm, it could significantly decrease the output power. Therefore, when they are too close, we strongly suggest removing the copper in the layer beneath the RF_ANT pad. If RF trace routes to another layer, add GND via along with it to keep GND integral. The clearance between RF trace and ground plane in same layer should be at least twice the RF trace width.

- Make RF trace as short as possible
  Place the module and the matching circuit near the antenna pad. Shorten the length of RF trace. Place the antenna PAD in the corner or at the edge of host board.

- Protect RF trace
  Avoid placing noise generating traces such as digital signal or clock line near RF trace in the same layer. Carefully route other traces in the layers adjacent to the RF trace, remember not to route in parallel with the RF trace. If possible, keep those traces far away from the RF trace.

- An RF test point is located at the bottom side of M10 for manufacture purpose. The copper which is close to this test point in the top layer of customer's host board must be kept out or removed. No signal trace should be placed in the top layer and the second layer beneath this test point.

Insert a picture.

- Customer can use either antenna PAD or RF connector to connect the antenna.
  If antenna PAD is adopted, Figure 9 is a reference design for a four-layer PCB. Make the space on all layers beneath antenna pad keep-out. Place a ground PAD near the antenna PAD. The distance between GND PAD and antenna PAD can be around 1.8mm. The size of antenna PAD can be 1.8mm*1.8mm, and the GND PAD should be a little bigger, e.g. 2.5mm*2.5mm. Add several GND via near or on the GND PAD to reduce impedance from the GND PAD to the RF reference ground.

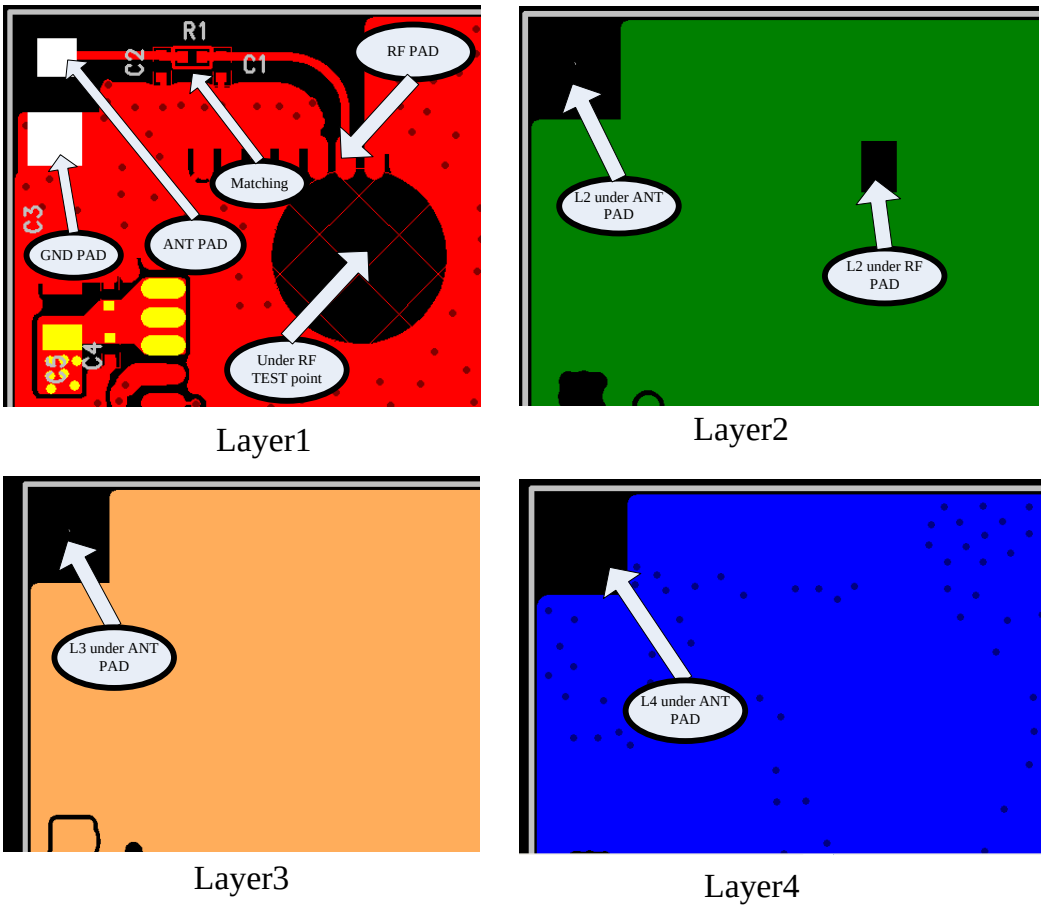


Figure 9: Reference PCB design with antenna pad in a four-layer PCB

If RF connector is adopted, place the RF connector close to module RF_ANT, and add several ground via close to the GND PAD of RF connector. Figure 10 is the reference PCB design.

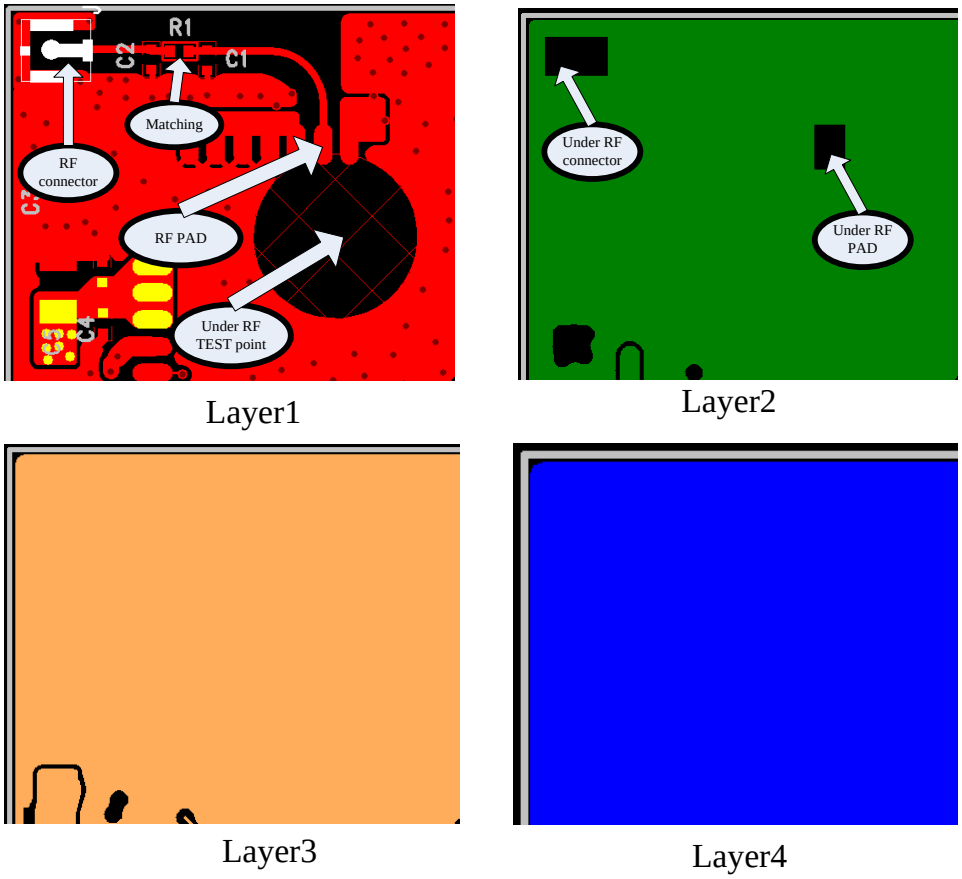


Figure 10: Reference PCB design with RF connector in a four-layer PCB

The stack-up of the four-layer PCB is shown in Figure 11.

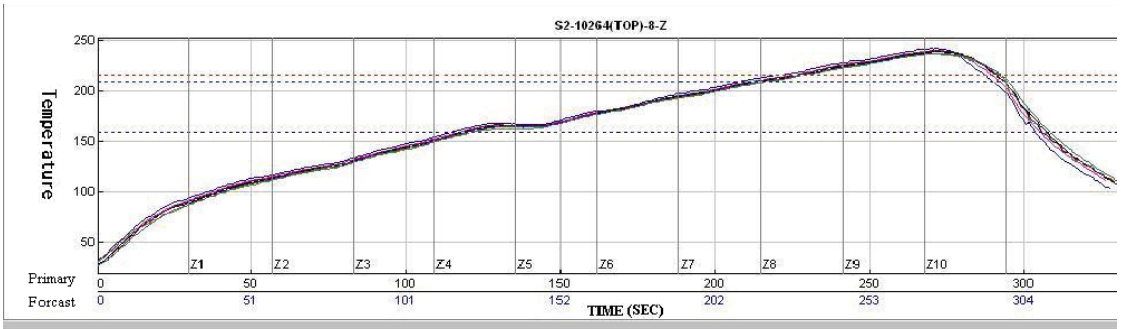
MATERIAL BUILD-UP

	Solder Mask	18.0 µm.	
	Chemical Gold	0.05 µm.	
	Electroless Nickel	2.54 µm.	
	Copper Plating	13 µm.	
LAYER1	Copper	12.0 µm.	
	1080LDP	200.0 µm	standard via 1-4.
LAYER2	Copper	18.0 µm.	
	Prepreg 2116	500.0 µm.	
LAYER3	Copper	18.0 µm.	
	1080LDP	200.0 µm.	
LAYER4	Copper	12.0 µm.	
	Copper Plating	13 µm.	
	Electroless Nickel	2.54 µm.	
	Chemical Gold	0.05 µm.	
	Solder Mask	18.0 µm.	

TOTAL OVERALL THICKNESS **1.0 mm +0.1/-0.1mm**

Figure 11: Stack-up of the four-layer PCB

11. The Recommended Ramp-soak-spike Reflow Profile



TCs	RAMP		SOAP between 160 to 210C		REFLOW /217C		PEAK TEMPERATURE	
2	1.3	-12%	94.4	15%	63.1	-48%	241.2	12%
3	1.3	-11%	93.2	11%	64.4	-42%	243.0	30%
4	1.3	-14%	94.6	15%	62.2	-51%	237.7	-23%
5	1.3	-12%	94.9	16%	62.4	-50%	240.0	0%
6	1.3	-11%	94.2	14%	64.3	-43%	240.8	8%
7	1.3	-13%	94.9	16%	63.6	-46%	238.3	-17%
Difference in	0.05		1.71		2.21		5.34	
P.2	1.3	-12%	94.4	15%	63.1	-48%	241.2	12%

Figure 12: The recommended ramp-soak-spike reflow p rofile

FCC Statement

Any Changes or modifications not expressly approved by the party responsible for compliance could void the user's authority to operate the equipment.

FCC RF exposure statement:

This equipment complies with FCC radiation exposure limits set forth for an uncontrolled environment. This equipment should be installed and operated with minimum distance 20cm between the radiator& your body.This transmitter must not be co-located or operating in conjunction with any other antenna or transmitter.

FCC Label Instructions

The outside of final products that contains this module device must display a label referring to the enclosed module. This exterior label can use wording such as: "Contains Transmitter Module FCC ID: YQD-M10" or "Contains FCC ID: YQD-M10" Any similar wording that expresses the same meaning may be used.