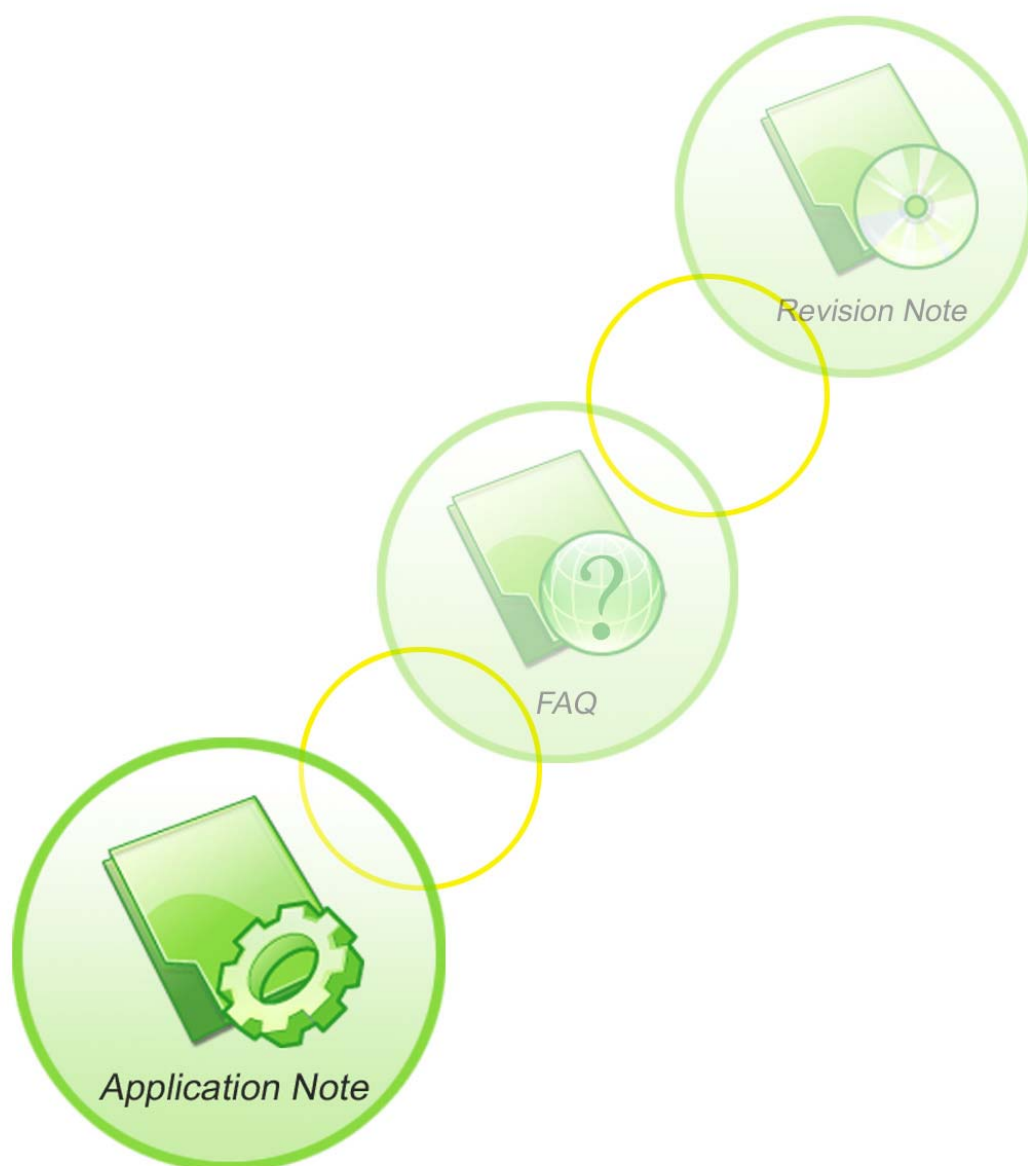




# SIM900 Reference Design Guide

**SIM900\_AN\_ Reference Design Guide\_V1.01**



<b>Document Title:</b>	SIM900 Reference Design Guide
<b>Version:</b>	1.01
<b>Date:</b>	2010-01-27
<b>Status:</b>	Released
<b>Document Control ID:</b>	SIM900_AN_Reference Design Guide_V1.01

### 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

This document contains proprietary technical information which is the property of SIMCom Limited., copying of this document and giving it to others and the using or communication of the contents thereof, are forbidden without express authority. Offenders are liable to the payment of damages. All rights reserved in the event of grant of a patent or the registration of a utility model or design. All specification supplied herein are subject to change without notice at any time.

***Copyright © Shanghai SIMCom Wireless Solutions Ltd. 2010***

## Contents

Version history .....	4
1. Introduction.....	5
1.1 References Document .....	5
2 Application Interface.....	2
2.1 Power Supply Design .....	2
2.1.1 VBAT design .....	2
2.1.2 Battery Charge design .....	3
2.1.3 PWRKEY ON/OFF design.....	4
2.1.4 Reset design.....	5
2.1.5 RTC backup design .....	5
2.2. UART interface .....	5
2.2.1 UART interface design .....	5
2.3 GPIO selection .....	6
2.3.1 STATUS and NETLIGHT .....	6
2.3.2 GPIO.....	7
2.4 Audio interfaces .....	7
2.4.1 Referenced circuit of headset design .....	8
2.4.2 Referenced circuit of hand free design .....	8
2.5 Antenna matching circuit design .....	8
2.6 SIM card interface.....	9
2.7 ESD protection .....	10
2.8 Consideration in PCB layout.....	10

## Version history

Date	Version	Description of change	Author
2010-01-01	1.01	First Release	Zhou Qiang

## 1 Introduction

This document describes how to design peripheral interface of SIMCOM module providing in your application.

### 1.1 References Document

SN	Document name	Remark
[1]	SIM900_ATC	SIM900 AT command Set
[2]	SIM900_HD	SIM900 Hardware Specification
[3]	SIM900_EVB_UGD	SIM900 EVB User Guide
[4]	SMT MODULE RF Design Guide	SMT MODULE RF Design Guide

## 2 Application Interface

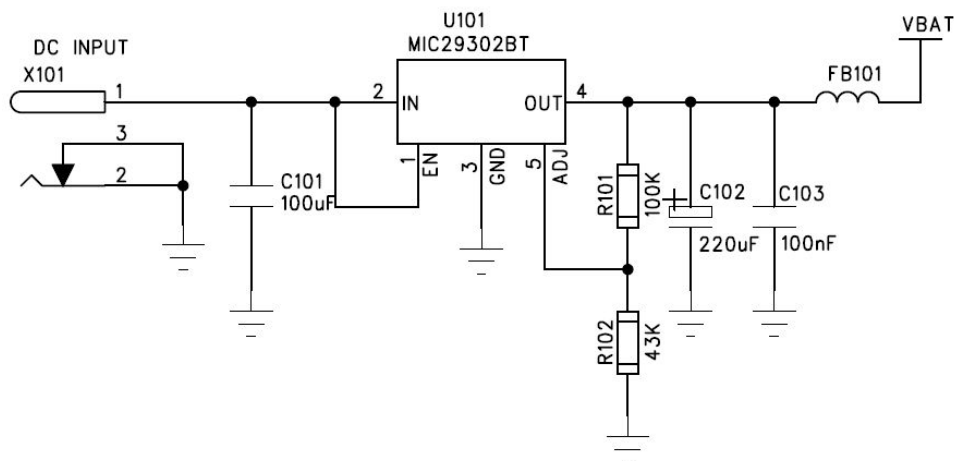
### 2.1 Power Supply Design

#### 2.1.1 VBAT design

The power supply of SIM900 is from a single voltage source of VBAT which normal operating range is from 3.4V to 4.5V. The peak working current can rise up to 2A in maximum power transmitting period, which will cause a voltage drop. So the power supply must be able to provide sufficient peak current, if not, the voltage may drop lower than 3.4V, and the module will auto power down. Typically, VBAT can be set to 4V.

SIM900 can be used in a wide range of application, the power supply design is deeply depending on the power source.

When the input is a 5V/2A adapter, a LDO linear regulator can be used in the design because the drop out between input and output is not so big. Figure 1 is the recommended circuit with MIC29302. Please also pay attention to the heat dissipation of the LDO. Usually, pouring a copper plane on the PCB is an effective way to the heat sink problem of the power IC.



**Figure 1: LDO Power Supply**

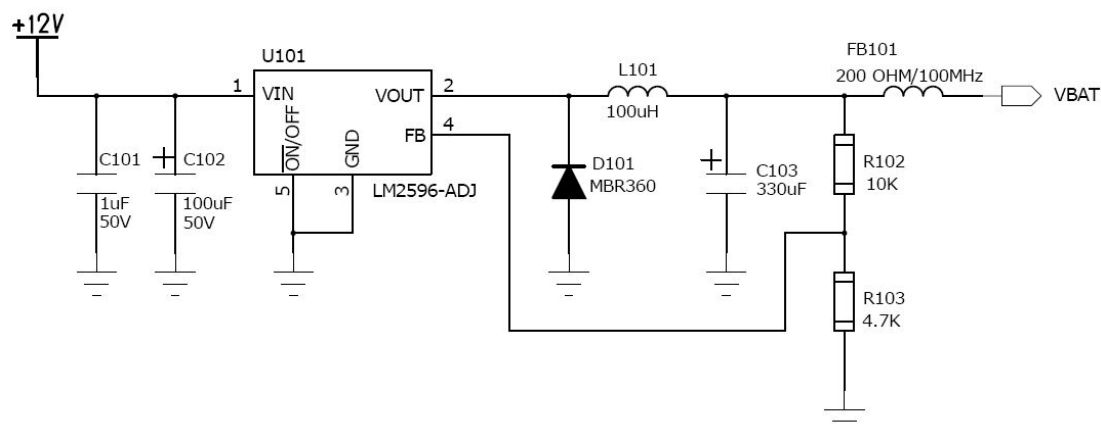
#### NOTE 1:

The bypass capacitor C102 in Figure 1 is selected strongly depending on the rated current of the power source and the power IC. If both of the rated current are 2A, a low ESR tantalum capacitor (220uF or smaller) close to the VBAT pin is enough. Or you should change C102 to a big value according to the practical application. The rated current should not be less than 800mA (5V output).

The power IC and the bypass capacitor should be placed near the module, and the VBAT trace should be routed as wide and short as possible to reduce the PCB copper resistance when layout. When the input voltage is 12V or more high, a DC to DC converter is the best choice as its high

efficiency.

When the module is powered by a DC-DC converter, then should pay more attention to the switching noise suppression design, otherwise the RF performance of SIM900 will be interfered by the switching noise of DC-DC, and cause some RF performance degraded, for example, modulation spectrum, switching spectrum will exceed the limit. As a solution, a large current ferrite bead FB101 (0805 size package, rated current > 2A, low DC resistance ) can be added between the DC-DC output and the VBAT in series. By default, FB101 can be mounted with a 0ohm resistor, and when need, it can be instead with a large current ferrite bead. For a typical application in automotive, following Figure 2 shows a reference circuit.

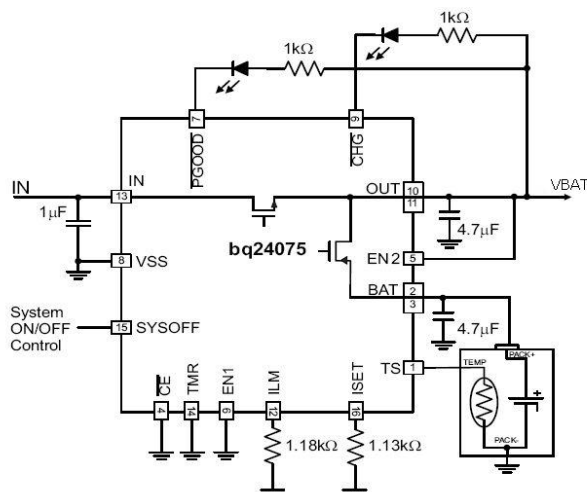


**Figure 2: DC-DC Power Supply**

When the input is from the USB port of a computer, the average current is 500mA according to the USB specification, so a super capacitor must be added near the module VBAT pins to compensate the peak current in transmit burst. A low ESR tantalum capacitor is usually used. The value for the capacitor should be not less than 470uF.

### 2.1.2 Battery Charge design

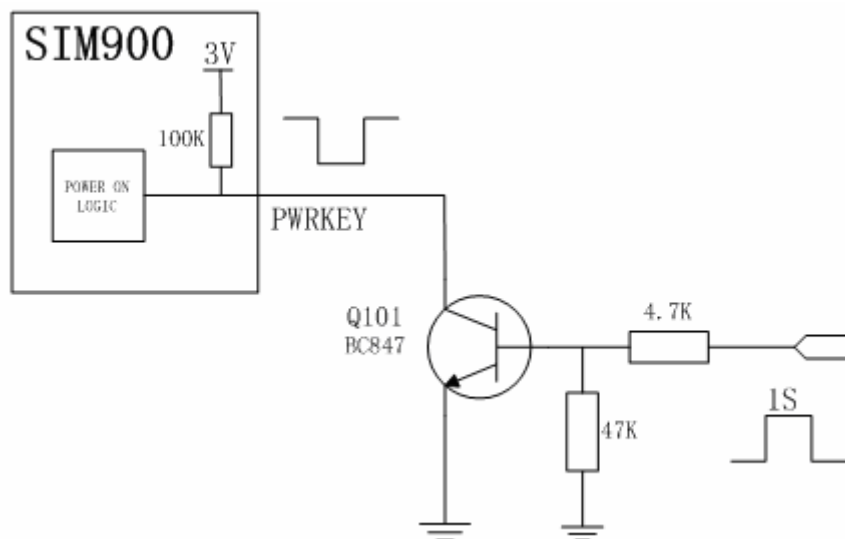
SIM900 do not support the battery charge function. When the Li-ion battery is needed in the application, a charger IC will be implemented. If the battery is only for backup (the AC adaptor is mostly powered), we suggest you choose a charger IC with power-path management for battery lifespan. For example: BQ2407x series from Texas Instruments. Following figure 3 is the reference circuit using BQ24075.



**Figure 3: Battery Charge Supply**

### 2.1.3 PWRKEY ON/OFF design

The simplest way to turn on/off SIM900 is driving the PWRKEY to a low level for 1S then release. PWRKEY pin has been pulled up to 3V inside the module. Following figure is the recommended connection with a NPN transistor. You can choose a GPIO of your MCU to control the POWER ON/OFF. Please note that don't add a capacitor on PWRKEY pin, or it may cause some unexpected problems when power on/off process.



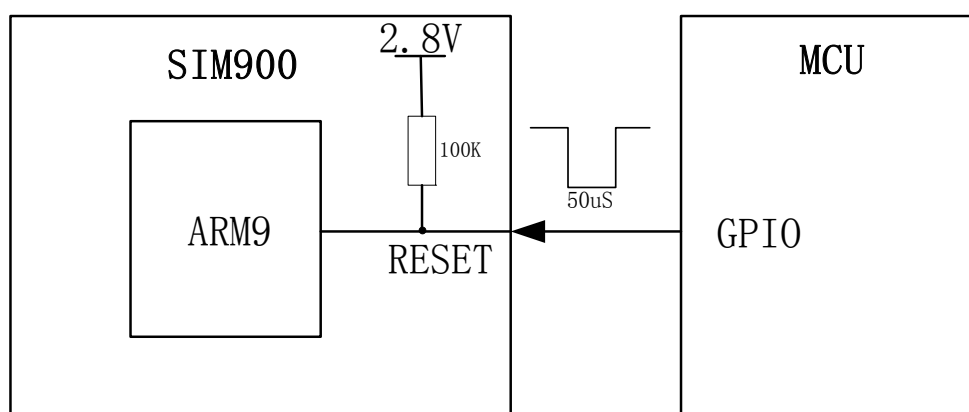
**Figure 4: Turn on/off module using transistor**

The other way to power on/off SIM900 is connecting PWRKEY and PWRKEY\_OUT together. In this way, a P-channel MOSFET is needed. The first way is recommended. In this way, PWRKEY\_OUT should be kept open.



## 2.1.4 Reset design

SIM900 support the reset function, when the MCU find the module is in an abnormal state, SIM900 can be restarted by pulling the RESET pin to ground for a typical 50uS. Reset is a noise sensitivity pin, it should be kept away from the high speed signal line (eg. clock) when layout. This pin is internal pull up to 2.8V through a 100k resistor. So it's not necessary to add an external pull up resistor for stable consideration if it is not used. Please pay attention that reset is only used in emergency situation, such as software break down, module does not response the AT command. Reset the module continually is not recommended and it may cause some unexpected fault.



**Figure 5: Reset SIM900 with a MCU**

## 2.1.5 RTC backup design

SIM900 integrates a RTC backup interface which can be connected to a backup battery or a capacitor. Please pay attention to the rated voltage of the battery should be 3V. The RTC current consumption is about 2uA when the VBAT is removed.

## 2.2. UART interface

### 2.2.1 UART interface design

SIM900 integrates two UART port, one is Serial Port, and the other is Debug Port. Serial port is for AT command with the MCU while Debug port is for firmware update and bug trace. It is suggested to connect Debug port to an external connector for module debug consideration. If hardware flow control is not used, DCD, DSR can be left floating. Please refer to the following figure. DTR can be used to wake up the module from sleep mode and RI can be used to detect a coming call or SMS. These two should connect to GPIO of the MCU.

#### NOTE 3:

Please note that the UART level is 2.8V, if the level is not matching, a level shift circuit is needed.

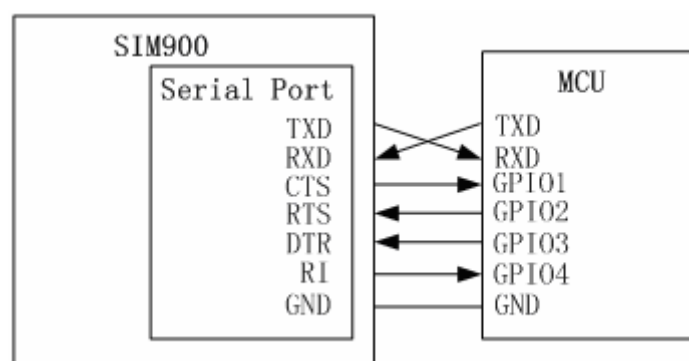


Figure 6: Serial Port Connection

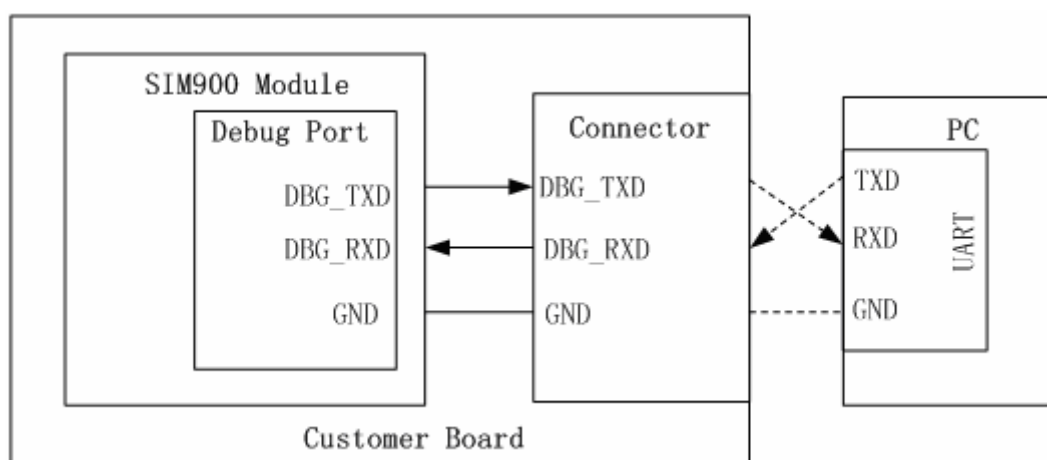


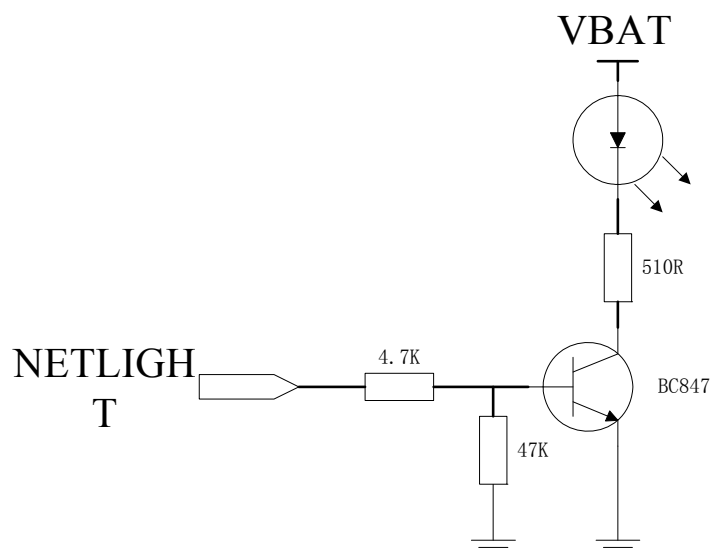
Figure 7: Debug Port Connection

## 2.3 GPIO selection

### 2.3.1 STATUS and NETLIGHT

STATUS pin can be used to monitor the module state during the power on/off process. After power up, AT command will response till the STATUS change high, and it will change low after the module log off from the base station in a power down procedure. It can be connected to a GPIO of the MCU.

NETLIGHT is a net status indicator. It can drive a transistor to control a LED which will blink slowly or quickly according to different states. Please note that it can't drive a LED directly. Both STATUS and NETLIGHT is dedicated in SIM900, they can not be used as a GPIO for the customer.



**Figure 8: NETLIGHT indicator**

### 2.3.2 GPIO

SIM900 serves 12 GPIO which can be controlled by the customer. As a GPI, the status can be read by the AT command and as a GPO, it also can be controlled by the AT command. The customer can use this GPIO to achieve some simple control. For more detail, please refer to Document[1].

### 2.4 Audio interfaces

SIM900 provides a pair of differential analogy audio channel, MICP & MICN can be connected to an electret condenser microphone directly and SPKP & SPKN can drive a 32 Ohm speaker. The maximum output power is 96mW. If the speaker is 8 Ohm, you should add an audio amplifier between the module output and the speaker. We recommend National Semiconductor's LM4890. Also, the microphone input channel can be configured to a single ended mode. In this mode, the MICN can be left open. The negative terminal of the microphone can be connected to GND in the customer's board.

For adjustment, you can use AT+CMIC to set the input gain level of microphone, AT+CLVL to change the volume of the speaker. For details, please refer to [2].

### 2.4.1 Referenced circuit of headset design

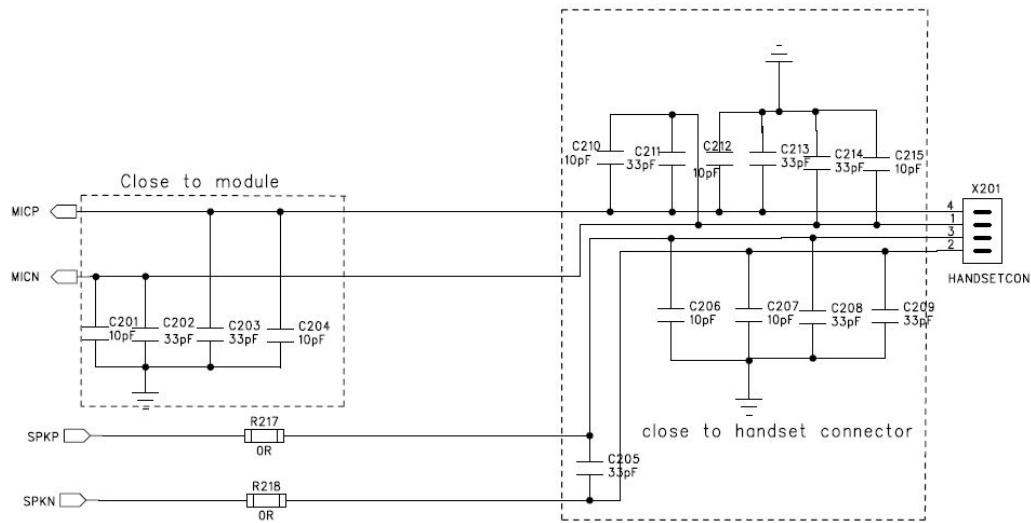


Figure 9: Referenced circuit of headset

### 2.4.2 Referenced circuit of hand free design

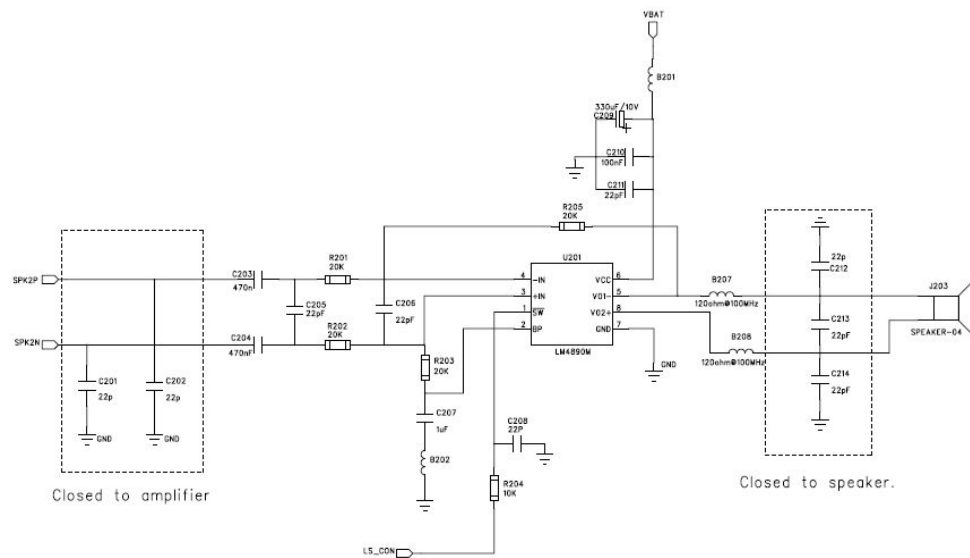
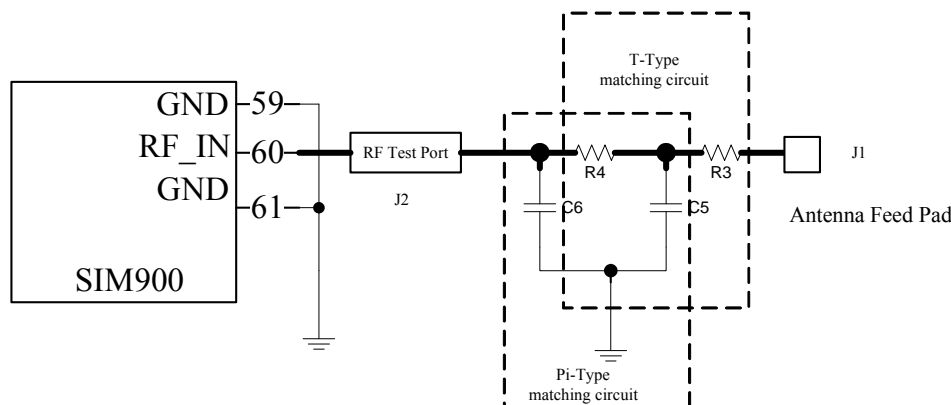


Figure 10: Referenced circuit of audio PA

## 2.5 Antenna matching circuit design

Because the module is working under 50ohm system in RF part, so, to get the best RF performance, the SMT module's load impedance should be tuned to 50ohm. But in fact, the most antenna's port impedance is not a purely 50ohm, so, to meet the 50ohm requirement, an additional

antenna matching circuit should be needed. Furthermore, to facilitate the antenna debugging and certification testing of RF performance, we suggested the customer add a RF test connector in series between the module's RF port and the antenna matching circuit. The recommended antenna matching circuit is shown as below:



**Figure 11: Antenna Matching Circuit**

In the Figure10, the components, R4, C5 and C6 make up a pi-type matching circuit structure. If add the optional component R3, then a T-type matching circuit structure will be made up with another two components R4 and C5. But usually, a pi-type matching circuit is enough in antenna tuning process. The component J2 is a RF test Port, used for conduct RF test. The traces in Bold type must be 50 ohm impedance controlled when layout a design.

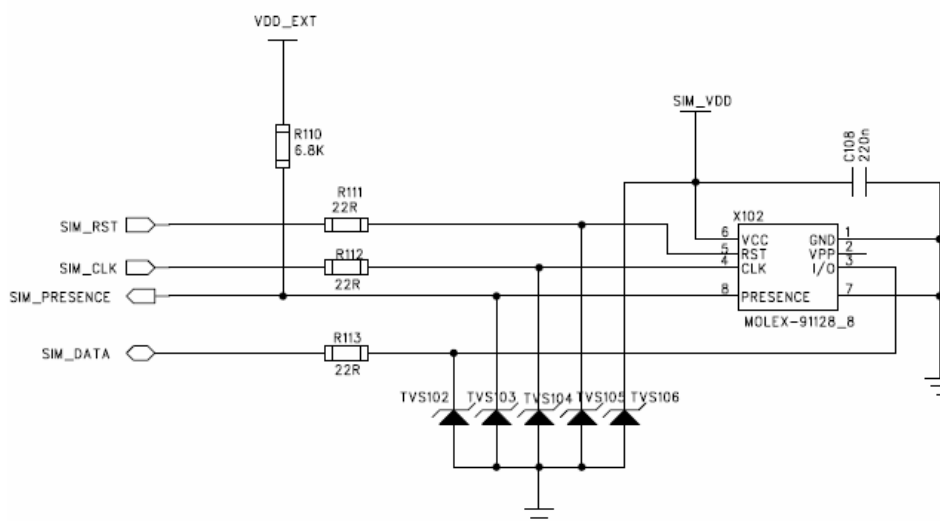
For the RF test connector, we suggested the customer use the part vended by Murata, its part number is MM8430-2610. For detail information about this part, the customer can visit Murata's website: <http://www.murata.com>.

**NOTE: For detail of RF Layout information, please to [4], SMT Module RF Design Guide.**

## 2.6 SIM card interface

The SIM interface is powered from an internal regulator in the module. Both 1.8V and 3.0V SIM Cards are supported. You can select the 8-pin SIM card holder. The reference circuit with 8-pin SIM card holder illustrates as following figure. We recommend an Electro-Static discharge device ST ([www.st.com](http://www.st.com)) ESDA6V1W5 or ON SEMI ([www.onsemi.com](http://www.onsemi.com)) SMF05C for ESD protection. The 22Ω resistor showed in the following figure should be added in series on the IO line between the module and the SIM card for matching the impedance. The SIM peripheral circuit should close to the SIM card socket.

The SIM\_PRESENCE pin is used for detecting the SIM card. There is a 100k pull down resistor in SIM900 module. So the R110 should not be bigger than 10K. If you don't use the SIM card detection function, you can let the SIM\_PRESENCE pin open or connect to the GND.



## 2.7 ESD protection

## 2.8 Consideration in PCB layout

- 1) The power trace should be short and wide, recommended above 80mil.
- 2) The layout of GROUND is very important. You should keep a full ground on top layer, and most of traces should be layout on the bottom layer, especially the audio traces, keep a whole GROUND under the module shield.
- 3) The audio traces (MIC & SPK) are better to be protected with ground.
- 4) The ground layer (top and bottom) must be connected with many vias, especially under the module's shield case.
- 5) The width of MIC traces should be 8-10mil. The width of SPK traces should be 12-14mil. The audio traces had better use difference connection and keep parallel.
- 6) The Layer1.Layer2 under SMT module test port should be copper keep out , layer3 should be GND;
- 7) The Layer2 under SMT module RF pad should be copper keep out , layer3 should be GND;
- 8) The Layer1.Layer2 under RF test connector should be copper keep out, layer3 should be GND

## **SIM900 Reference Design Guide Notes**

---

- 9) RF trace between SMT module RF pad with the RF test connector, RF trace between RF test connector with the antenna matching circuit, RF trace between the antenna matching circuit with the antenna feed PAD all should be controlled to 50 Ohm
- 10) All layers under the antenna feed pad should be copper keep out.
- 11) Do not layout RF trace in orthogonal
- 12) When layout surface Microstrip Transmission Line or offset Stripline Transmission Line , 3W rule should be followed, that means the distance between reference GND with RF trace should three times more than the width of RF trace.

**Contact us:**

**Shanghai SIMCom Wireless Solutions Ltd.**

Add: Building A, SIM Technology Building, No.633, Jinzhong Road, Changning District, Shanghai, P. R. China 200335

Tel: +86 21 3235 3300

Fax: +86 21 3235 3301

URL: [www.sim.com/wm](http://www.sim.com/wm)