EC25 Reference Design

LTE Module Series
Rev. EC25_Reference_Design_Rev.D
Date: 2016-11-11
Our aim is to provide customers with timely and comprehensive service. For any assistance, please contact our company headquarters:

Quectel Wireless Solutions Co., Ltd.
Office 501, Building 13, No.99, Tianzhou Road, Shanghai, China, 200233
Tel: +86 21 5108 6236
Email: info@quectel.com

Or our local office. For more information, please visit:
http://www.quectel.com/support/salesupport.aspx

For technical support, or to report documentation errors, please visit:
http://www.quectel.com/support/techsupport.aspx
Or email to: Support@quectel.com

GENERAL NOTES
QUECTEL OFFERS THE INFORMATION AS A SERVICE TO ITS CUSTOMERS. THE INFORMATION PROVIDED IS BASED UPON CUSTOMERS’ REQUIREMENTS. QUECTEL MAKES EVERY EFFORT TO ENSURE THE QUALITY OF THE INFORMATION IT MAKES AVAILABLE. QUECTEL DOES NOT MAKE ANY WARRANTY AS TO THE INFORMATION CONTAINED HEREIN, AND DOES NOT ACCEPT ANY LIABILITY FOR ANY INJURY, LOSS OR DAMAGE OF ANY KIND INCURRED BY USE OF OR RELIANCE UPON THE INFORMATION. ALL INFORMATION SUPPLIED HEREIN IS SUBJECT TO CHANGE WITHOUT PRIOR NOTICE.

COPYRIGHT
THE INFORMATION CONTAINED HERE IS PROPRIETARY TECHNICAL INFORMATION OF QUECTEL CO., LTD. TRANSMITTING, REPRODUCTION, DISSEMINATION AND EDITING OF THIS DOCUMENT AS WELL AS UTILIZATION OF THE CONTENT ARE FORBIDDEN WITHOUT PERMISSION. OFFENDERS WILL BE HELD LIABLE FOR PAYMENT OF DAMAGES. ALL RIGHTS ARE RESERVED IN THE EVENT OF A PATENT GRANT OR REGISTRATION OF A UTILITY MODEL OR DESIGN.

Copyright © Quectel Wireless Solutions Co., Ltd. 2016. All rights reserved.
About the Document

History

<table>
<thead>
<tr>
<th>Revision</th>
<th>Date</th>
<th>Author</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>2016-04-01</td>
<td>Winter CHEN</td>
<td>Initial</td>
</tr>
<tr>
<td>B</td>
<td>2016-08-22</td>
<td>Yeoman CHEN</td>
<td>1. Added ADC interface design in Sheet 1.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2. Added power supply for codec in Sheet 3.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>4. Changed some DGND to AGND in audio design in Sheet 5.</td>
</tr>
<tr>
<td>C</td>
<td>2016-10-14</td>
<td>Eden LIU</td>
<td>Added the reference design of SGMII and FC20 module</td>
</tr>
<tr>
<td>D</td>
<td>2016-11-11</td>
<td>Power JIN</td>
<td>1. Modified the connection of network name PCM_IN_BT and PCM_OUT_BT in Sheet 1</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>2. Added note 7 in Sheet 7</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>3. Added note 6 in Sheet 9</td>
</tr>
</tbody>
</table>
## Contents

About the Document ................................................................................................................................ 2  
Contents.................................................................................................................................................... 3  

1  Reference Design.................................................................................................................................. 4  
  1.1. Introduction ............................................................................................................................. 4  
  1.2. Schematics ............................................................................................................................. 4
1 Reference Design

1.1. Introduction

This document provides the reference design for Quectel EC25 module.

1.2. Schematics

The schematics illustrated in the following pages are provided for your reference only.
Module Interface

Notes:
1. It is recommended to reserve the test points for upgrading the firmware over USB interface and minimizing the stub length of USB test signals.
2. Do not pull up WAKEUP_IN_MODULE unless the module starts up successfully.
3. Do not pull up NET_MODE unless the module starts up successfully.
4. ADC pin can not be directly connected to the power supply and must not exceed the voltage range.
5. C101/C102 must be close to the SGMII interface of the module.
6. Keep all RESERVED and unused pins unconnected, and all GND pins are connected to the ground network.
7. Pin 73~84 are unused in the design. You can ignore them in schematic, PCB decal and the keepout area of pin 73~84.
8. PCM_***_BT network is used for the communication with FC20 module and CODEC_PCM_*** network is used for the communication with CODEC. FC20 and CODEC can not be connected to the same PCM interface at the same time.
Notes:
1. U201 represents customer's MCU.
2. EC25 can only work as a USB device and supports Full Speed and High Speed modes. To communicate with USB interface, MCU needs to support USB host or OTG function. The VBUS pins of MCU and EC25 should be powered by a 5V power system for USB detection, and VBUS-CTRL is used to turn on/off VBUS power supply.
3. AP READY can be configured for high level detection and low level detection. For more details about AP READY, please refer to Quectel_EC25_USB_Design and Quectel_EC25_USB_Commands_Manual.
4. Transistor circuits (Q203~Q206) are used for level translation.
Power Supply Design

DC-DC Application

It is used when the input voltage is above 7V. Use a DC-DC converter to convert a high input voltage into 5V output, and then the LDOs will generate 3.8V, 3.3V and 1.8V typical voltages.

e.g. DC12V IN DC-DC 5V OUT LDO DC 3.8V 2.0A EC25

LDO Application

It is used when the input voltage is below 7V.

LDO 3.3V
LDO 1.8V
LDO 3.3V 1.2A
Codec

Supply Power for FC20 and SGMII

VDD3V3=(R314/R317+1)*1.24=3.3V

Supply Power for PCM Codec

VDD_1V8=(R310/R312+1)*1.207=1.8V

VDD_3.3V=(R305/R308+1)*1.207=3.3V

LDO VBAT=(R301/R306+1)*1.24=3.88V

VBAT Design

Connect to VBAT_BB pins.

Connect to VBAT_RF pins.

Notes:
1. The power supply must be able to provide sufficient current up to 2A or more.
2. VBAT should be routed in star mode to VBAT_BB and VBAT_RF pins.
3. The recommended operating voltage of VBAT is 3.3V~4.3V.

Notes:
1. Recommended load current is greater than 10mA.

Notes:
2. VBAT should be routed in star mode to VBAT_BB and VBAT_RF pins.

Notes:
1. Recommended load current of MIC29302WU is greater than 10mA.
2. RC circuit, which is assembled with R318 and C327, is used to delay the start-up of MOSFET switch circuit.

Supplementary Information

Recommended load current is greater than 10mA.

RC circuit, which is assembled with R318 and C327, is used to delay the start-up of MOSFET switch circuit.
USIM and UART Design

Notes:
1. The decouple capacitor of USIM_VDD should be less than 1uF and must be near to USIM socket.
2. EC25 module provides an input pin (USIM_PRESENCE) to detect whether the USIM card exists or not. It supports both low level and high level detections. For more details, please refer to Quectel_EC25_Hardware_Design.

Notes:
1. R401~R403 are applied to suppress the EMI spurious transmission and enhance the ESD protection.
2. It is recommended to take electrostatic discharge (ESD) protection measures near the USIM card socket. The TVS diode with junction capacitance less than 50 pF must be placed as close as possible to the USIM socket.
3. R404 can improve anti-jamming capability of the USIM circuit, and it should be placed close to the USIM connector.
4. VCCA should not exceed VCCB. For more information about TXS0108E, please refer to the datasheet from TI.
5. If you need to enable high baud rate, it is highly recommended to install a 1nF capacitor (C407/C408/C409/C410) on transistor circuit.
6. The DTR transistor circuit is similar to RTS interface. The RI and DCD transistor circuits are similar to CTS interface.

Notes:
1. The decouple capacitor of USIM_VDD should be less than 1uF and must be near to USIM socket.
2. EC25 module provides an input pin (USIM_PRESENCE) to detect whether the USIM card exists or not. It supports both low level and high level detections. For more details, please refer to Quectel_EC25_Hardware_Design.

Notes:
1. R401~R403 are applied to suppress the EMI spurious transmission and enhance the ESD protection.
2. It is recommended to take electrostatic discharge (ESD) protection measures near the USIM card socket. The TVS diode with junction capacitance less than 50 pF must be placed as close as possible to the USIM socket.
3. R404 can improve anti-jamming capability of the USIM circuit, and it should be placed close to the USIM connector.
4. VCCA should not exceed VCCB. For more information about TXS0108E, please refer to the datasheet from TI.
5. If you need to enable high baud rate, it is highly recommended to install a 1nF capacitor (C407/C408/C409/C410) on transistor circuit.
6. The DTR transistor circuit is similar to RTS interface. The RI and DCD transistor circuits are similar to CTS interface.
To make sure the function works normally, please follow ALC5616 power on and off sequence.

Note:
For more details please refer to ALC5616 datasheet.

1. The analog output only drives earphone and headset. For larger power loads such as speakers, the design needs to increase the audio power amplifier.
2. The maximum capacitive loading for SPK is 330 pF and the maximum capacitive loading for MIC is 250 pF.
RF and GNSS Design

Main Antenna Interface

Diversity Antenna Interface

GNSS Antenna Circuit

FC20 Antenna Circuit

Notes:
1. The main antenna circuit, diversity receiving antenna circuit and FC20 antenna circuit are recommended to use Pi type circuit, which is convenient for subsequent debugging.
2. The diversity reception function is ON by default. If diversity antenna is not used, there is a need to use AT command to turn off diversity reception.
3. You can choose an external LDO according to the active antenna to supply power.
4. If you design the antenna circuit with passive antenna, the R603 and L603 are not needed.
5. ESD protection devices should be added to the GNSS antenna interface, and the parasitic capacitance should be less than 0.05pF.
1. SGMII_MDI0_DAT should be connected to the USIM2_VDD beside the module with a 1.5K pull-up resistor.

2. To minimize crosstalk, the reset trace must be at least 20 mils away from other signal traces.

3. R715 must be close to AR8033. The traces of the resistor must be away from other traces (especially the clock and MDI interface traces), and the trace width needs to be at least 25 mils.

4. The two capacitors should be selected according to the actual load capacitance of crystal and the board-level test results, at full load application temperature and voltage range.

5. The differential trace impedance of SGMII must be limited to 100Ω.

6. EMI filter is reserved. If LED pins are not used, keep C714, C719, C721=470pF.

7. The space between SGMII differential pair (RX and TX traces) should be 3 times of the trace width at least. Also, SGMII should keep a distance of 3 times of its trace width from other signal lines.

Note 1: 1. SGMII_MDI0_DAT should be connected to the USIM2_VDD beside the module with a 1.5K pull-up resistor.

Note 2: 2. To minimize crosstalk, the reset trace must be at least 20 mils away from other signal traces.

Note 3: 3. R715 must be close to AR8033. The traces of the resistor must be away from other traces (especially the clock and MDI interface traces), and the trace width needs to be at least 25 mils.

Note 4: 4. The two capacitors should be selected according to the actual load capacitance of crystal and the board-level test results, at full load application temperature and voltage range.

Note 5: 5. The differential trace impedance of SGMII must be limited to 100Ω.

Note 6: 6. EMI filter is reserved. If LED pins are not used, keep C714, C719, C721=470pF.

Note 7: 7. The space between SGMII differential pair (RX and TX traces) should be 3 times of the trace width at least. Also, SGMII should keep a distance of 3 times of its trace width from other signal lines.
SGMII Design (Part 2)

<table>
<thead>
<tr>
<th>Configuration Signal</th>
<th>Description</th>
<th>Default Internal weak pull-up/down</th>
<th>Application External weak pull-up/down</th>
</tr>
</thead>
<tbody>
<tr>
<td>PHY_AD2</td>
<td>PHY_AD[2:0] set the lower three bits of the physical address.</td>
<td>1</td>
<td>0</td>
</tr>
<tr>
<td>PHY_AD1</td>
<td>The upper two bits of the physical address are set to 00.</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>PHY_AD0</td>
<td>MODE 3</td>
<td>Mode select bit 3</td>
<td>0</td>
</tr>
<tr>
<td>MODE 2</td>
<td>Mode select bit 2</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>MODE 1</td>
<td>Mode select bit 1</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>MODE 0</td>
<td>Mode select bit 0</td>
<td>0</td>
<td>1</td>
</tr>
<tr>
<td>EXT_INT_SEL</td>
<td>An external 10K pull-down resistor is required.</td>
<td>1</td>
<td>0</td>
</tr>
</tbody>
</table>

0=Pull-down, 1=Pull-up.

**Notes:**
1. Differential pair P/N skew must be less than 20 mils, and the maximum trace length must be less than 10 inches.
2. Using the 100 ohm ± 10% differential impedance with 50 ohm ± 10% single-ended impedance.
3. To minimize crosstalk, the distance between separate adjacent pairs that are on the same layer must be equal to or larger than 40 mils.
4. Copper filling around transformers is prohibited for better ESD protection performance.
5. For better EMI suppression performance, do not route in top layer.
1. Keep all RESERVED and unused pins unconnected.
2. BT function of FC20 is under development.
3. SDIO_DATA2 is a signal that affects the boot of the module, and should be kept at high level when the power supply resets.
4. The impedance of the SDIO data signal line must be controlled as 50Ω when routing.
5. SDIO data lines should be wrapped together by GND lines; SDIO_CMD and SDIO_CLK network lines should be wrapped with GND lines separately.
6. Please keep the pin 5~8, 10, and 13~18 open in FC20-N.

R806 must be mounted.
Indicators and Test Points

Indicators

Notes:
1. The STATUS is an open drain output pin, and its drive current is less than 1mA.
2. For more details about NET_MODE and NET_STATUS, please refer to the document Quectel_EC25_Hardware_Design.

Reserved Test Points

Notes:
1. Both USB and debug UART interfaces are reserved for software debugging.
2. USB interface also can be used to upgrade firmware.
3. Keep USB test points as close as possible to USB pins.
   Junction capacitance of ESD component on USB data lines might influence the signal, please pay attention to it. Typically, the capacitance should be less than 1pF.

Note:
When USB_BOOT is at high level, the module will be forced to be in download mode.