WT12-A

Design Guide

Version 1.1

Wednesday, September 12, 2007

bluegiga TECHNOLOGIES

Contents:

1. Introduction 4

2. Design recommendations 5

2.1 PCB material 5

2.2 Grounding 5

2.3 Signal tracks 6

2.4 Filtering 7

2.5 Antenna 8

3. Example designs 8

3.1 Audio design 8

3.2 UART interface 9

3.3 USB interface 10

3.4 PCB 11

VERSION HISTORY

Version:	Author:	Comments:
1.1	Pasi Rahikkala	Example designs added
1.0	Teemu Ruokokoski	First release 9.9.2005

TERMS & ABBREVIATIONS

Term or Abbreviation:	Explanation:
Bluetooth	Set of technologies providing audio and data transfer over short-range radio connections
EMC	Electromagnetic compatibility
I/O	Input/Output
PCB	Printed circuit board
RF	Radio frequency
UART	Universal Asynchronous Receiver / Transmitter
USB	Universal Serial Bus

1. INTRODUCTION

This guide gives recommendations how to design electronics around WRAP THOR WT12-A module. EMC issues are considered in chapter 2 and example designs for the most common ways of connecting WT12 in different applications are provided in chapter 3.

Although WT12-A module has an integrated antenna there are still things that can go wrong. It's strongly advised to treat RF with respect to achieve the best performance possible. Careless electronics design can result in EMC problems and a deteriorated performance. An awareness of the EMC design of a product is the only way to guarantee disturbance resistant products.

A day spent at the beginning of the project preventing EMC problems can save a month of fixing problems at the end.

Following chapters give several recommendations how to do a good design. Nevertheless, a designer should never copy and paste a reference design and expect it to work. In the world of RF nothing is certain. A situation varies from design to design. So the designer should always stop to think. Do not trust on luck. It is a certain way to encounter problems.

2. DESIGN RECOMMENDATIONS

2.1 PCB material

A multilayer (for example 4-layer) PCB is recommended. A good grounding is always available in a multilayer design. You can also hide signals traces and supply lines inside the PCB. By doing that they are safe from a radiating antenna and do not pick up RF so easily.

However, if 2-layer PCB is chosen for cost effective reasons take special care of grounding, filtering etc. A design should be small and simple with the 2-layer PCB.

Never use 1-layer PCB. It won't work.

2.2 Grounding

- Do not remove copper from the PCB more than needed. Use ground filling as much as possible. However remove small floating islands after copper pour.

- Do not place a ground plane underneath the antenna. The area around the antenna must be designed as shown in Figure 1.

- Use conductive vias separated max. 3 mm apart at the edge of the ground areas. This prevents RF to penetrate inside the PCB. Use ground vias extensively all over the PCB. If you allow RF freely inside the PCB, you have a potential resonator in your hand. All the traces in (and on) the PCB are potential antennas.

- Avoid loops.

- Ensure that signal lines have return paths as short as possible. For example if a signal goes to an inner layer through a via, always use ground vias around it. Locate them tightly and symmetrically around the signal vias.

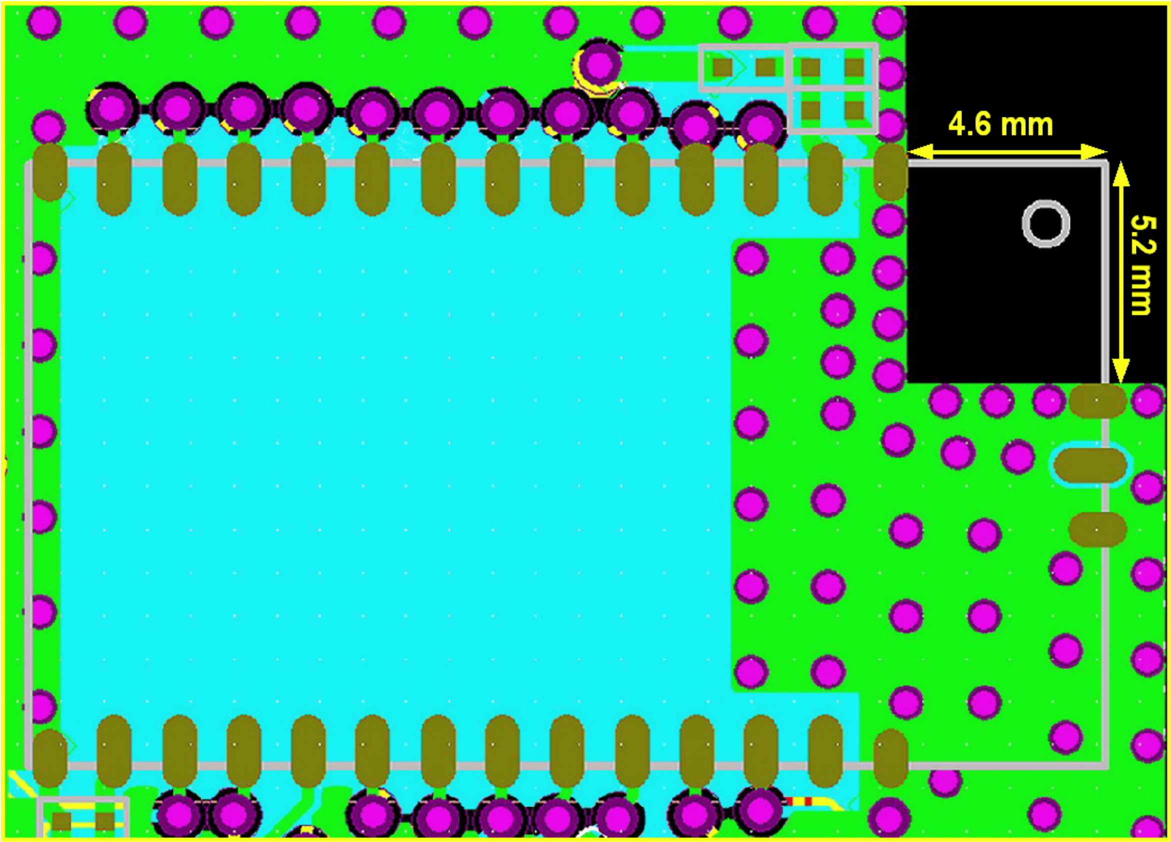

Figure 1: Ground areas under the WT12-A module

(green color is layer 1 copper and light blue is layer 2 copper)

2.3 Signal tracks

- Routing should be done in the inner layers of the PCB. For example use layer 2 for the signals and layer 3 for the supply lines.
- Traces should have a ground area above and under the line. If this is not possible make sure that the return path is short by other means (for example using a ground line next to the signal line).
- Avoid long parallel lines close to each other.
- When using two signal vias close to each other, block the direct coupling path with ground vias.
- Avoid crossings. Examples:
  - A signal line in layer 2 crossing a supply line in layer 3 is not recommended.
  - A signal crossing a gap in a ground area is not recommended. By doing that a return current coming back in a ground has to find a path by circling around the gap and this results in a loop.

2.4 Filtering

Always use high-Q components!

Use filtering especially on supply lines. A special care should be taken to filter RF frequencies at 2441 MHz.

You should make pads for two bypass capacitors near the 3V3 pin of the module. One capacitor is from 10 nF to 100 nF. The other one is meant to filter the RF and it is initially Not Placed. This capacitor is only used in the case of problems. The value is from 5 pF to 100 pF. The value varies from design to design and it must be found out experimentally.

You should also place one series inductor in the supply trace. The value of the inductor is expected to be from 10 nH to 100 nH.

Signal lines too can pick up noise, interfere with each other or pick up RF. Filtering can help in case of problems.

The following figures show some ideas for the designs.

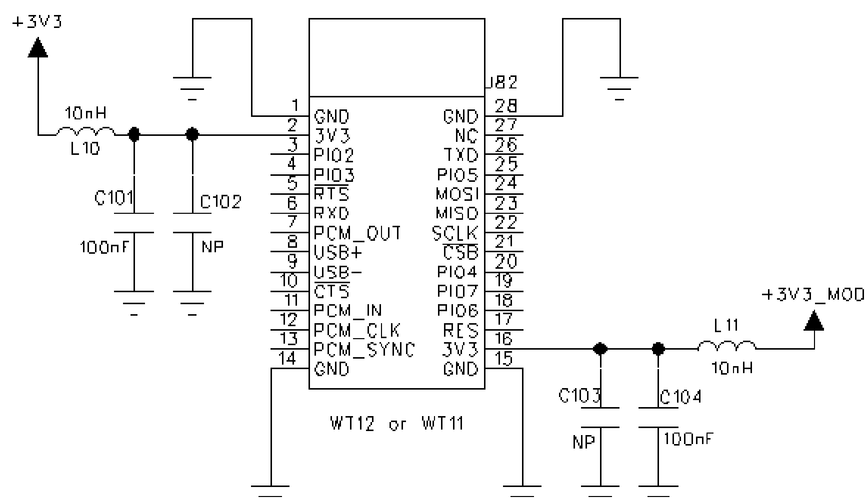

Figure 2: Filtering the supply lines

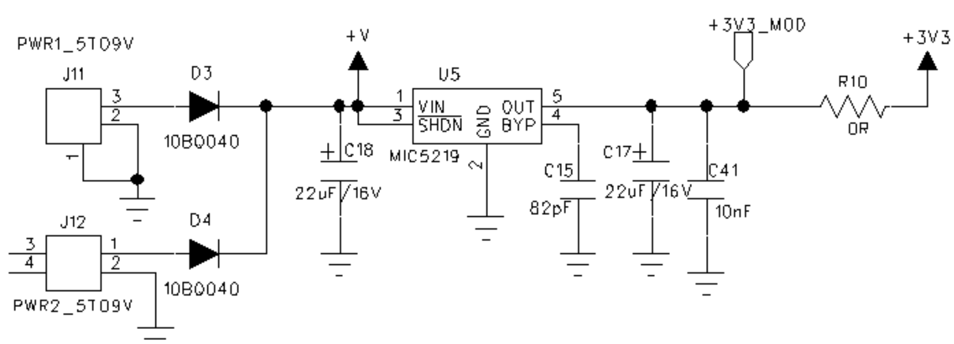

Figure 3 Example of a power supply design

2.5 Antenna

WT12-A module has an internal antenna which makes the module easy to integrate into a product. Nevertheless, to ensure proper operation it is very important to follow recommendations regarding the copper keep out area. Bluegiga offers design references and gerber files how to design grounding around the module.

The following list can be used as a checklist for the design:

- Place the WT12-A module to the edge of the PCB
- Do not enclose the module in the metal shielding.
- Do not place ground plane or traces underneath the antenna.
- Do not place the antenna close to any metal objects.
- Do not place wiring near the antenna.
- Do not use very thin PCB tracks.

Test the plastic casing for high RF losses.

3. EXAMPLE DESIGNS

3.1 Audio design

WT12 interfaces directly to several PCM audio devices. Figure 4 shows an example of an audio design using MSM7702 audio codec IC. R1 and R2 provide a bias voltage for the microphone. C1 together with R1 filters noise coupled from the power supply. R3 and R4 set the gain for the signal from the microphone. Consult the datasheet of the appropriate codec IC for detailed description. PCM signals (CLK, SYNC, IN, OUT) are connected respectively to the PCM interface of WT12.

For the output volume control an external amplifier or attenuation circuitry is required when using 8-bit sample format in 8-bit slot. When using 16-bit slots, a programmable 3-bit audio attenuation compatible with some Motorola audio codecs can be used. In this case the output volume can be tuned using PSKEY settings or dynamically through software by connecting the volume control buttons to certain PIO pins of the WT12 module.

SPI interface of WT12 is used to program the PSKEY settings (see the datasheet of WT12) with PSTool software available in bluegiga's Techforum. The example design shown in figure 4 is verified with following PSKEY settings:

PCM_DATAFORMAT:	0x0200
PCM_CONFIG32:	0x180000C0
HOSTIO_MAP_SCO_PCM:	TRUE

UART interface is used to set the appropriate configurations for the module through iWRAP. See iWRAP user guide for detailed description how to set up a headset application.

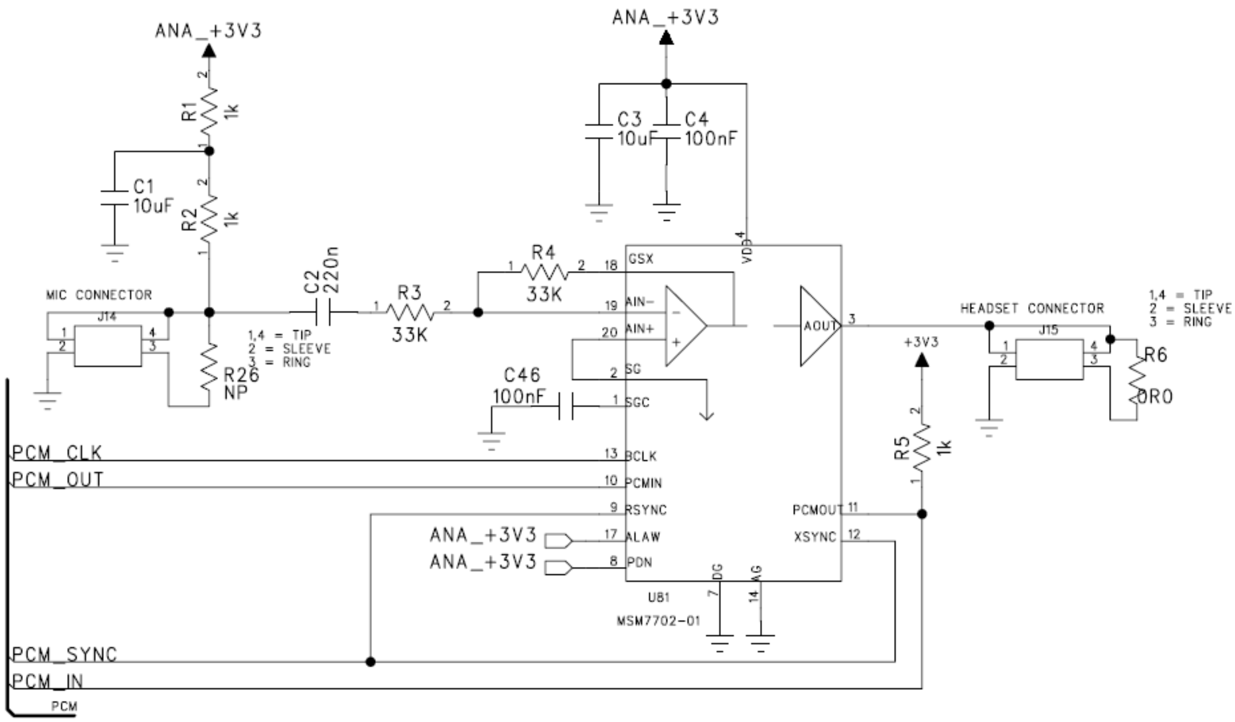

Figure 4: Audio design for the WT12 bluetooth module

3.2 UART interface

UART interface of WT12 uses voltage levels of 0V and Vdd. Thus an external RS232 transceiver IC is required. Figure 3 shows an example of an UART implementation. PIO3 – PIO5 pins of WT12 are used for DSR, DCD and DTR respectively.

If UART_RX signal pin is not connected during normal operation, a pull-up resistor has to be used, unless UART interface is disabled using PSKEY settings. See datasheet of WT12 for detailed description of UART configurations.

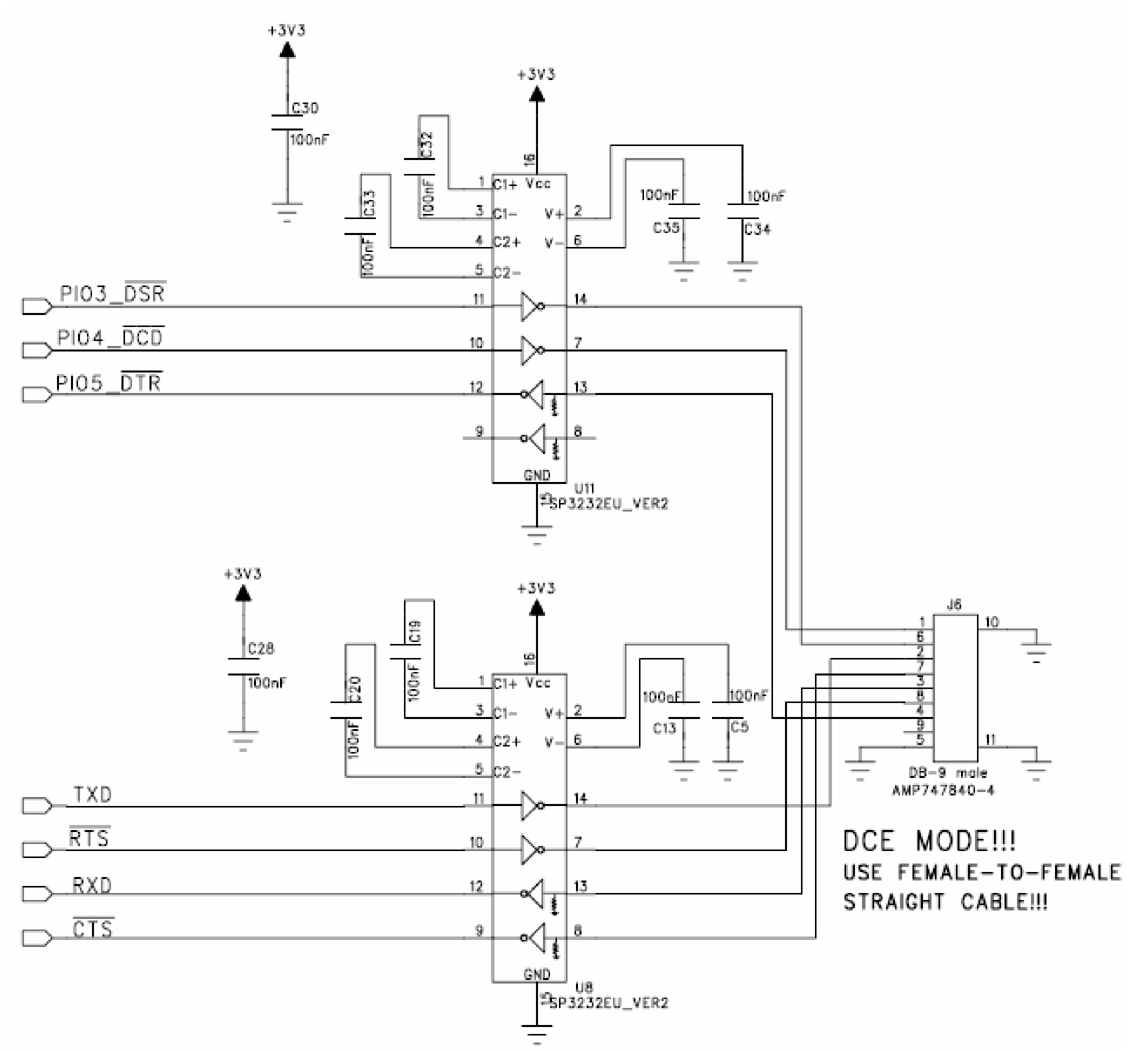

Figure 5: Example of an UART implementation for WT12

3.3 USB interface

Figure 6 shows an example of how to connect the USB interface of WT12. Transient voltage suppressor array U1 provides an ESD protection for the WT12 USB interface. Resistors R7 – R9 are connected as described in the datasheet of WT12.

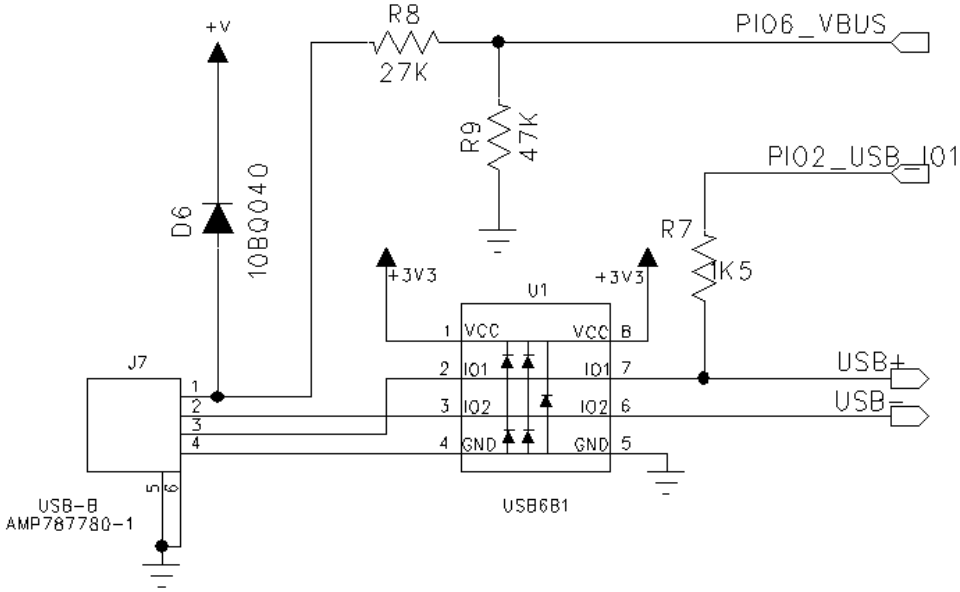

Figure 6: Example of an USB interface circuitry

3.4 PCB

Following picture shows an example of a good PCB design. The module is placed at the edge of the PCB and the antenna can radiate freely as it has free space around it. Ground planes are large and ground vias are used extensively. Signal lines are situated mostly inside the PCB.

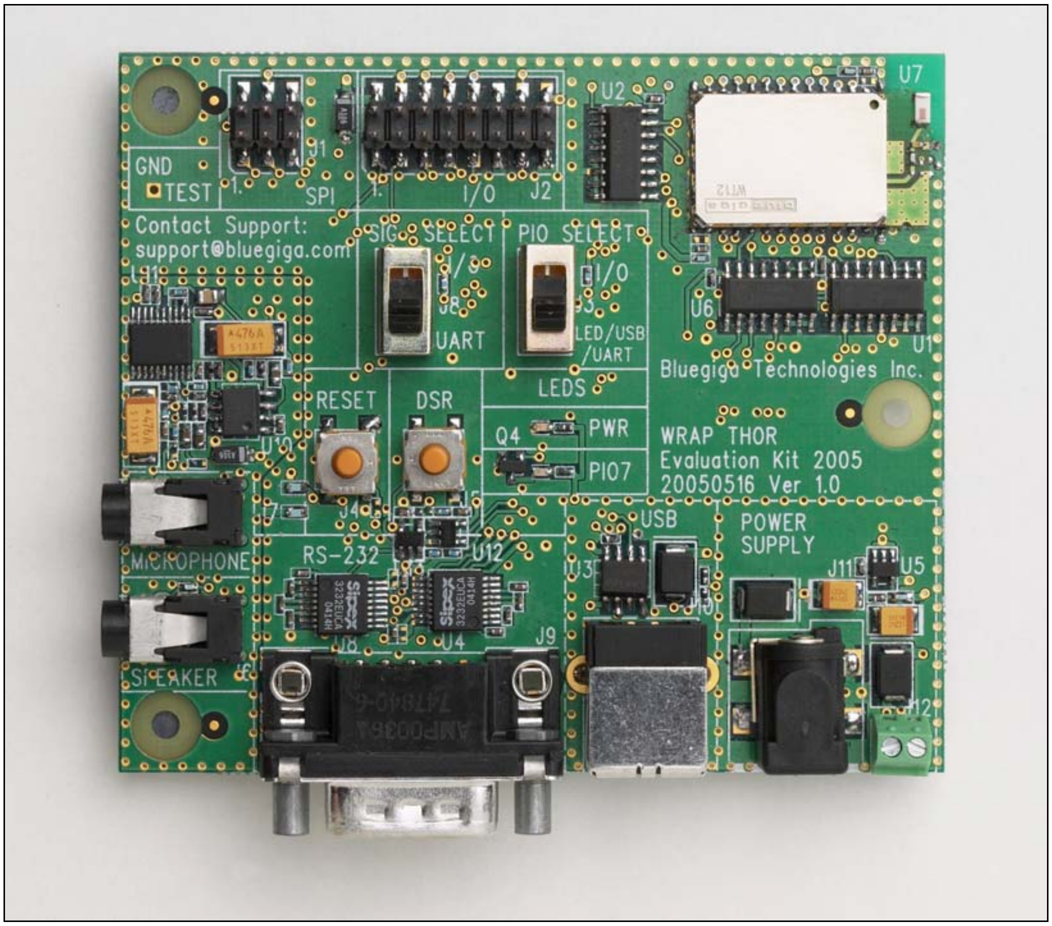

Figure 7: Design example (WT12 Evaluation Kit)