WRAP THOR 2022-1

Design Guide

Version 1.0

Friday, June 17, 2005

blue giga
TECHNOLOGIES

Contents:

1. VERSION HISTORY

Version:	Author:	Comments:
1.0	Teemu Ruokokoski	First release 27.5.2005

2. 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

3. INTRODUCTION

Although Bluegiga offers design references and an other material to support customers design process it's not always straightforward to integrate WRAP THOR 2022-1 module onto ones own design. This guide gives recommendations how to design electronics around the module. 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. However 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.

4. DESIGN RECOMMENDATIONS

4.1 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. That way 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, antenna placement, filtering etc. A design should be small and simple with the 2-layer PCB.

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

4.2 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.

- 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 a return path 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.

4.3 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.

4.4 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. Initially use 0 ohm resistor in its place. In case of problems change the component to an inductor. 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 of filtering.

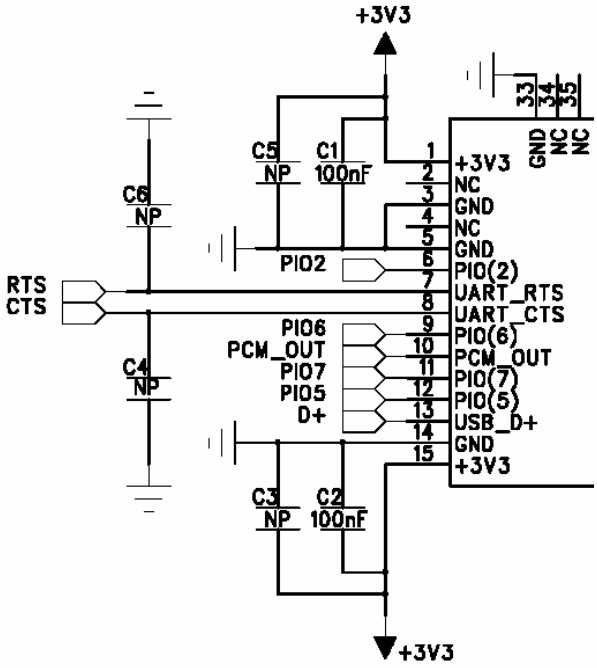

Figure 1: Using of bypass capacitors in supply lines and UART signals

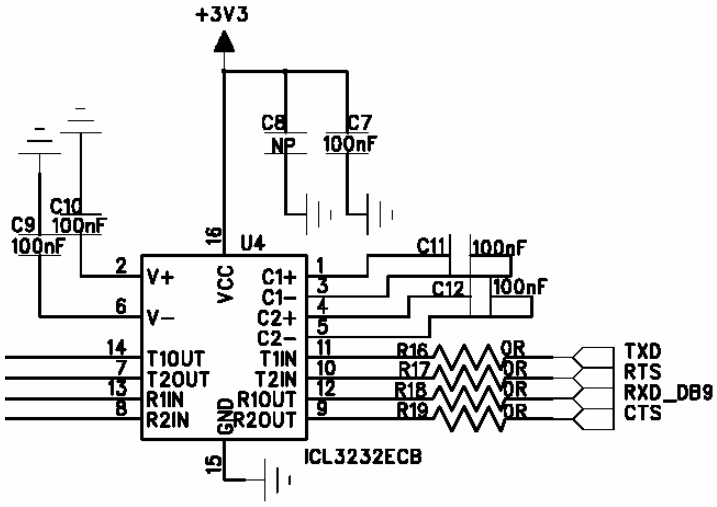

Figure 2: UART

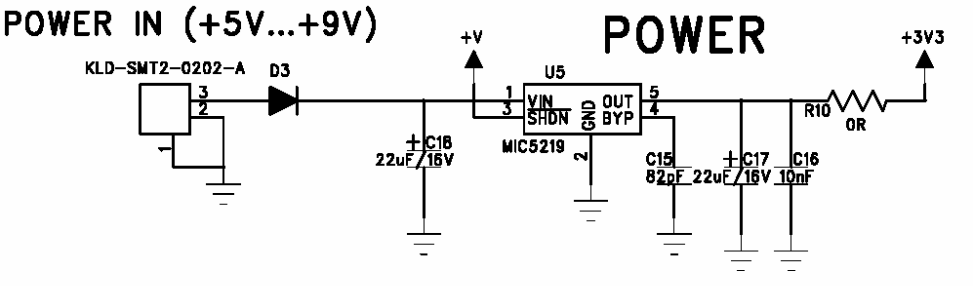

Figure 3: An example of a good power supply design and filtering the supply line

4.5 Antenna

The most important factor in obtaining good range and stable throughput in a wireless application is good antenna design. A designer should have at least a basic knowledge about how antennas function. It is important to understand design parameters such as antenna tuning, matching, gain, loss and radiation pattern. A lot of external factors affect the antenna matching, gain and radiation patterns. An antenna tuned for one environment may require completely different tuning in another environment.

4.5.1 Antenna structures

There are numerous antenna structures that provide good efficiency and impedance match. However most of these are derived from a few basic structures. A monopole type of antenna must be used with WRAP THOR 2022-1. This means that the ground plane makes up the remaining half of the antenna. In a practical application the ground plane of the antenna is in fact made of ground and supply planes, traces and components of the PCB. As a result the ground plane should be a reasonably sized area. Use ground filling as much as possible. Some antenna types are introduced in the following.

F-Antenna:

The F-Antenna is a tilted whip where impedance matching is done by tapping the antenna at the appropriate impedance point. The antenna has an omnidirectional radiation pattern and good efficiency. It needs large sized ground plane to be efficient. The F-Antenna is recommended for the cost efficient applications which have enough space for the antenna.

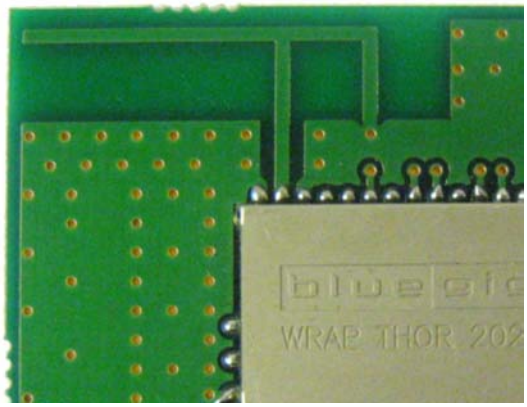

Figure 4: WRAP THOR 2022-1 with F-Antenna

Chip antenna:

Many different chip antennas are available commercially. To ensure proper operation it is very important to follow manufacturer's recommendations regarding footprint and ground areas. The keep out area around the antenna is especially important. The fine tuning of the matching network must be done for the end product. All the nearby objects affect the matching and the performance of the chip antenna. The chip antennas are usually made on a ceramic material which has higher dielectric constant and lower loss than the FR4. As a result chip antennas provide the smallest antenna solution. However the size reduction comes at a cost both in performance and pricing.

A good and small chip antenna is for example ACX AT3216.

Meander antenna:

The meander antenna is an antenna with the line folded back and forth. The resonance is found in a more compact structure than otherwise possible.

4.5.2 Antenna matching

The antenna must be matched to 50 ohms. There are numerous ways to do the matching and it depends on the type of the antenna. Usually you can match a chip antenna with two components: one component in series and one component in parallel in a transmission line. The components you can use for matching is inductor and capacitor (LC-network). It is very important to use high-Q components. Never change the matching component to another manufacturers "equal" type without testing. This is because the type is never equal.

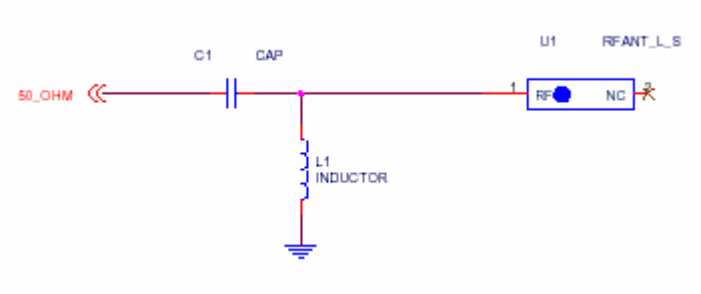

Figure 5: An example of antenna matching

4.5.3 Checklist

Many things can go wrong with an antenna design. The following list can be used as a checklist:

- Do not enclose the antenna 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.
- A monopole antenna should have large ground plane to be efficient.
- Match the antenna to 50 ohms. However do the final tuning in the end product.
- Use high-Q matching components. Never change manufacturer without retesting.
- Do not copy and paste a reference design and expect it to work without testing and tuning. Use the reference design only as a starting point.
- Do not use very thin PCB tracks.
- Test the plastic casing for high RF losses.

4.5.4 Transmission lines

Transmission lines are designed to transport RF power with as little radiation loss as possible. The structure is designed to contain the electromagnetic fields. A recommended type for a RF-pin of the WRAP THOR 2022-1 module is grounded coplanar waveguide. The coplanar stripline contains the RF very well and the matching components of the antenna are easy to implement to the design.

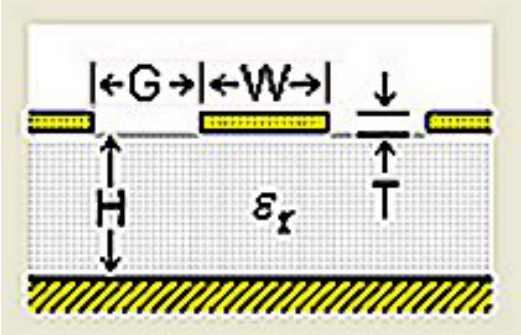

Figure 6: Grounded coplanar waveguide

The impedance of the transmission line must be 50 ohms. Parameters affecting the impedance of the grounded coplanar waveguide are:

- w width of the stripline
- h height of the stripline from the ground plane
- g gap between the stripline and the ground plane
- ε_r dielectric constant of the PCB
- t thickness of the copper
- f frequency

Free software to calculate TX-lines is available and can be found from the web easily.

5. DESIGN EXAMPLE

Following picture shows an example of a good design. The chip antenna can radiate freely as it has a lot of space around it. Ground planes are large and ground vias are used extensively. Tracks are situated mostly inside the PCB.

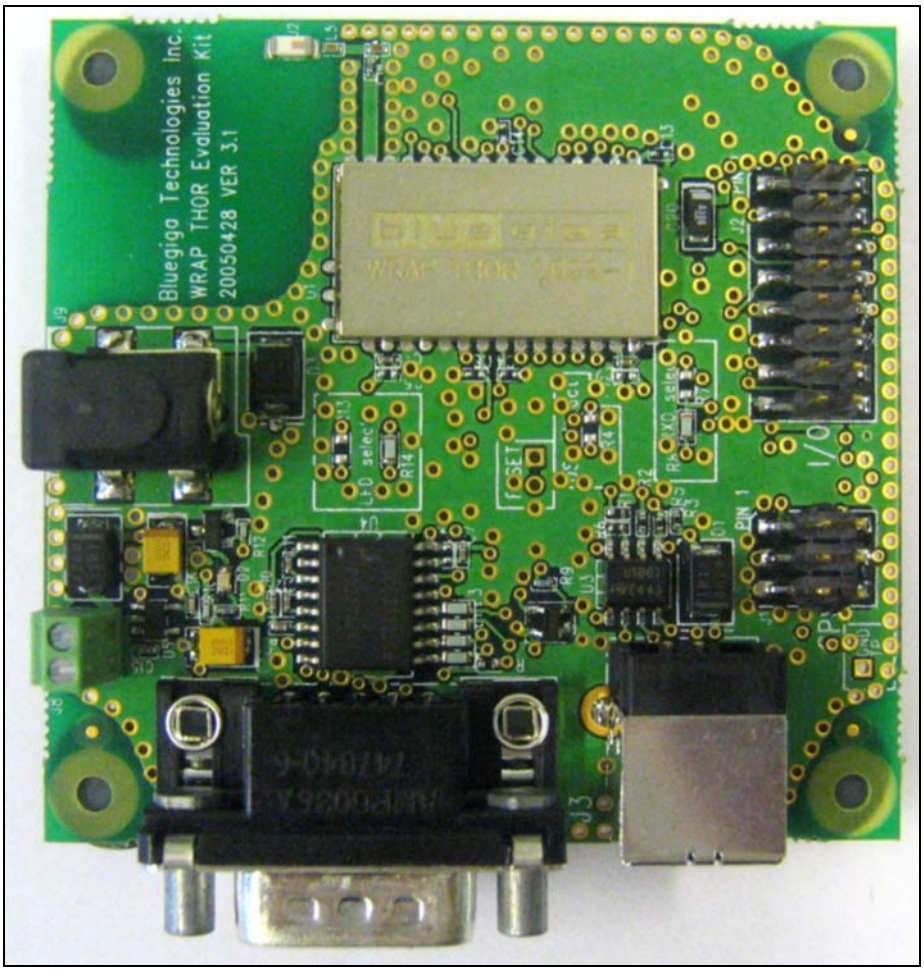

Figure 7: Design example (WRAP THOR 2022-1 Evaluation Kit)