

# US2400

Low Power 2.4 GHz Transceiver  
IEEE 802.15.4 Standard

## PCB Layout Guide AN-2400-90

The content of this technical document is subject to change without notice. Please contact UBEC for further information.

**Version: 0.0**

**Released Date: 2011/08/09**

All rights are strictly reserved. Any portion of this document shall not be reproduced, copied, or transformed to any other forms without permission from Uniband Electronic Corp.

## 1. Introduction

---

To achieve the best RF performance out of US2400, the 2-Layer PCB layout is recommended as described in this document.



Figure 1: Top view of US2400

## 2. PCB Layout Rules

---

Key points to a good PCB Layout:

(1) PCB:

- A. Refer to the IC outline drawing, which is given in Datasheet, DS-2400-55, for related US2400 dimensions.
- B. To realize the best performance out of US2400, the ground plane should be as a continuous plane to the extent possible. Any isolated ground planes should be connected or bridged with wide-width (typically, 10 to 16 mil wide) copper traces.
- C. Since the bottom side ground of US2400 is divided into a 3-by-3 patch grid, if the ground plane on the PCB, onto which the US2400 is to be mounted, is a whole continuous piece, one can employ a stencil with openings matching the US2400's 3-by-3 patch grid during the solder paste application process. The stencil design is described in more details below.

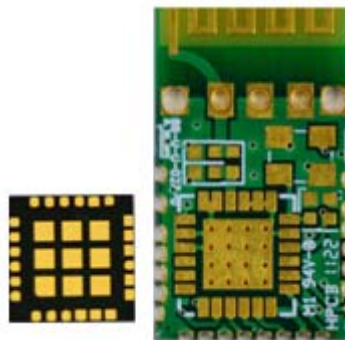


Figure 2: Bottom view of US2400 and sample PCB

- (2) Stencil
  - A. Follow the patch location and its size and create a corresponding opening area in the stencil for receiving the solder paste.
  - B. The size of the opening area in the stencil should be around 80% ~ 90% of the US2400 metal patch size. However if the patch size is large than 1 mm<sup>2</sup>, the size of the stencil opening area should be reduced to about 60% ~ 75% of the patch area. This is to prevent undesirable effects arising from the cohesive force of the excessive amount of the molten solder.
- (3) The thickness for the 2-layer FR4 board is 0.8mm. The thickness of the cooper foil is 0.5 ounce. The width of 50 ohm trace is 0.6096mm.
- (4) Reference schematic and layout are attached as Appendix A to this document in Protel 99SE format.

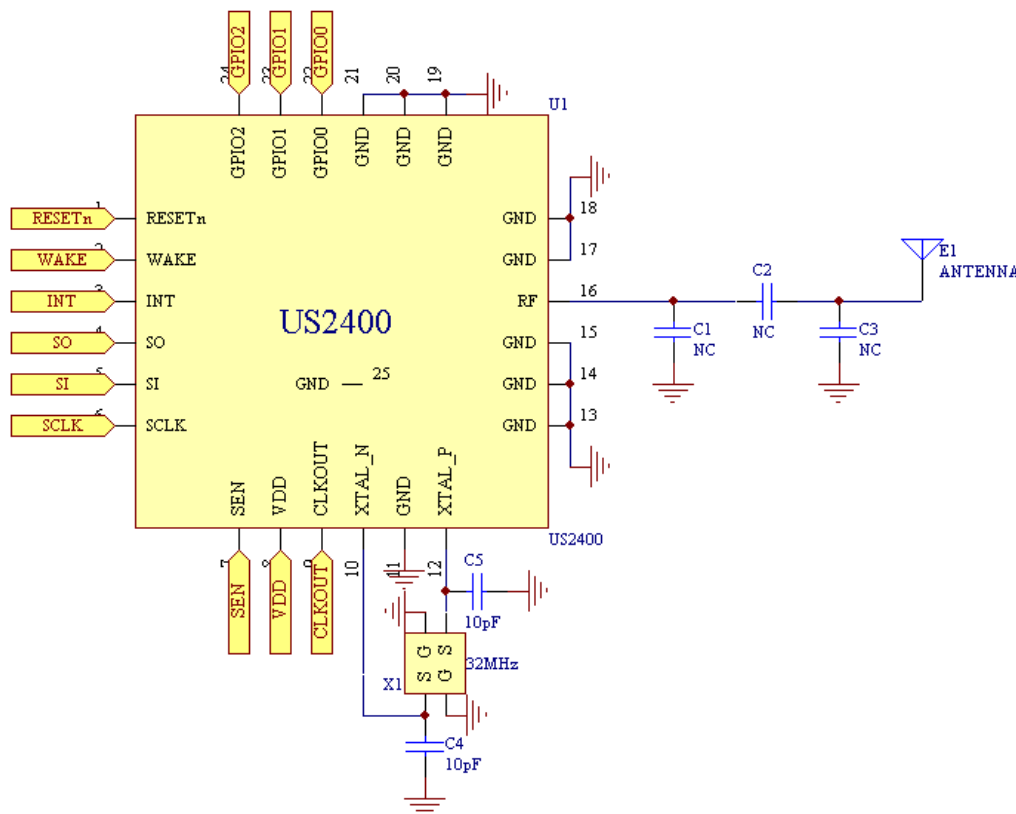
### 3. Summary

It is highly recommend that users follow this guide to avoid the potential pitfalls when doing the layout design.

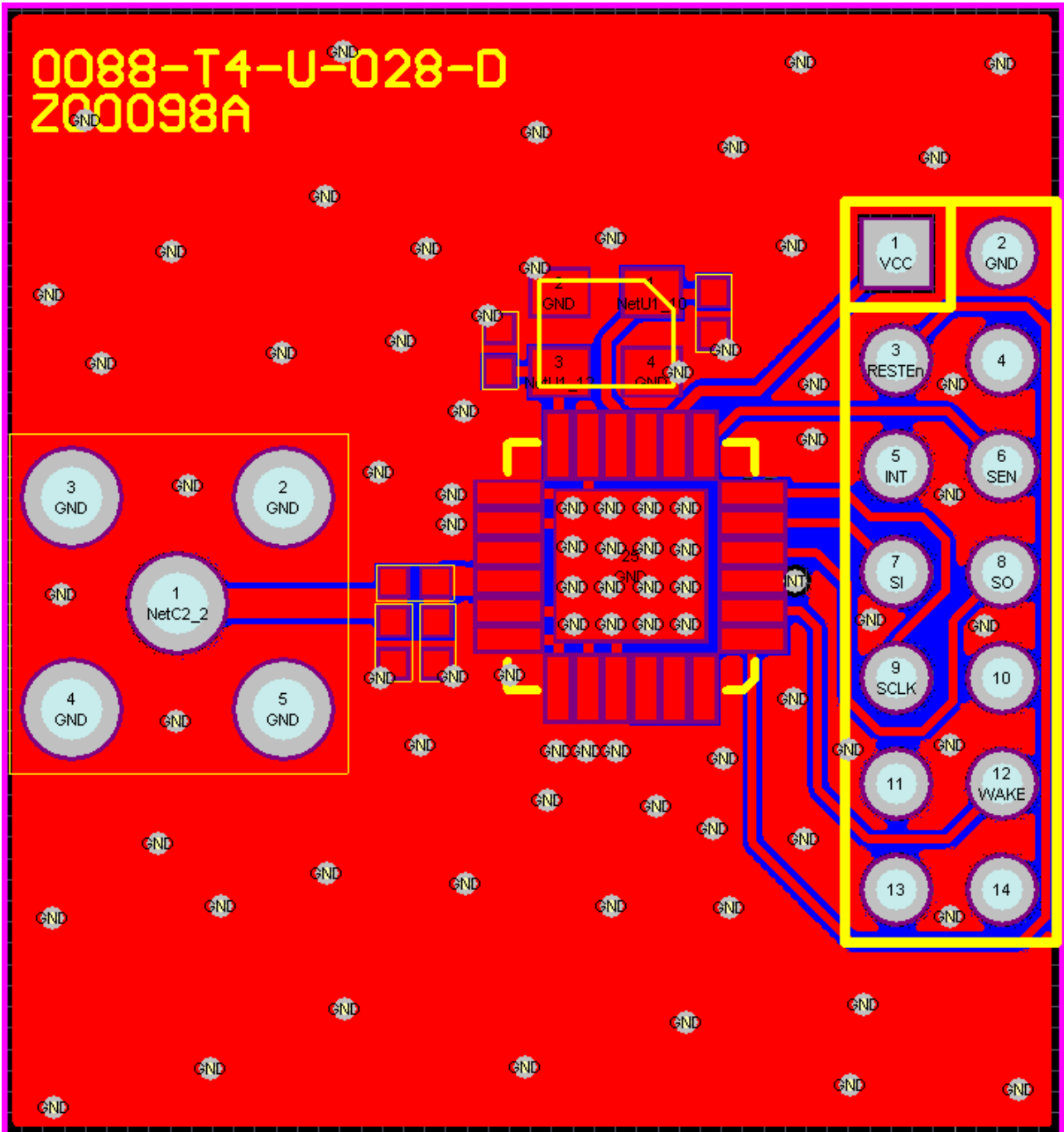
### 4. Attachment

Appendix A: Reference schematic and layout for US2400.

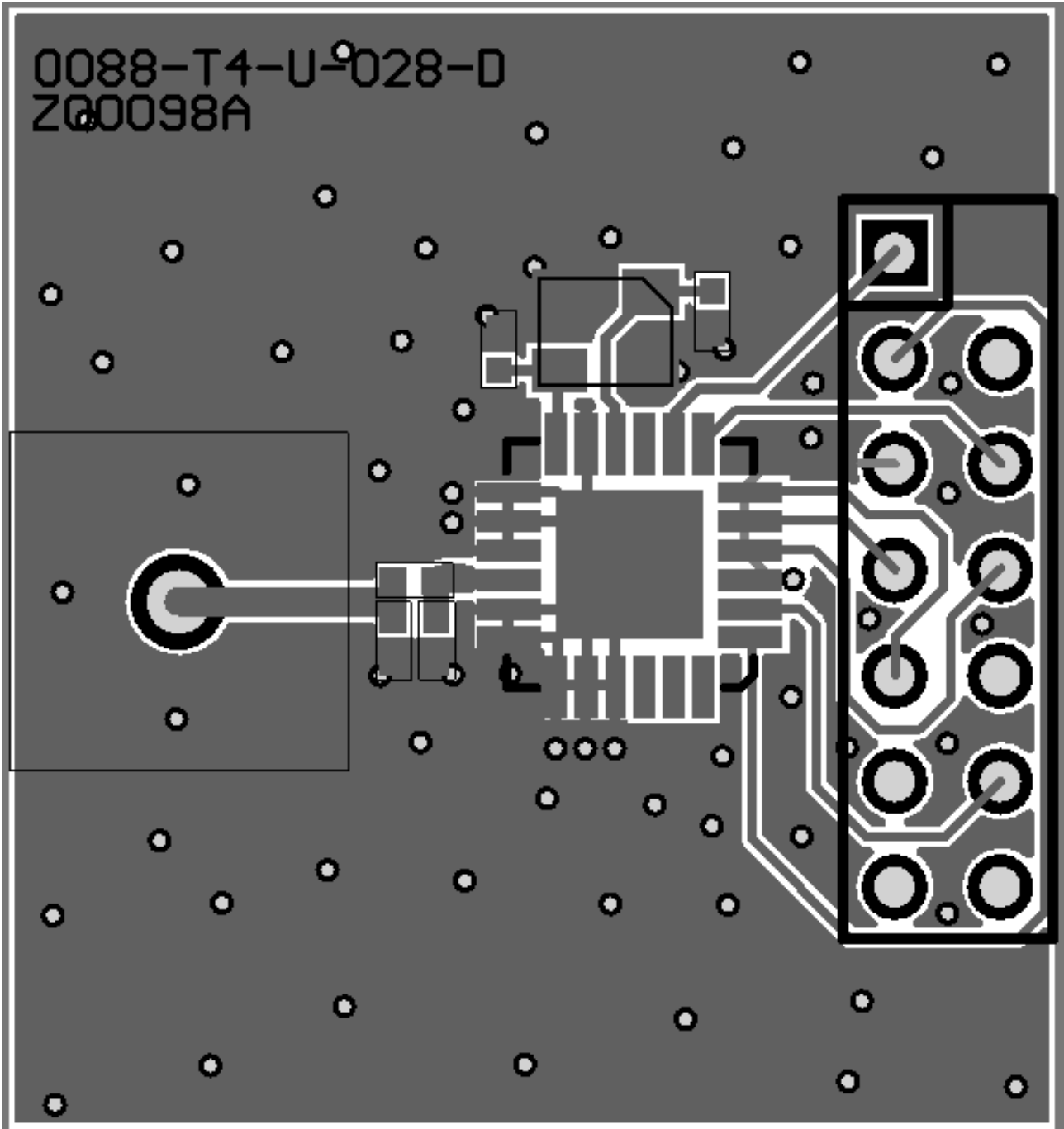
#### A. schematic



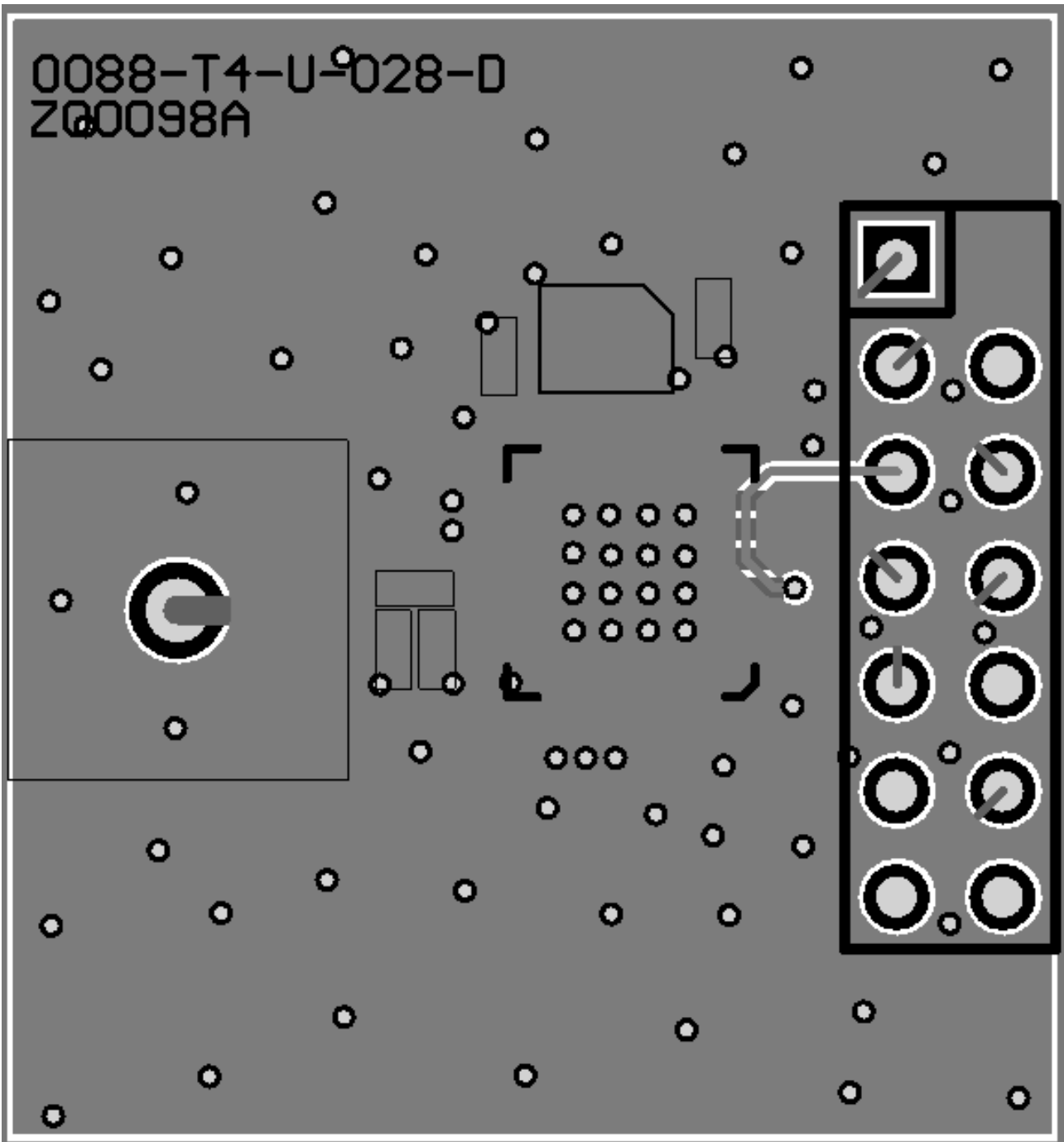
**B. 2-Layer PCB Layout**



C. PCB Top Layer



D. Bottom Layer



## Revision History

Revision	Date	Description of Change
0.0	2011/7/11	Initial release.

---

## Contact UBEC:

### *Headquarters*

Address: 6F-1, No. 192, Dongguang Rd., Hsinchu, 300 Taiwan

Tel: +886-3-5729898

Fax: +886-3-5718599

Website: <http://www.ubec.com.tw>

### *Sales Services*

Tel: +886-3-5729898

Fax: +886-3-5718599

E-mail: [sales@ubec.com.tw](mailto:sales@ubec.com.tw)

### *FAE Services*

Tel: +886-3-5729898

Fax: +886-3-5718599

E-mail: [fae@ubec.com.tw](mailto:fae@ubec.com.tw)

## DISCLAIMER

TO THE BEST KNOWLEDGE OF THE UNIBAND ELECTRONIC CORPORATION (UBEC), THIS DOCUMENT IS ADEQUATE FOR ITS INTENDED PURPOSES. UBEC MAKES NO WARRANTY OF ANY KIND WITH REGARD TO ITS COMPLETENESS AND ACCURACY. UBEC EXPRESSLY DISCLAIMS ANY AND ALL OTHER WARRANTIES, WHETHER EXPRESSED, IMPLIED, OR STATUTORY INCLUDING WITHOUT LIMITATION WARRANTIES OF TITLE, MERCHANTABILITY, NON-INFRINGEMENT, AND FITNESS FOR A PARTICULAR PURPOSE, WHETHER ARISING IN LAW, CUSTOM, CONDUCT, OR OTHERWISE.