Author: Joseph Alameda

*Application Note for Infineon BGT24MTR11 PCB Breakout Board with 4x4 Patch Array Antenna*

This application note is intended for individuals planning on designing a PCB breakout board for the Infineon BGT24MTR11 24GHz MMIC. The features of the Infineon BGT24MTR11 are :

1. Fully integrated low phase noise VCO
2. Switchable prescalar with 1.5GHz and 23kHz output
3. On chip power and temperature sensors
4. Gilbert based homodyne quadrature multiplier
5. Single ended rf and lo terminal
6. Low noise figure NFSSB 12dB
7. High conversion gain:26dB
8. Single supply voltage 3.3V
9. Low power consumption 500mW
10. 5.3x4.3mm size

The issues taken into account for this application note are exclusively using Rogers RO4350b PCB material. This is a very light material and has excellent electrical qualities from 8 to 40 GHz and university students can get three free sheets from Rogers by simply following this link to their page and filling out the information requested:

<http://rogerscorp.force.com/samples/samples_university>

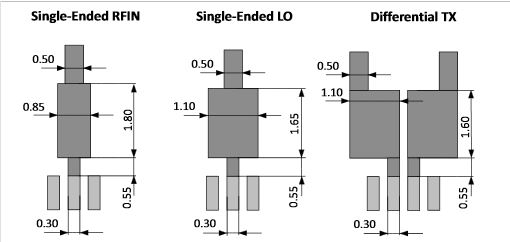
The first action of the designer is to thoroughly read the Infineon BGT24MTR11 datasheet and become familiarized with the requirements of the system. Here, the designer will find a block diagram of the signal processing of the chip, the electrical characteristics and the impedance matching information. Here is a link to the datasheet:

<http://www.infineon.com/dgdl/BGT24MTR11_v3.1.pdf?fileId=db3a304339dcf4b10139def491930214>

The next consideration is the placement and the impedance matching of the antenna. If the antenna is well designed, it should have an input impedance of 50 ohms. This application note will focus on implementing the design using Cadence Allegro PCB editor, although, any PCB CAD software may be used to design the breakout board. The reason for the use of Cadence Allegro is due to the integration with ANSYS HFSS.

**Impedance Matching**

Impedance matching is one of the most important considerations when designing the breakout PCB, especially for an MMIC operating at 24 GHz. There are approximately four ports that need to be impedance matched: Tx,Txx,Rx and the single ended LO. both Txx and the single ended LO will be terminated to 50 ohm resistors because we do not have room for an additional patch array for the differential output. The material used should be taken into account when calculating the impedance matching networks. Infineon includes the impedance transformers for the Infineon BGT24MTR11 and are shown below:

**INFINEON BGT24MTR11 Impedance Matching Networks**

These dimensions are for Rogers RO4350b with 2oz copper cladding, but the impedance mismatch from using 1oz copper is negligible. All dimension sizes given are in millimeters. These images can be found with additional information on the datasheet for the Infineon BGT24MTR11. It should also be addressed that the information provided by Infineon in the data sheet refers to a three layer PCB. This is not necessary, and a two layer board will work just as well.

Changing the trace width to match the dimensions shown above is a tedious process, especially trying to create an offset impedance transformer like the one shown for the differential transmit output. From the authors personal experience, it is easier to create these odd shapes in Cadence Allegro PCB editor than in Eagle CAD by simply clicking on the Dynamic Shapes tab at the top of the tool bar. An excellent tutorial on many of the features of Cadence Allegro can be found by clicking on this link:

<https://www.csee.umbc.edu/~robucci/class_wiki/index.cgi/Allegro%20PCB%20Tutorial#Dynamic_Shapes_.28for_Ground.2BAC8-Power_Planes.29>

Becoming familiarized with Cadence Allegro takes time, but it will be well worth it. Cadence Allegro is widely used by professionals, and makes importing PCB footprints very easy. Your TA is also an excellent resource for learning all of the features of Cadence. The next step in designing the breakout board is taking the Infineon MMIC’s footprint into consideration.

**MMIC PCB Footprint**

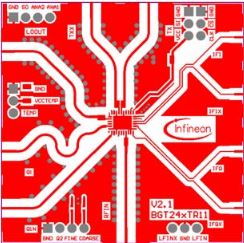
Designing a footprint for an integrated circuit is tedious and nerve wracking, considering the cost of fabricating the wrong PCB, and the delay in the design process and testing. Fortunately, a wide variety of IC manufacturers provide a PCB footprint of their products for PCB designers to download directly from the website. Infineon is one of these manufacturers. It is strongly advised to follow this link:

<http://www.infineon.com/cms/en/product/rf-and-wireless-control/mm-wave-mmic/24-ghz-radar/BGT24MTR11/productType.html?productType=db3a30443ff7943901400b1ba90516fa>

and scroll to the bottom of the page, where you will find a number of download links titled: PCB Design Data. From this spot on the page, you can download the footprint for three different PCB CAD tools: Eagle CAD, Cadence Allegro and Mentor. This significantly reduces the amount of time spent designing the breakout PCB by virtually removing the design of the IC footprint.

**Noise Considerations**

One of the most important considerations when designing an RF board is taking into account noise. A good designer will do their best to keep it far from the system. In the case of the Infineon MMIC, the SPI data is the greatest source of noise. The SPI (serial peripheral interface) is used to initialize the power amplifier for the Infineon and to adjust it from 11dBm to 0 dBm. This data is passed at a variable clock rate (chosen by the designer) and has three different connections: SPI clock, data and chip select. This data comes in pulses of 3.3V and was the main source of noise in our first quarters radar. It is highly recommended that all three SPI traces DO NOT cross underneath either the Tx or the Rx trace. This will result in severe noise problems and hinder the operation of the Infineon. The power line should also not cross under the Tx and Rx trace. Another consideration to keep in mind is the fact that the Infineon chip only needs to be initialized one time. This means that the user does not need to keep sending data to the chip, but rather only initialize it once. It is also advised by the author to not use a DAC that uses SPI for function generation in FMCW mode, but rather to use an analog function generator. SPI is a system that is attached to all different devices using this method of data transfer and causes considerable noise throughout the system.

**Ground Plane**

To the right is pictured an image of the recommended PCB layout for the Infineon BGT24MTR11. Upon inspection the distance between the traces for the Tx,Txx Rx and LO port and the ground plane are very wide. This is to avoid the fringing effects of the microstrip line if the top ground plane is too close. The designs given for the dimensions of the impedance transformers are for a microstrip line NOT for a coplanar waveguide. To avoid this problem all together, the designer may choose to remove the top plane ground plane entirely and ground everything through VIAS to the bottom ground plane. The author chose this method and it worked well.

**Conclusion**

When designing the PCB for the Infineon it is suggested to keep in mind the overall system layout. The PCB view above is designed to send a signal through an SMA, which is not ideal for power consumption. Keeping the layout of the system in mind while designing the PCB is highly recommended. This entails visualising which sides of the PCB the signals will be coming and going from and trying to route all traces while keeping noise considerations in mind. If everything is done correctly, the designer should have a well designed, low noise breakout board with an excellent low power 24 GHz MMIC Transceiver!