TI mmWave Radar sensor RF PCB Design, Manufacturing and Validation Guide

Chethan Kumar Y.B., Anil Kumar KV, and Randy Rosales

ABSTRACT

This application report helps TI mmWave Radar sensor designers navigate the series of tasks and key concerns when designing, manufacturing and validating a new mmWave sensor board. This document is only concerned with the RF portions of the design. It is beneficial for PCB designers that do not have experience with RF PCB design at mmWave frequencies. This document is applicable to sensor designs using IWR/ AWR mmWave Radar chips.

Contents

List of Figures

List of Tables

Trademarks

Teflon is a registered trademark of Chemours.
All other trademarks are the property of their respective owners.

1 Introduction

The goal of this application report is to familiarize you with the hardware design aspects of the radar system using TI Radar chips. However, depending on the application, sensor designers need to decide for themselves how many of these steps and to what extent they are followed based on time available for design/simulation, lab test time and sensor validation required.

Section 2 covers the aspects of PCB manufacturing concerns. This includes the selection of the RF PCB material, discussion of the CAD to CAM RF design documentation, and how to evaluate PCB fabricators for RF fabrication quality.

Section 3 covers the RF PCB design. Also, simulation flow and key concerns are discussed, which includes a brief overview of the key antenna design requirements and how they map to the radar equation budgets.

Section 4 covers thermal considerations, followed by enclosure and Radome design aspects.

2 RF PCB Selection and Fabrication

PCB fabrication concerns from the RF perspective are covered in this section. This includes a discussion of CAD to CAM RF design documentation and how to evaluate PCB fabricators for RF fabrication quality. The goal here is to describe the key points to bring up and align on with a selected PCB fabricator to achieve the first pass success when fabricating the mmWave PCB.

2.1 PCB RF Design

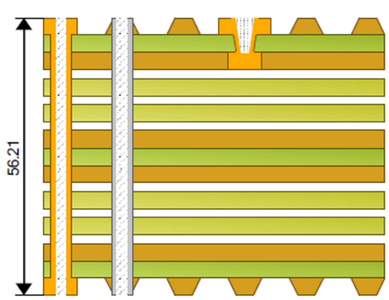
This section discusses a typical PCB to Ball Grid Array (BGA) transition and Grounded Coplanar Waveguide (GCPW) structures using the example of the AWR1642 EVM board. The BGA to PCB transition in the EVM board is compatible with all of the 77 GHz mmWave sensor devices.

The above mentioned board was designed with RO4835 + FR4 hybrid stackup. It could also be designed with a stackup using RO3003 or any other RF-friendly substrate.

The choice of the stackup is based on the tradeoff between electrical performance and substrate availability and manufacturing yield. For example, the Rogers RO3003 stackup (along with rolled copper foil) yields lower loss and better phase repeatability. Rogers RO4835 LoPro, on the other hand, has better manufacturing yield while it has slightly inferior electrical properties.

2.1.1 PCB Stackup

This section discusses the PCB stackup of the sensor board. As an example, the AWR1642 EVM BoosterPack board uses a hybrid stackup. Rogers RO4835 LoPro core is used between metal layers 1 and 2. The remaining layers 2 through 6 are etched on FR4 core and prepreg substrates. The stackup is shown in Figure 1.



Layer	Stack up	Description	Type	Base Thickness	Processed Thickness	εr	Copper Coverage	Mask Thickness
1				0.689	2.067		100.000	
		Rogers 4835 4mil coreH/1 Low Pro	Rogers 4835	4.000	4.000	3.480		
2				1.260	1.260		73.000	
		Iteq IT180A Prepreg 1080	Dielectric	4.195	2.830	3.700		
		Iteq IT180A Prepreg 1080	Dielectric	4.195	2.830	3.700		
3				1.260	1.260		69.000	
		Iteq IT180A 28 mil core 1/1	FR4	28.000	28.000	4.280		
4				1.260	1.260		48.000	
		Iteq IT180A Prepreg 1080	Dielectric	4.195	2.691	3.700		
		Iteq IT180A Prepreg 1080	Dielectric	4.195	2.691	3.700		
5				1.260	1.260		72.000	
		Iteq IT180A 4 mil core 1/H	FR4	4.000	4.000	3.790		
6				0.689	2.067		100.000	

Figure 1. AWR1642 EVM Board Stackup

For RO3003 based boards, the stackup is the same as Figure 1 except that the RO4835 LoPro 4mil substrate is replaced with RO3003 5mil substrate.

2.1.2 Package to PCB transition

To get an optimal match between the device RF ball and PCB trace, the package to PCB transition layout needs to be designed appropriately. TI has built Rogers-based EVM boards and recommends the customer who uses Rogers materials, to use TI's recommended transition. This transition allows the package to match to a 50 Ω transmission line at the edge of the package (approximately 1.3 mm from the center of the signal pad). If the customer chooses to use a different PCB substrate, the package to PCB transition needs to be re-designed, which is beyond the scope of this application report.

Example:

Layout for an optimized AWR1642 BGA package to GCPW transition is shown below. These dimensions are for stackups based on Rogers PCB substrates.

The 77 GHz TX/RX signal pads and signal traces are surrounded by continuous reference ground plane on L1 and ground reference on L2. Three ground-stitching (L1-L2 ground) micro-vias are via-in-pad with the surrounding AWR1642 BGA ground pads. There are four additional ground-stitching micro-vias that are placed just outside the ground plane keep out radius of the signal pad. There is a circular ground plane cutout placed underneath the RF signal pad on L2 that provides right impedance matching.

Together, with the anti-pad separation, these radial ground vias and L2 ground plane cutout, smoothly transition the package BGA impedance to the GCPW PCB impedance around the AWR1642 RF operating bands.

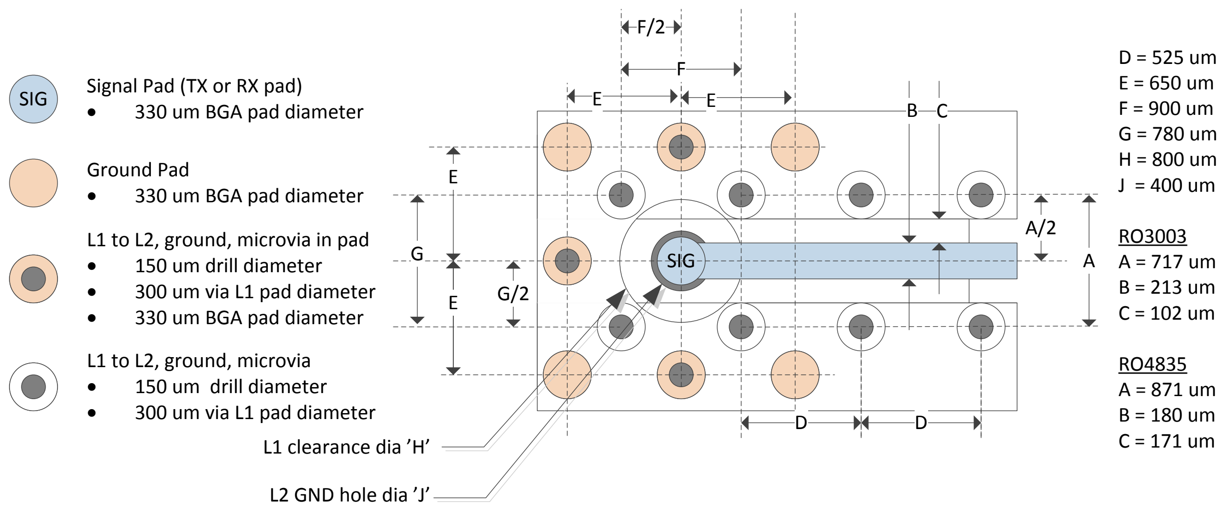


Figure 2. AWR1642 BGA to PCB Transition and GCPW Dimensions (drawing not to scale)

Note that dimensions A, B and C are different for the RO3003 stackup and the RO4835 LoPro stackup as shown in Figure 2.

2.2 Assessing PCB Fabricator Experience With RF Substrates

Designers should discuss with their PCB vendor the vendor's experience with fabricating PCB with high-frequency substrates. PCB substrate fabrication documentation covers material storage, handling, and processing techniques. All of these recommendations must be followed to achieve consistent performance when utilizing these materials. For details of fabrication using Rogers materials, see [1] and [2].

Sequential lamination of RF and non-RF substrate core and pre-preg material is typically required for completing RF designs such as the BoosterPack EVM. Designers should discuss with their PCB vendor the vendor's experience and capabilities when fabricating mixed material hybrid stackups. Different core and pre-preg materials typically have different curing requirements and procedures and may not always be compatible.

2.3 Material Properties and Manufacturing Tolerances Affecting Critical RF Performance

RF signal paths exhibit high sensitivity to small geometry changes such as:

- Substrate thickness
- Metal thickness
- Metal roughness
- Plating
- Via placement tolerance
- Etch tolerances (LDI vs. LPI masks)
- Air gap tolerances
- Solder-mask tolerance (LDI vs. LPI accuracy)
- Sequential stack-up layer registration
- Peel strength vs plating height

Substrate thickness directly determines performance of the RF structures. RO4835 LoPro and RO3003 substrates should maintain their designed thicknesses as received from Rogers. However, improper handling or fabrication steps can damage these substrates causing delamination and other adverse effects that will severely impair any RF structure performance.

Overall etch tolerances must be controlled so that the line widths, air gaps and planar antenna structures stay close to their designed dimensions. TI recommends using the Laser Direct Imaging (LDI) etch-mask over the more common Liquid Photoimageable (LPI) etch-mask because LDI enables fabrication with tighter tolerances.

Solder-mask has different dielectric properties compared to the RF substrate and the free-space surrounding the PCB. Solder-mask should typically be avoided over the RF transmission lines and antenna. In the case of solder-mask near the RF BGA, it is critical that the solder-mask registration and thickness must be tightly controlled. Changes in thickness or registration can have an effect on variability of RF performance from PCB to PCB.

Plating affects RF characteristics. Most common plating used in industry, Electroless Nickel-Immersion Gold (ENIG), is not a good choice for mmWave boards due to its high losses. TI EVM uses immersion silver plating that has lesser loss, comparatively. However, immersion silver is susceptible to oxidation on prolonged exposure to air. This oxidation does not impact the RF performance. However, proper storage of the PCB is recommended to reduce the oxidation. Over and under plating of top-layer copper can result in phase and loss/reflection variations.

Number of types of vias determines the PCB process complexity. Higher number of types of vias typically causes higher processing steps, which uses sequential lamination and can cause via registration error. They also increase the PCB cost and lowers yield. Therefore, it is desirable to keep via types to a minimum. TI mmWave BoosterPack designs use two types of vias.

Multiple types of via processing also increases the plating thickness. Thicker plating on traces that have narrow widths causes the peel strength to be poorer. Thicker top layer geometry also has higher coupling between the antenna elements, which is undesirable. Therefore, it is recommended to keep the top layer metal plus plating thickness low. TI mmWave BoosterPack designs use a 0.5 oz base copper on the top layer as shown in Figure 1.

The copper lamination will be of difference kinds. Rolled copper offers the lowest surface roughness and the lowest loss. Electro-deposited (ED) copper has relatively higher roughness and loss.

The placement error on ground vias around the transmission line have detrimental impact on its characteristics such as its impedance and bandwidth. Typically, in manufacturing, the via placement errors can have different offsets in x and y-directions. Therefore, the impact on transmission lines or antenna structures that are aligned to x-axis and y-axis can exhibit different characteristics.

There are many via-s on the PCB that are either on the pads or close to pads. These are filled with non-conductive epoxy and capped with copper plating to ensure flat surface.

2.3.1 Determining Absolute Tolerance Limits

Absolute tolerance limits on each dimension and placement can only be derived from margin studies using RF simulators and EM theory. The process involves sweeping various parameters, through tolerance limit specified by the PCB manufacturer and determining how the parameter changes effects the performance of the structure. For example, designers can simulate with different air gaps, via placement distances, line widths and see the resulting change in GCPW impedance or antenna gain or directionality. Such studies are beyond the scope of this application report.

2.4 RF Critical CAD to CAM Documentation and Verification

Designers are encouraged to clearly document the areas of the PCB that are RF design critical along with the intended design dimensions for each of these locations. Controlled impedance trace dimensions and stack-up thicknesses for high-speed digital signals are typically dictated and verified by a PCB fabricator. However, RF design dimensions should be dictated completely by the PCB designer and verified after fabrication.

In the case of these mmWave sensor designs, the areas around the RF signal BGA footprints, the RF signal transmission-lines and the antennas must be carefully drilled and etched. Ideally, the tooling error must be constrained to zero-mean error around the designed dimension. Typical PCB fabrication error results in a low variance skew in one direction of the tolerance window. PCB designers need to discuss methods with their fabricator for bringing this skew as close as possible to the designed dimensions.

It is recommended that PCB designers explicitly ask for a small sample run of PCB to be used for process inspection purposes. Any problems meeting critical RF design dimensions can be dealt with before proceeding to larger volume production. This process can be repeated until the zero-mean error between fabricated and designed dimensions are achieved.

A report of critical RF design dimensions should be presented to the PCB fabricator as part of the the PCB CAD and CAM board design documents and files. The PCB fabricator should be explicitly asked to verify what the expected tolerances are going to be for each of the critical dimensions. It is recommended to provide the RF coupon structure on the PCB/Panel. This helps in assessment of material property, fabrication parameters etching tolerances, manufacturability variations in impedance, and so forth.

For example, AWR1642 BoosterPack EVM has transmission line coupon structures for probe as shown in Figure 3.

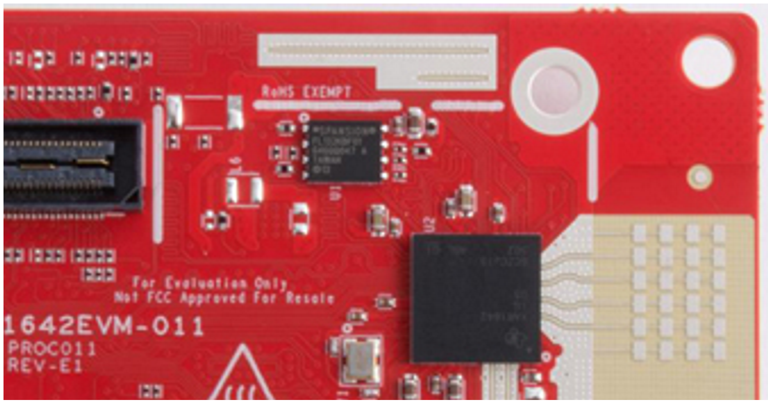


Figure 3. Example of Coupon Structure on the AWR1642 Booster Pack EVM

3 Integrating the Antenna

This section is a brief overview of key antenna design requirements and how they map to radar equation budgets and FMCW radar processing. This also includes a discussion on how to best re-use the BoosterPack EVM designs with custom etched antenna.

Knowledge of basic antenna metrics is necessary for an mmWave sensor designer to understand the antenna performance requirements needed for their sensor.

3.1 Radar Equation and Link Budgeting

All FMCW mmWave sensors work on the underlying principle that a transmitting antenna can radiate out an RF "Chirp" signal and an associated receiver antenna can detect the resulting chirp echo reflected from a target. The received chirp reflection is mixed with the transmitted chirp and the resulting IF signal is sampled to determine range, angle and velocity of the target. *Programming Chirp Parameters in TI Radar Devices* [4] covers the process of how to combine with proper chirp configurations.

If too little energy is radiated from the transmit antenna, or too little energy is incident to the target, or too little energy is incident to the receive antenna, the sensor ADC will not have enough IF amplitude to resolve the target above the noise floor and target detection will fail. Significant signal to noise ratio, or SNR, is required to resolve the target in the resulting IF FFT spectrum.

All of these parameters are combined in the "Radar Equation" to help designers perform a radio link budget analysis. For details of the link budget analysis, see [4].

The Radar equation that gives the range of a target is shown in Equation 1.

$$\text{Range}_{MAX} = \sqrt[4]{\frac{P_{TX} \times D_{RX} \times D_{TX} \times c^2 \times \sigma_{Target} \times N_{Chirps} \times T_R}{f_C^2 \times (4\pi)^3 \times kT \times NF \times SNR_{detect}}} \tag{1}$$

where:

- P_{TX}: The transmitted output power, incident to the antenna in Watts.
- D_{TX},D_{RX}: The transmit and receive antenna directionality, respectively. This takes into account antenna efficiency as well as directional gain – unit-less gain.
- c: Speed of light in free-space in meters/second.
- σ_{Target}: The radar cross section (RCS), which is a unit-less gain factor relating incident power to reflected power of a target object.
- f_C: Means frequency of the chirp ramp in Hz.
- N_{Chirps}: Number of chirps in a chirp frame
- T_R: Chirp ramp time in seconds
- k: Boltzman's constant – 1.38 x 10^{-23} J/K
- T: Temperature in Kelvin
- NF: Is the Noise Factor of the receiver path. This coefficient takes into account the noise introduced by the stages of the receive signal path like the low-noise amplifier (LNA), low-pass and high-pass filtering and the ADC sampling.
- SNR_{detect}: Minimum signal to noise ratio for detection of the target. This is expressed in absolute ratio.

In terms of antenna and transmission-line design, there are only a few terms that can be optimized for the radar equation: P_{TX}, D_{TX} and D_{RX}. All of the other terms are determined by the chirp parameters, target, environment or mmWave sensor device native performance.

The power delivered to the antenna, P_{TX}, is a function of programmed transmit power backoff, PCB transmission-line losses and transmission-line to antenna reflections and BGA to transmission-line reflections. Maximizing P_{TX} is the goal. Therefore, minimizing losses and reflections along the RF path to the antenna and maximizing programmed transmit power is important. The mean chirp frequency, f_C, is a function of chirp configuration, but it should be taken into account during the antenna design phase if possible so that the bandwidth of the antenna can be optimized around the intended chirp bandwidth.

Directivity D_{TX} and D_{RX} are entirely a function of the antenna design and transmission-line design.

3.2 Custom Antenna Design Options

Depending on the amount of experience with the mmWave RF design, mmWave sensor designers have a few options when it comes to acquiring application-specific antennas. Starting with the requirements derived from the sections above and constraints of the TI supported RF substrate stack-ups, designers can:

- Create their own antenna
- Have a third-party RF design firm create an antenna

- Use one of the TI provided reference antenna designs

If a designer is familiar enough with the antenna design concepts and simulation tools, then the best way to start a custom antenna design is to reference the TI mmWave Radar BoosterPack EVM boards. These boards can serve as a reference RF layout design that can be extended to work with a custom designed antenna.

The general flow is to assume the same layer-1 RF substrate of the stackup and then completely re-use the BGA to PCB transition footprint and GCPW transmission-line RF fan-out. The custom antenna is then designed and a transmission-line feed path can be laid out to interface the antenna to the GCPW fan-out for best impedance and phase match to the antenna. The recommended process for third-party designers is the same.

If one of the TI provided reference antenna meets the design requirements, then no re-design is necessary and these can be integrated into the new design.

4 Thermal Design

This section discusses the thermal aspects of the system design. The need for effective heat dissipation is to keep the die temperature within the operational and reliable limits. Some of the various ways to manage heat dissipation from the Radar device are listed below:

- Limiting the chirp duty cycle and limiting the power dissipation in the device
- Using multiple copper layers in the board with large GND planes and having enough number of thermal vias below the device directly connecting the device GND balls to the PCB GND planes
- Using heat-sink that touches the device top or bottom of the board, directly below the device

The thermal parameters of the device are provided in the device-specific data sheet. For more details on the definition of these parameters and their use, see [3].

Thermal simulation of the sensor system with heat sink or enclosures is recommended during the mechanical design of the system.

Heat-sink option: A very effective way to conduct heat away from device is to use heat-sink. The heat-sink can be either on device side (top heat-sink) or on side opposite to the device (bottom heat-sink) or both. In a simple form, the top heat-sink can be a sufficiently large metal piece that makes good thermal contact with the device top. If the antenna is etched on the same side as the device, care should be taken to see that the metal heat-sink does not distort the antenna characteristics. The bottom heat-sink is also a piece of metal or protrusion from the enclosure that touches the board directly below the device on the side of the PCB opposite to the device. While using bottom heat-sink, care should be taken to see that the metal does not touch any capacitors or other components that may cause electrical short.

5 Enclosures and Radome

Radome is a cover or enclosure in order to protect Radar antennas from environmental influences. It functions as a structural and weatherproof enclosure to the antenna in such a manner that it has least interference with the transmission from it. It is derived from the expression Radar and Dome. Covers will have some influence on the shape of radiation pattern or Field of view and achievable maximum distance. Radars can see through plastic and glass of any color. While designing housing, pay attention to the aspects being discussed in Section 5.1.1 and Section 5.1.2.

The objective of an efficient Radome design is to reduce reflections at its surface and transmit the signal with minimum loss and beam distortion. For a general-purpose enclosure that covers the radiating side of the sensor, the material should have a uniform thickness and must also have a good surface smoothness. In some cases, a Radome could be constructed as a lens that alters the beam characteristics intentionally. Such Radoms/lens need to be designed using electro-magnetic simulation tools in conjunction with the antenna.

5.1 Guideline for Radome Design

Some key important parameters in a typical Radome design are its wall thickness and its distance to the antenna.

5.1.1 Consideration for Radome Wall Thickness

Thickness of the Radome plays a key role in arriving at the optimum performance of the mmWave sensor. Wavelength in the Radome becomes shorter in the material than in free air. The wavelength in the material is a function of its dielectric constant. The goal is to make the wall thickness equal to the integer multiple of the wavelength in the material. This is to make sure that the Radome becomes nearly transparent for the mmWave signals.

$$t = n * \lambda_m / 2 \quad (2)$$

where:

$$\lambda_m = \frac{C}{f \times \sqrt{\varepsilon_r}} \quad (3)$$

where:

- *t*: thickness of Radome wall
- *n*: 1,2,3…
- λ_m: wavelength in Radome material
- *c*: speed of light
- *f* : mean carrier frequency used
- ε_r: relative permittivity

Material with lower Dk and Df (dielectric constant and loss tangent) are recommended. Typical materials used in Radome are Polycarbonate, Teflon® (PTFE), Polystyrene, and so forth. Typically, with Radome and Antenna, simulations are done to see there is very little degradation in the Radiation pattern.

5.1.2 Consideration for Antenna to Radome Distance

The optimal distance between the antenna and the Radome helps to minimize the effects of reflections caused by the Radome. These effects become minimal if the waves returned at the antenna are in phase with the transmitted waves.

$$D = n.\lambda_0 / 2 \quad (4)$$

where:

- *n*: 1,2,3…
- *D*: optimal distance between Radome and Antenna
- λ_0: wavelength in air

6 References

1. *RO3000® and RO3200™ Series High Frequency Laminates*
2. *RO4003C™/RO4350B™/RO4835™ Laminates Circuit Processing Guidelines*
3. *Semiconductor and IC Package Thermal Metrics*
4. *Programming Chirp Parameters in TI Radar Devices*

IMPORTANT NOTICE FOR TI DESIGN INFORMATION AND RESOURCES

Texas Instruments Incorporated ('TI") technical, application or other design advice, services or information, including, but not limited to, reference designs and materials relating to evaluation modules, (collectively, "TI Resources") are intended to assist designers who are developing applications that incorporate TI products; by downloading, accessing or using any particular TI Resource in any way, you (individually or, if you are acting on behalf of a company, your company) agree to use it solely for this purpose and subject to the terms of this Notice.

TI's provision of TI Resources does not expand or otherwise alter TI's applicable published warranties or warranty disclaimers for TI products, and no additional obligations or liabilities arise from TI providing such TI Resources. TI reserves the right to make corrections, enhancements, improvements and other changes to its TI Resources.

You understand and agree that you remain responsible for using your independent analysis, evaluation and judgment in designing your applications and that you have full and exclusive responsibility to assure the safety of your applications and compliance of your applications (and of all TI products used in or for your applications) with all applicable regulations, laws and other applicable requirements. You represent that, with respect to your applications, you have all the necessary expertise to create and implement safeguards that (1) anticipate dangerous consequences of failures, (2) monitor failures and their consequences, and (3) lessen the likelihood of failures that might cause harm and take appropriate actions. You agree that prior to using or distributing any applications that include TI products, you will thoroughly test such applications and the functionality of such TI products as used in such applications. TI has not conducted any testing other than that specifically described in the published documentation for a particular TI Resource.

You are authorized to use, copy and modify any individual TI Resource only in connection with the development of applications that include the TI product(s) identified in such TI Resource. NO OTHER LICENSE, EXPRESS OR IMPLIED, BY ESTOPPEL OR OTHERWISE TO ANY OTHER TI INTELLECTUAL PROPERTY RIGHT, AND NO LICENSE TO ANY TECHNOLOGY OR INTELLECTUAL PROPERTY RIGHT OF TI OR ANY THIRD PARTY IS GRANTED HEREIN, including but not limited to any patent right, copyright, mask work right, or other intellectual property right relating to any combination, machine, or process in which TI products or services are used. Information regarding or referencing third-party products or services does not constitute a license to use such products or services, or a warranty or endorsement thereof. Use of TI Resources may require a license from a third party under the patents or other intellectual property of the third party, or a license from TI under the patents or other intellectual property of TI.

TI RESOURCES ARE PROVIDED "AS IS" AND WITH ALL FAULTS. TI DISCLAIMS ALL OTHER WARRANTIES OR REPRESENTATIONS, EXPRESS OR IMPLIED, REGARDING TI RESOURCES OR USE THEREOF, INCLUDING BUT NOT LIMITED TO ACCURACY OR COMPLETENESS, TITLE, ANY EPIDEMIC FAILURE WARRANTY AND ANY IMPLIED WARRANTIES OF MERCHANTABILITY, FITNESS FOR A PARTICULAR PURPOSE, AND NON-INFRINGEMENT OF ANY THIRD PARTY INTELLECTUAL PROPERTY RIGHTS.

TI SHALL NOT BE LIABLE FOR AND SHALL NOT DEFEND OR INDEMNIFY YOU AGAINST ANY CLAIM, INCLUDING BUT NOT LIMITED TO ANY INFRINGEMENT CLAIM THAT RELATES TO OR IS BASED ON ANY COMBINATION OF PRODUCTS EVEN IF DESCRIBED IN TI RESOURCES OR OTHERWISE. IN NO EVENT SHALL TI BE LIABLE FOR ANY ACTUAL, DIRECT, SPECIAL, COLLATERAL, INDIRECT, PUNITIVE, INCIDENTAL, CONSEQUENTIAL OR EXEMPLARY DAMAGES IN CONNECTION WITH OR ARISING OUT OF TI RESOURCES OR USE THEREOF, AND REGARDLESS OF WHETHER TI HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

You agree to fully indemnify TI and its representatives against any damages, costs, losses, and/or liabilities arising out of your non-compliance with the terms and provisions of this Notice.

This Notice applies to TI Resources. Additional terms apply to the use and purchase of certain types of materials, TI products and services. These include; without limitation, TI's standard terms for semiconductor products http://www.ti.com/sc/docs/stdterms.htm), evaluation modules, and samples (http://www.ti.com/sc/docs/sampterms.htm).

Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265