IWR/AWRx RF Hardware Design Guide

1 Summary

This application note presents two IWRx PCB as example RF reference designs. The designs discussed are proven, functional, designs and may be used to accelerate time to market when initially designing with the IWRx family of SoC.

The first board presented is the IWR1642 BoosterPack EVM which utilizes a Rogers RO4835 LoPro sequential lamination stackup. The second board presented is an internal IWR1642 GSG RF probe board which utilizes a Rogers RO3003 sequential lamination stackup.

For both designs the BGA to PCB transition and the grounded co-planar waveguide (GCPW) transmission line RF fan-out are described. This discussion covers the sequential lamination RF stackup material, copper thicknesses and finishes selected, via placement and solder mask and copper etching used to construct both the BGA landing and the grounded co-planar waveguide (GCPW) transmission line fan- out.

Throughout this application note, the reader is assumed to be familiar with PCB layout and fabrication terminology as well as some RF design terminology.

2 Table of Contents

3 Glossary

BGA – Ball Grid Array
EVM – Evaluation Module
SMD – Solder Mask Defined
NSMD – Non-Solder Mask Defined
GCPW – Grounded Co-Planar Waveguide Transmission-Line
GSG – Ground Signal Ground – Typical RF Probe Format
IWR/AWRx – Referring to the AWRx and IWRx automotive and industrial mmWave sensor family of devices

4 IWR/AWRx PCB RF Design

This section will describe the PCB to BGA transition and GCPW structures present on the IWR1642 EVM and AWR1642 RF Probe Board. The EVM and Probe Board BGA to PCB transition and GCPW design is compatible with all of the IWR/AWRx mmWave sensor devices.

Both of these example PCB are designed using a sequential lamination process with the top layer substrate composed of a higher performance RF substrate and the lower layers constructed of an FR4 glass weave substrate. The IWR1642 EVM was designed with a Rogers RO4835 LoPro based sequential lamination stackup. The AWR1642 probe board was designed with a Rogers RO3003 based sequential lamination stackup.

It should be noted that while the Layer 1 and Layer 2 RF structures (BGA transition, GCPW transmission line, and etched antennas) are all dependent on the RF substrate used, the inner and bottom layers are not. Designers should be able to use any general purpose, FR4 substrate for these inner and bottom layers as long as the fabrication process is compatible with that of the RF substrate. All power integrity and signal integrity of the various IWR/AWRx peripherals must also be maintained.

TI recommends that PCB designers verify the RF substrate, sequential lamination and HDI production capabilities and all design tolerances with their chosen PCB fabrication vendor. This will help ensure first pass success of these types of sequential stackup, HDI, RF PCB. This includes verifying core and prepreg storage and curing along with drill, mill and etching accuracy and method.

4.1 RO4835 LoPro Sequential Stackup Example - IWR1642 EVM Boosterpack

Although the IWR1642 EVM Boosterpack RF design was specifically used with the IWR1642 device, due to the package similarities, the RF signal PCB footprint and GCPW transmission lines design is applicable across the AWR16x/14x and IWR16x/14x family of devices when using the RO3003 sequential stackup described here. The IWR1443 EVM BoosterPack utilizes the same layer stack up, RF BGA footprint and GCPW transmission line design.

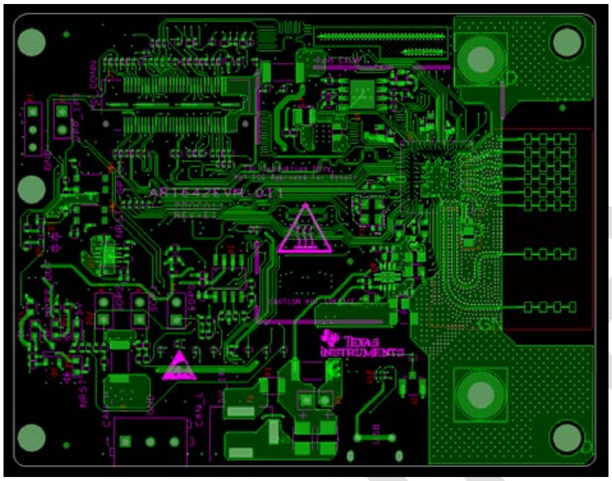


Figure 1 - IWR1642 Boosterpack EVM – Allegro PCB Top Layer View

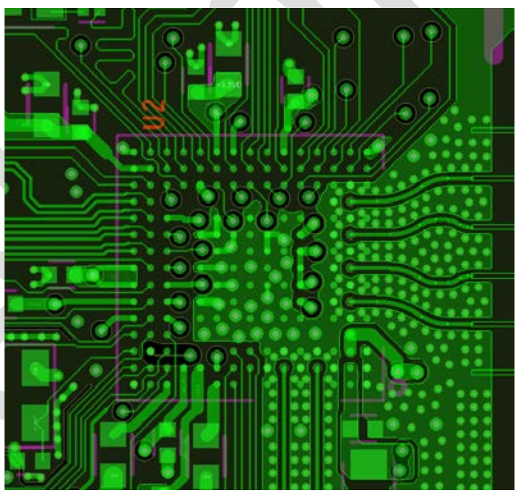


Figure 2 - IWR1642 Boosterpack EVM – Allegro PCB Top Layer View

The IWR1642 EVM Booster Pack design files (http://www.ti.com/tool/iwr1642boost) can be referenced for the full EVM schematic, bill of materials, layout, layer stackup, Allegro CAD and Gerber CAM files. The IWR1642 Hardware Design Checklist (http://www.ti.com/lit/zip/swrr151) can also be referenced.

4.1.1 PCB Stackup

RF structures on the IWR1642 EVM Boosterpack are implemented on routing layers 1 and 2. Rogers RO4835 LoPro core is used between metal layers 1 and 2. The remaining, general purpose, layers 3 through 6 are etched on an ITEQ IT 180A core and prepreg substrate. A sequential lamination fabrication method is then used to attach the RO4838 LoPro layers to the ITEQ IT 180A layers.

Along with the stackup substrate and copper base thicknesses, the top layer copper roughness and finish are also critical to achieving best RF performance. RO4835 Lo Pro (low profile) copper foil provides a very smooth copper profile. This decreased surface roughness reduces frequency dependent loss and phase variation.

Table 1 - RO4835/Iteq IT 180A Sequential Lamination Stackup for AWR1642 EVM

Number	Name	Material Type	Thickness (mils)	Dielectric Constant, ε_r	Loss Tangent, δ	Primary Uses	Notes
		Solder Mask					2
1	Top	Copper + Immersion Silver	2.067			RF routing, and top-layer impedance controlled lines	3, 4
		Rogers RO4835 LoPro	4.000	3.66	0.0037		
2	L2	Copper	1.260			Solid ground reference layer	
		Iteq IT180A Prepreg 1080	2.830	3.700	0.016		
		Iteq IT180A Prepreg 1080	2.830	3.700	0.016		
3	L3	Copper	1.260			Digital signal routing and ground reference	
		Iteq IT180A 28 mil core	28.000	4.280	0.016		
4	L4	Copper	1.260			Power and signal routing	
		Iteq IT180A Prepreg 1080	2.691	3.700	0.016		
		Iteq IT180A Prepreg 1080	2.691	3.700	0.016		
5	L5	Copper	1.260			Solid ground reference layer	
		Iteq IT180A core	4.000	3.790	0.016		
6	Bottom	Copper + Immersion Silver	2.067			Power and signal routing	
		Immersion Silver					
		Solder Mask					

Note 1: All thicknesses shown here are finished or processed thicknesses. Check with your PCB fabricator to determine final process thicknesses.
Note 2: Solder mask was not used on the RF portions of the design due to effect it would have on the GCPW and antenna RF performance. Only a small solder mask “damn” was included near the RF BGA.
Note 3: An immersion silver finish was applied to the RF areas of the PCB to prevent oxidation on the RF areas that would not receive solder mask. Total thickness of base ½ oz copper and immersion silver finish was intended to be 50 micron (1.968 mils). Check with your PCB fabricator to determine final process thicknesses.
Note 4: The 1/2 oz / in^2 (17 micron) electrodeposited copper foil option was selected for the 4835 LoPro core. Additional thickness was added as part of the sequential lamination process. Check with your PCB fabricator to determine final process thicknesses.

4.1.2 PCB to BGA transition and GCPW Transmission Line

The IWR1642 EVM includes an optimized BGA package to PCB landing transition based on the RO4835 LoPro stackup shown already shown above in Table 1.

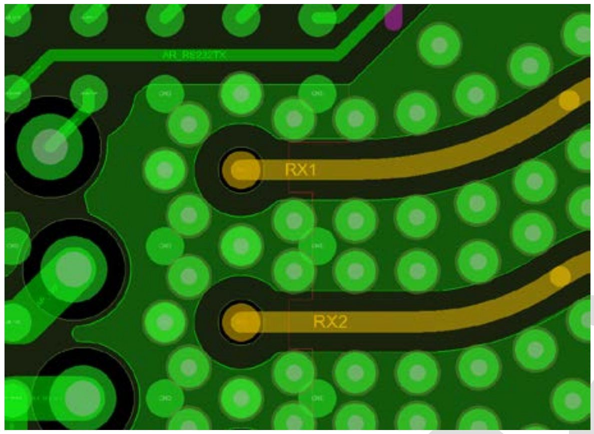


Figure 3 - IWR1642 Boosterpack EVM – Showing detail of the RX1 and RX2 BGA pads and GCPW fan-out

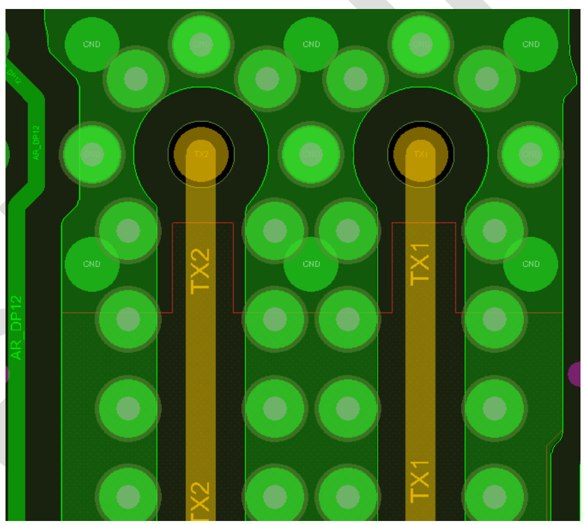


Figure 4 - IWR1642 Boosterpack EVM – Showing detail of the TX1 and TX2 BGA pads and GCPW fan-out

The goal of the RF BGA pad design is to create a controlled impedance transition between the package BGA, the solderballs and the to the GCPW transmission-lines. This allows for the most power, across the widest bandwidth, to be received from the RX antennas and sent to the TX antennas.

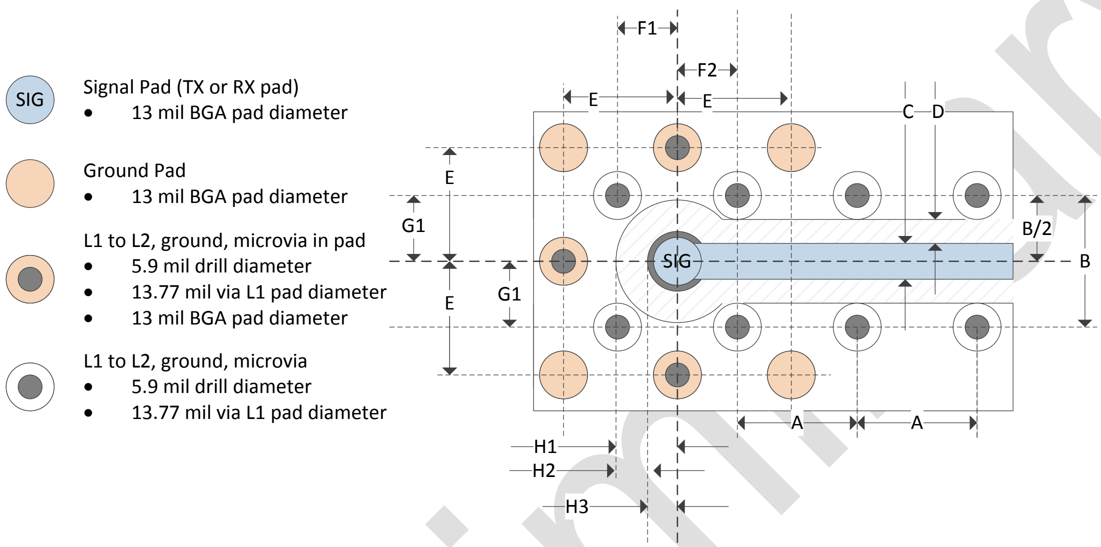
The TX and RX signal pads are non-solder mask define (NSMD) BGA pads[1] consisting of an opening in the solder mask slightly larger than the BGA pad itself.

[1] See [5] for details on SMD vs. NSMD defined BGA pads

The RF signal traces and pads are surrounded by reference ground etch on L1 and ground plane reference on L2. Three ground-stitching micro-vias (L1 to L2 ground net) are placed via-in-pad with the surrounding XWR16x BGA ground pads. There are 4 additional micro-vias (L1 to L2 ground net) stitching which are placed just outside ground plane keep out radius of the signal pad.

There is also a circular ground plane cutout on L2 placed underneath the RF signal pad which further minimizes pad capacitance. L3 is again a ground plane reference directly below the RF pads.

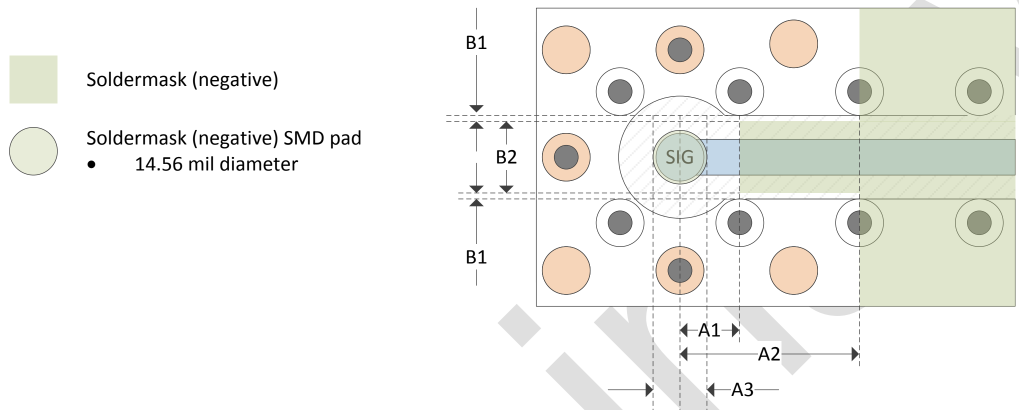
Together all of these structures create a package to PCB grounded co-planar waveguide (GCPW) transmission-line transition.



Measurement	Value	Unit
A	20.6700	mil
B	34.3100	mil
C	7.0800	mil
D	6.73000	mil
E	25.5900	mil
F1	17.7100	mil
F2	17.7200	mil
G1	15.3500	mil
H1	15.7488	mil
H2	9.2488	mil
H3	7.8800	mil

Figure 5 – IWR1642 RO4835 LoPro BGA to PCB Transition and GCPW Dimensions (Drawing Not to Scale)

Special care must be taken to ensure the accuracy and consistency of the solder mask near the RF BGA. Effects on the impedance of the RF lines due to solder mask variations near the RF BGA should be minimized by only including a minimal solder mask definition (solder-damn) near the BGA to prevent overflow during initial soldering. TI recommends utilization of laser direct imaging (LDI) solder mask methods over standard liquid photoimagible (LPI) solder mask due to the greater accuracy and consistency board to board that can be achieved with the LDI methods. TI does not recommend use of solder mask along the length of the GCPW transmission lines or etched antenna designs as this will significantly alter the performance of these structures.



Measurement	Value	Unit
A1	16.00	mil
A2	42.00	mil
A3	14.56	mil
B1	2.00	mil
B2	16.70	mil

Figure 6 – IWR1642 RO4835 LoPro BGA to PCB Transition Dimensions – Solder Mask Details (Drawing Not to Scale)

4.2 RO3003 Sequential Stackup Example - AWR1642 RF Probe Test Board

The AWR1642 RF Probe Test Board was specifically designed for the AWR1642 device. But due to the package similarities, the RF signal PCB footprint and GCPW transmission lines design is applicable across the full IWR/AWR1642, IWR/AWR1443 and AWR1243 family of devices when using the RO3003 sequential stackup described here.

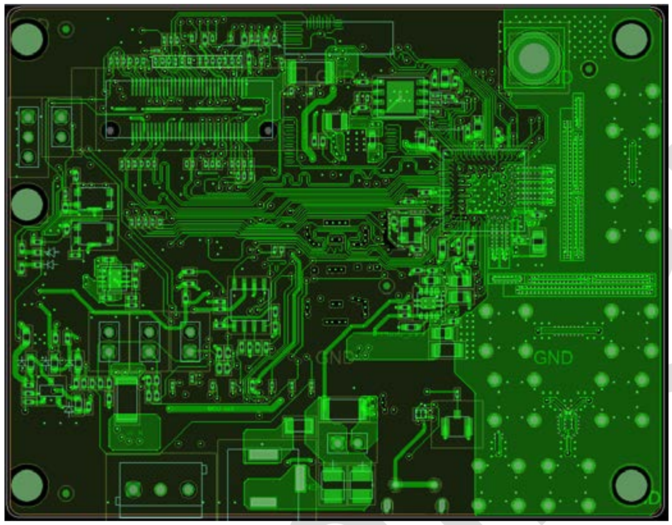


Figure 7 - IWR1642 RF Probe Test Board – Allegro PCB Top Layer View

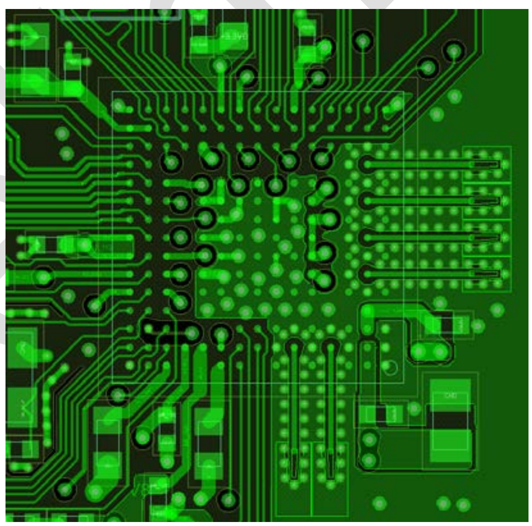


Figure 8 - IWR1642 RF Probe Test Board – RF BGA Detail

4.2.1 PCB Stackup

RF structures on the AWR1642 RF Probe Test Board are implemented using a sequential lamination stackup. Rogers RO3003 core substrate is used between metal layers 1 and 2. The remaining, general purpose, layers 3 through 6 are etched on a Technolam NP-175F core and prepreg substrate. Stack up, GCPW, and BGA fan-out dimensions are all shown below.

Table 2 - RO3003/NP-175F Sequential Lamination Stackup for IWR1642 RF Probe Test Board

Number	Name	Material Type	Thickness (mils)	Dielectric Constant, ε_r	Loss Tangent, δ	Primary Uses	Notes
		Solder Mask	0.400				2
1	Top	Copper + Immersion Silver	1.600			RF routing, and top-layer impedance controlled lines	3, 4
		Rogers RO3003	5.000	3.66	0.0037		
2	L2	Copper	1.200			Solid ground reference layer	
		NP-175F Prepreg 1080	3.000	3.700	0.016		
		NP-175F Prepreg 1080	3.000	3.700	0.016		
3	L3	Copper	1.200			Digital signal routing and ground reference	
		NP-175F core	28.000	4.280	0.016		
4	L4	Copper	1.200			Power and signal routing	
		NP-175F Prepreg 1080	3.000	3.700	0.016		
		NP-175F Prepreg 1080	3.000	3.700	0.016		
5	L5	Copper	1.2			Solid ground reference layer	
		NP-175F core	5.000	3.790	0.016		
6	Bottom	Copper + Immersion Silver	1.600			Power and signal routing	
		Solder Mask	0.400				

Note 1: All thicknesses shown here are finished or processed thicknesses. Check with your PCB fabricator to determine final process thicknesses.
Note 2: Solder mask was not used on the RF portions of the design due to effect it would have on the GCPW and antenna RF performance. Only a small solder mask "damn" was included near the RF BGA.
Note 3: An immersion silver finish was applied to the RF areas of the PCB to prevent oxidation on the RF areas that would not receive solder mask. Total thickness of base ½ oz copper and immersion silver finish was intended to be 50 micron (1.968 mils). Check with your PCB fabricator to determine final process thicknesses.
Note 4: The 1/2 oz / in^2 (17 micron) electrodeposited copper foil option was selected for the RO3003 core. Additional thickness was added as part of the sequential lamination process. Check with your PCB fabricator to determine final process thicknesses.

4.2.2 PCB to BGA transition and GCPW Transmission Line

An optimized AWR1642 BGA package to GCPW transition has been created based on the RO3003 stackup shown above.

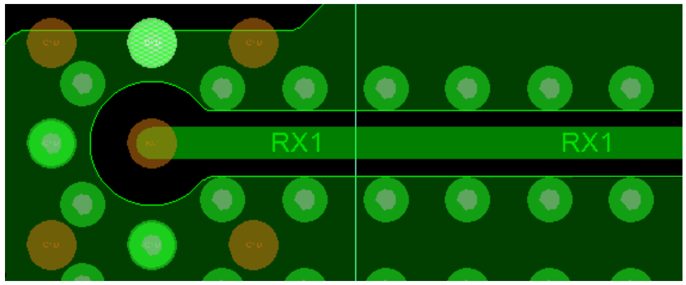
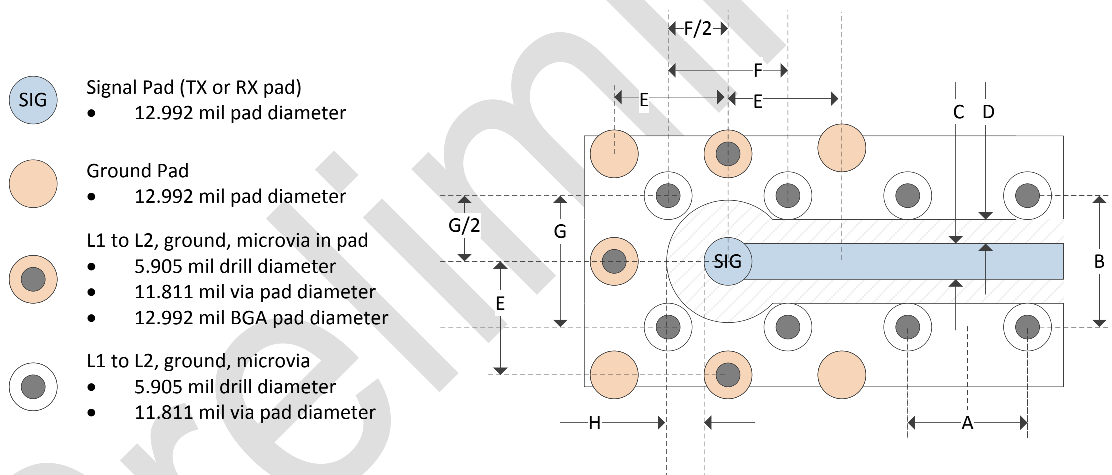


Figure 9 - AWR16142 RX1 BGA to PCB transition and GCPW fan out excerpt from RO3003 microprobe layout

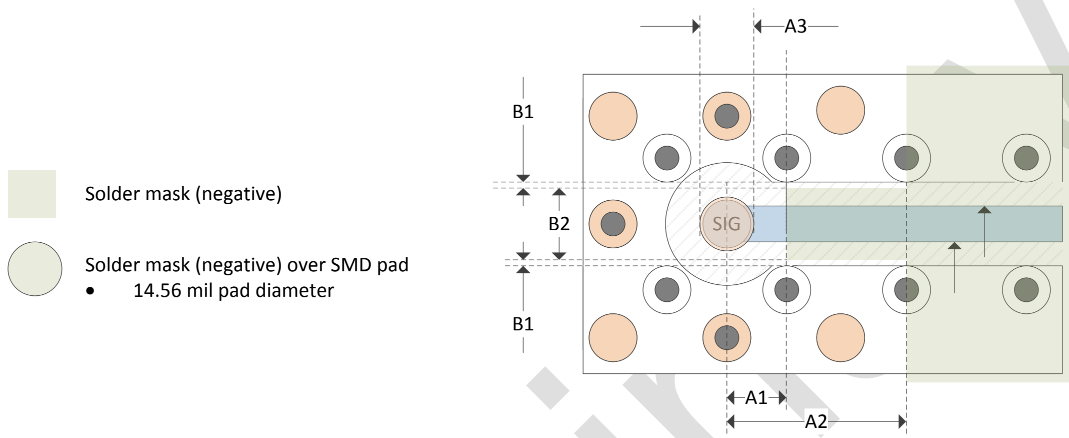
The TX/ RX signal pads and signal trace are surrounded by continuous reference ground plane on L1 and ground reference on L2. Three ground-stitching (L1-L2 ground) micro-vias are via-in-pad with the surrounding AWR1642 BGA ground pads. There are 4 additional ground-stitching micro-vias which are placed just outside ground plane keep out radius of the signal pad.
Together, with the anti-pad separation, these radial ground vias create a controlled impedance transition from the package BGA and solderballs to the GCPW PCB impedance in the AWR1642 RF operating bands.



Measurement	Value	Unit
A	20.669	mil
B	30.472	mil
C	8.385	mil
D	4.015	mil
E	25.590	mil
F	35.433	mil
G	30.708	mil
H	9.0551	mil

Figure 10 – AWR16142 RO3003 BGA to PCB Transition and GCPW Dimensions (Drawing Not to Scale)

Special care must be taken to ensure the accuracy and consistency of the solder mask near the RF BGA. Effects on the impedance of the RF lines due to solder mask variations near the RF BGA should be minimized by only including a minimal solder mask definition (solder-damn) near the BGA to prevent overflow during initial soldering. TI recommends utilization of laser direct imaging (LDI) solder mask methods over standard liquid photoimagible (LPI) solder mask due to the greater accuracy and consistency board to board that can be achieved with the LDI methods. TI does not recommend use of solder mask along the length of the GCPW transmission lines or etched antenna designs as this will significantly alter the performance of these structures.



Measurement	Value	Unit
A1	16.00	mil
A2	42.00	mil
A3	14.56	mil
B1	2.00	mil
B2	16.70	mil

Figure 11 – IWR1642 RO4835 LoPro BGA to PCB Transition Dimensions – Solder Mask Details (Drawing Not to Scale)

5 IWR/AWRx EVM Antenna

The IWR/AWRx BoosterPack EVM integrates an etched, series-fed, patch antenna design with the IWR/AWRx SoC transmitters and receivers. This antenna was designed to enable a generic range of FMCW radar applications. Please reference the BoosterPack EVM User Guide for simulations showing the expected radiation pattern graph and key metrics.

Many use-cases may be able to take advantage of this antenna design as-is. However, other designs will need a more application specific antenna integrated to achieve the necessary RF performance.

5.1.1 IWR1443 BoosterPack EVM Antenna Design

This section describes the current IWR1443 BoosterPack EVM antenna etch dimensions. This antenna geometry and relative placement is provided here in the form of measurements. These designs are also in the available in the GDSII CAD files which can be found along with this application note. GDSII files can be easily imported into many electrical and mechanical CAD layout tools.

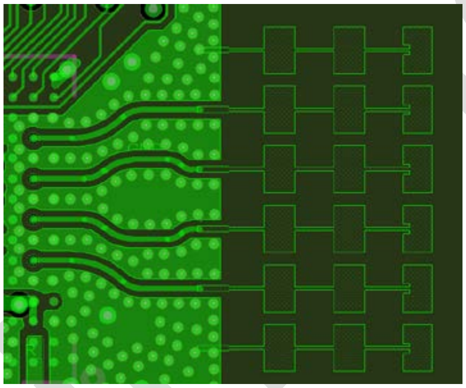


Figure 12 – IWR1443 Receiver Series-Fed Patch Antenna Array

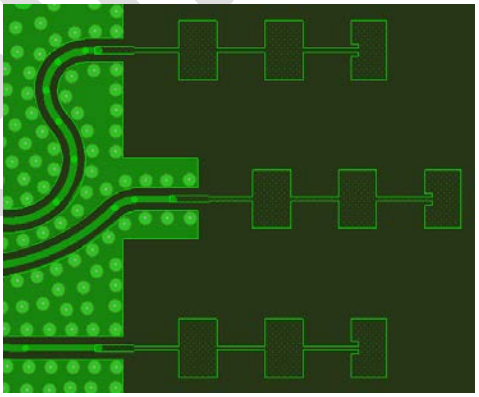


Figure 13 - IWR1443 Transmitter Series-Fed Patch Antenna Array

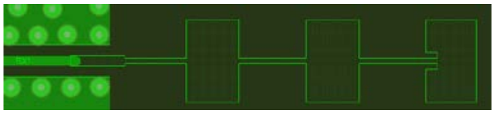
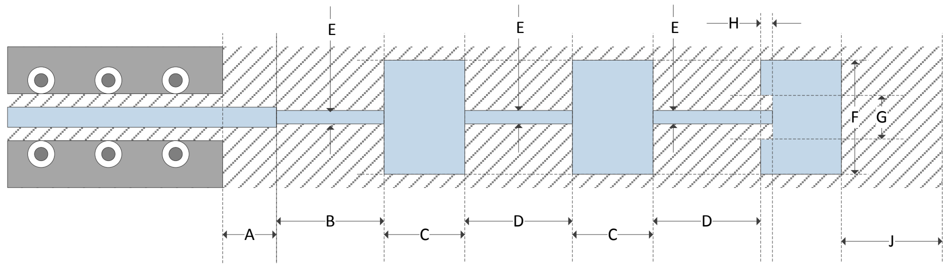


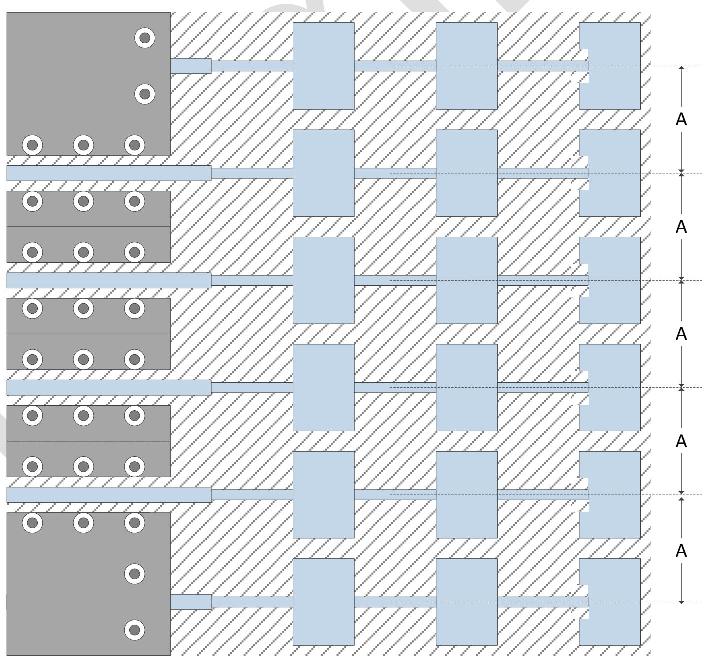
Figure 14 - IWR1443 Series-Fed Patch Antenna – Single Element



Measurement	Value	Unit	Note
A	11.210	mil	
B	44.490	mil	
C	38.190	mil	
D	48.420	mil	
E	3.9400	mil	
F	59.060	mil	
G	11.8200	mil	
H	7.9100	mil	
J	>2.000	mil	1

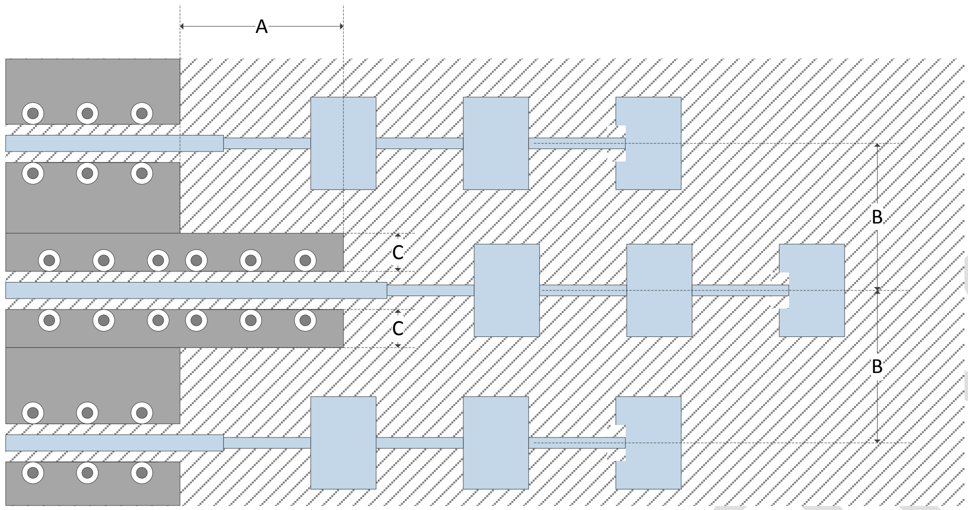
Figure 15 - IWR1443 Series-Fed Patch Antenna – Single Element Dimensions (drawing not to scale)

Note 1 - Dimension G denotes the minimum distance from the edge of the patch elements to the end of the continuous ground reference plane. This may be the edge of the PCB (as in the case of the BoosterPack EVM) or an internal edge.



Measurement	Value	Unit
A	74.800	mil

Figure 16 - IWR1443 Series-Fed Patch Antenna RX Array Element to Element Dimensions (drawing not to scale)



Measurement	Value	Unit
A	74.800	mil
B	149.6000	mil
C	29.6400	mil

Figure 17 - IWR1443 Series-Fed Patch Antenna TX Array Element to Element Dimensions (drawing not to scale)

5.1.2 IWR1642 EVM Antenna Design

- Description of the IWR1642 EVM antennas
  - Todo…

6 Extending the EVM BGA transition, GCPW and Antenna Designs

The RO3003 and RO4835 LoPro based layouts and layer stack ups presented are currently the only evaluated and supported RF BGA transition, transmission-line fan-out and antenna designs for the IWR/AWRx family of mmWave SoC. These structures were designed assuming they would be fabricated on the RF layer stack with the materials shown. TI has only evaluated functionality of these RF designs exactly as they are presented.

However, designers utilizing the IWR/AWRx family of devices will likely need to implement RF designs which are tailored for their end application. Currently the TI recommended flow is that RF designers start with the layer stackup, material options, BGA transition and GCPW transmission-line dimensions presented in this document and the EVM BoosterPack design layouts. This locks down the stack-up and fan-out transmission line dimensions, but it still allows for a wide variety of RF planar structures (other etched antenna, RF probe launches, and connector launches) to be fabricated on the L1 copper and referenced to the L2 ground layer.

With the stack-up and BGA transition and GCPW transmission-line chosen, the RF design task comes down to the following steps:

1. Designing the intended planar antenna,
2. Design any required impedance match transition to that planar antenna, and
3. Extending the length and curvature of the existing GCPW transmission-line for best phase match and location on the PCB to mate with the intended planar antenna

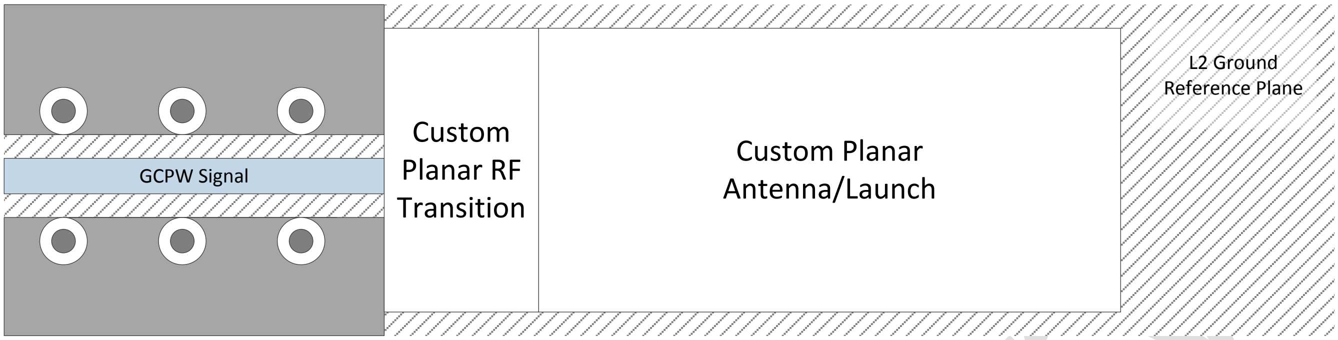


Figure 18 – Diagram of Custom RF Transition and Antenna Tasks

Many references exist to aid in the theory and design of these types of planar RF structures including *Antenna Theory: Analysis and Design* (Consantine A. Balanis, John Wiley & Sons, Feb 15, 2016).

7 References

1. Rogers Corporation RO4835 Laminates and Prepreg
   https://www.rogerscorp.com/documents/726/acs/RO4000-LaminatesData-sheet.pdf
2. Rogers Corporation RO3003 High Frequency Laminates
   https://www.rogerscorp.com/documents/722/acs/RO3000-Laminate-Data-Sheet-RO3003-RO3006-RO3010.pdf
3. Technolam NP-175F Core and Prepreg
   http://www.technolam.de/cms/pdf/upload/datenblaetter_en/Datasheets_NP-175F_E-9-2011.pdf
4. ITEQ IT 180A Low CTE / High Reliability Laminate & Prepreg: http://www.iteq.com.tw/wp-content/uploads/2017/03/IT-180A-Data-sheet.pdf
5. Flip Chip Ball Grid Array Package Reference Guide (SPRU811)
   http://www.ti.com/lit/ug/spru811a/spru811a.pdf
6. Antenna Theory: Analysis and Design (Consantine A. Balanis, John Wiley & Sons, Feb 15, 2016): http://www.wiley.com/WileyCDA/WileyTitle/productCd-1118642066.html

IMPORTANT NOTICE

Texas Instruments Incorporated and its subsidiaries (TI) reserve the right to make corrections, enhancements, improvements and other changes to its semiconductor products and services per JESD46, latest issue, and to discontinue any product or service per JESD48, latest issue. Buyers should obtain the latest relevant information before placing orders and should verify that such information is current and complete. All semiconductor products (also referred to herein as “components”) are sold subject to TI’s terms and conditions of sale supplied at the time of order acknowledgment.

TI warrants performance of its components to the specifications applicable at the time of sale, in accordance with the warranty in TI’s terms and conditions of sale of semiconductor products. Testing and other quality control techniques are used to the extent TI deems necessary to support this warranty. Except where mandated by applicable law, testing of all parameters of each component is not necessarily performed.

TI assumes no liability for applications assistance or the design of Buyers’ products. Buyers are responsible for their products and applications using TI components. To minimize the risks associated with Buyers’ products and applications, Buyers should provide adequate design and operating safeguards.

TI does not warrant or represent that any license, either express or implied, is granted under any patent right, copyright, mask work right, or other intellectual property right relating to any combination, machine, or process in which TI components or services are used. Information published by TI regarding third-party products or services does not constitute a license to use such products or services or a warranty or endorsement thereof. Use of such information 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.

Reproduction of significant portions of TI information in TI data books or data sheets is permissible only if reproduction is without alteration and is accompanied by all associated warranties, conditions, limitations, and notices. TI is not responsible or liable for such altered documentation. Information of third parties may be subject to additional restrictions.

Resale of TI components or services with statements different from or beyond the parameters stated by TI for that component or service voids all express and any implied warranties for the associated TI component or service and is an unfair and deceptive business practice. TI is not responsible or liable for any such statements.

Buyer acknowledges and agrees that it is solely responsible for compliance with all legal, regulatory and safety-related requirements concerning its products, and any use of TI components in its applications, notwithstanding any applications-related information or support that may be provided by TI. Buyer represents and agrees that it has all the necessary expertise to create and implement safeguards which anticipate dangerous consequences of failures, monitor failures and their consequences, lessen the likelihood of failures that might cause harm and take appropriate remedial actions. Buyer will fully indemnify TI and its representatives against any damages arising out of the use of any TI components in safety-critical applications.

In some cases, TI components may be promoted specifically to facilitate safety-related applications. With such components, TI’s goal is to help enable customers to design and create their own end-product solutions that meet applicable functional safety standards and requirements. Nonetheless, such components are subject to these terms.

No TI components are authorized for use in FDA Class III (or similar life-critical medical equipment) unless authorized officers of the parties have executed a special agreement specifically governing such use.

Only those TI components which TI has specifically designated as military grade or “enhanced plastic” are designed and intended for use in military/aerospace applications or environments. Buyer acknowledges and agrees that any military or aerospace use of TI components which have ***not*** been so designated is solely at the Buyer's risk, and that Buyer is solely responsible for compliance with all legal and regulatory requirements in connection with such use.

TI has specifically designated certain components as meeting ISO/TS16949 requirements, mainly for automotive use. In any case of use of non-designated products, TI will not be responsible for any failure to meet ISO/TS16949.

Products		**Applications**	
Audio	www.ti.com/audio	Automotive and Transportation	www.ti.com/automotive
Amplifiers	amplifier.ti.com	Communications and Telecom	www.ti.com/communications
Data Converters	dataconverter.ti.com	Computers and Peripherals	www.ti.com/computers
DLP® Products	www.dlp.com	Consumer Electronics	www.ti.com/consumer-apps
DSP	dsp.ti.com	Energy and Lighting	www.ti.com/energy
Clocks and Timers	www.ti.com/clocks	Industrial	www.ti.com/industrial
Interface	interface.ti.com	Medical	www.ti.com/medical
Logic	logic.ti.com	Security	www.ti.com/security
Power Mgmt	power.ti.com	Space, Avionics and Defense	www.ti.com/space-avionics-defense
Microcontrollers	microcontroller.ti.com	Video and Imaging	www.ti.com/video
RFID	www.ti-rfid.com		
OMAP Applications Processors	www.ti.com/omap	**TI E2E Community**	e2e.ti.com

Wireless Connectivity www.ti.com/wirelessconnectivity
Mailing Address: Texas Instruments, Post Office Box 655303, Dallas, Texas 75265