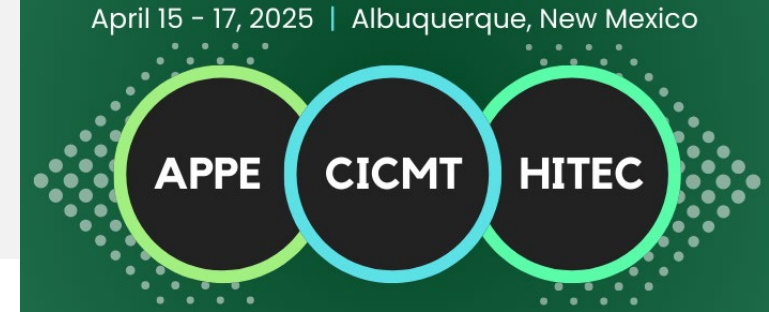




UNIVERSITY OF
ARKANSAS



Lessons Learned To Transfer a K-Band Radar System Designed on a Printed Circuit Board to a Low Temperature Co-Fired Ceramic Board

Alan Mantooth¹, Emmanuel Decrossas¹, Garrett Peterson² & Samir El-Ghazaly¹

¹ University of Arkansas

² Kansas State University

KANSAS STATE
UNIVERSITY





Overview

- Introduction
 - K-band agricultural radar board
 - Objective
- Lessons learned
 - Fabrication tolerances
 - PCB versus LTCC layers
- Fabrication cycle
- HiDEC fabrication LTCC lab
- Conclusion & Discussion



UNIVERSITY OF
ARKANSAS

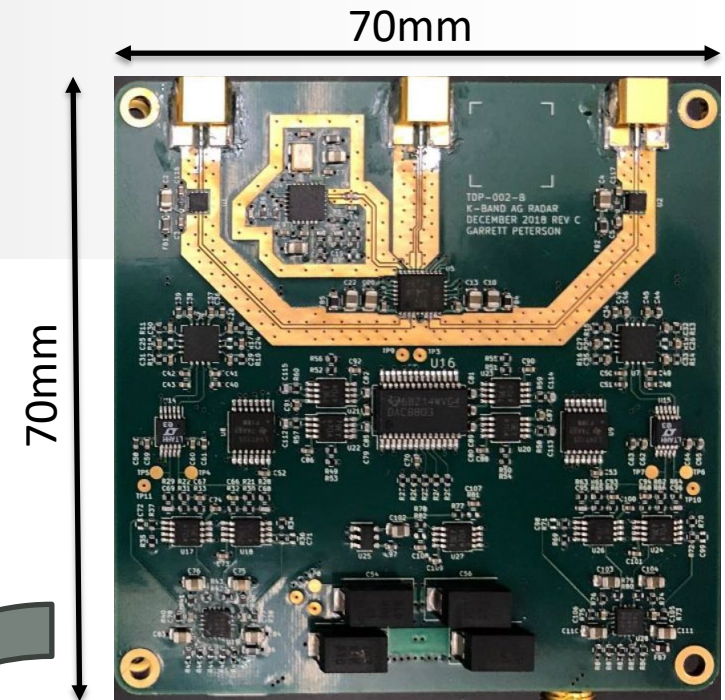
Introduction

- The K-band agricultural radar developed by KState is a stepped frequency continuous wave (SFCW) radar operating at 25 GHz
- 2.76" x 2.76" PCB footprint
- 6 (12.6 mils thick) layers FR4 board, 1 oz copper:
 - 3 power networks: +5V, -5V and 3.3V
 - 1 RF signal network
 - 1 ground network
 - Control signal lines

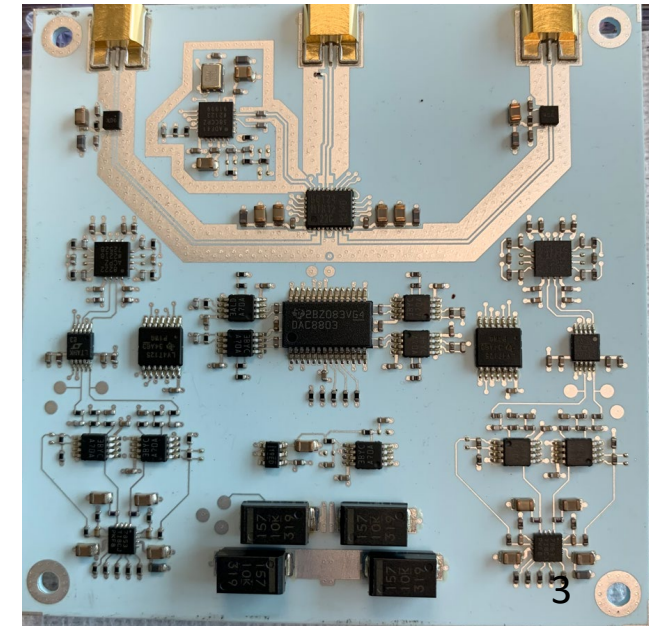
Objective

- Replace the FR4 PCB to Low temperature co-fired ceramics (LTCC)
- Keep the same footprint to match the existing electronic chassis
- Offer Better RF performance and thermal stability

000157



From PCB
to LTCC





Lessons learned

Fabrication tolerances design guidelines and consequences

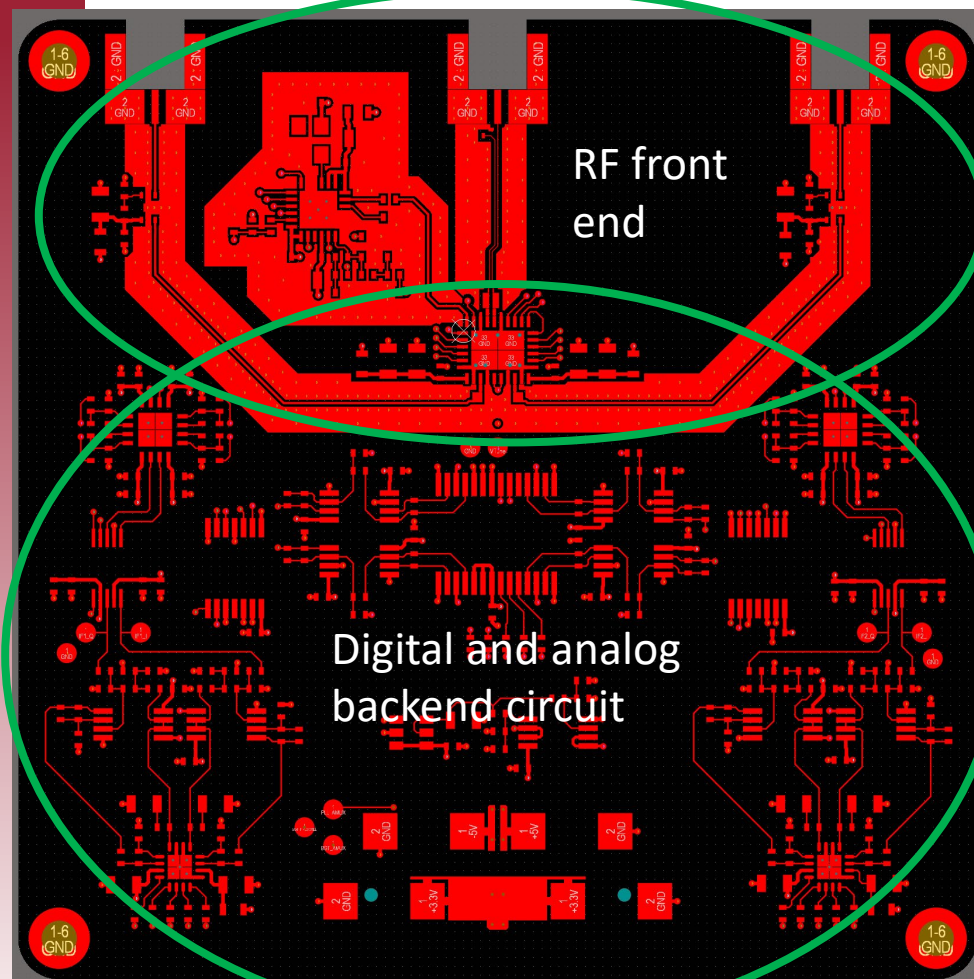
	FR4 PCB technology	LTCC technology	LTCC Design Consequences
Dielectric constant (ϵ_r) layer thickness (μm)	~ 4 320	~ 7.5 250	RF lines need to be adjusted and tuned to compensate for the change of materials and layer thickness
Minimum trace width (μm)	125-150	200-250	It is driven by the impedance of the RF transmission or the thickness of the tape used for LTCC for best RF performance
Minimum gap between traces	100- 150	200-250	Reroute the power and signal line and spread them within the board
Minimum Via diameter (μm)	200-300	100-254	Metal filled with ratio of 1:1 based on the thickness of the tape (in general 10 mils (254 μm))
Via pitch (μm)	2.5x via diameter	2x via diameter	Require pads between layers because of possible tape misalignment
Board warpage	Usually not an issue	Significant issue for large board	Develop vertical symmetry to control the shrinkage of the tape during lamination



UNIVERSITY OF
ARKANSAS

Top Layer: RF analog and digital signals

PCB

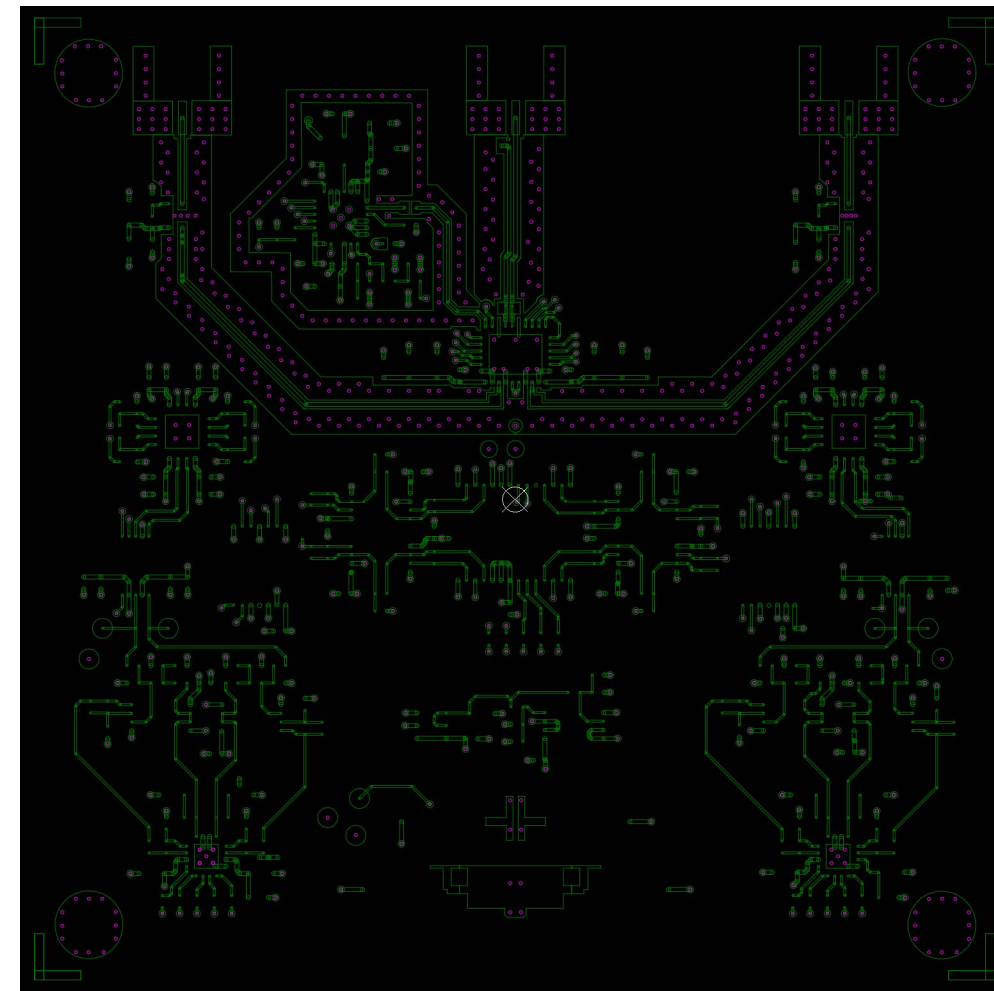


Redesign the coplanar waveguide and grounded vias to compensate for the change of material and layer thickness



Keep the digital and analog backend circuit unchanged; Keep the location of each component; Same board footprint to sit in same electronics box

LTCC

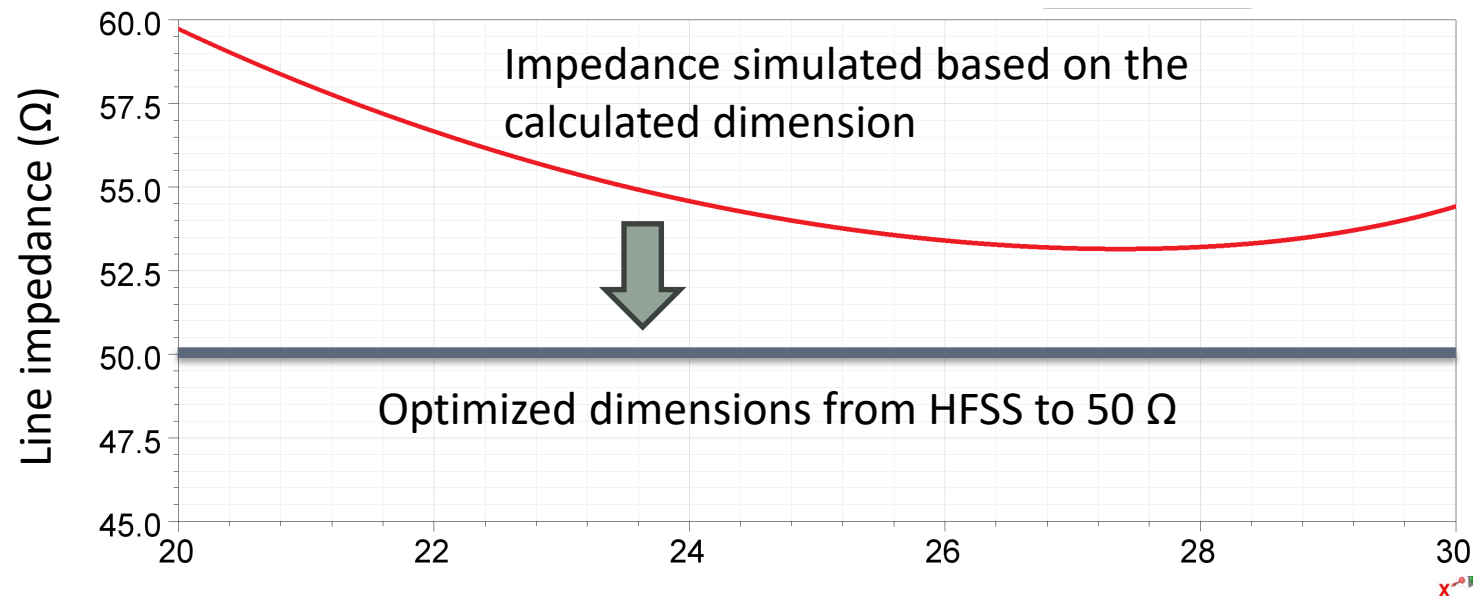




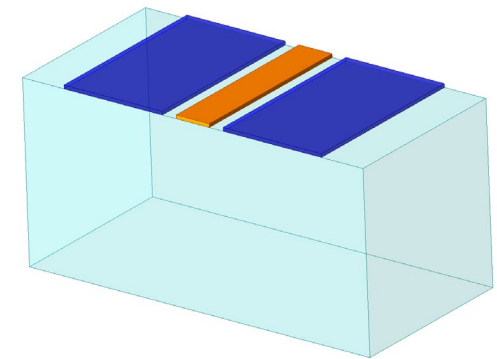
RF design

1. Simple impedance calculator are used to determine the first order width and gap of the coplanar waveguide
2. Full-wave modeling is then used to optimize the dimensions (using Ansys-HFSS, CST or FEKO)
3. After optimization, updated lines are implemented back in Altium designer

The same process is applied to all transmission lines such as microstrip, substrate integrated waveguide, etc...



Coplanar transmission line simulated in Ansys HFSS



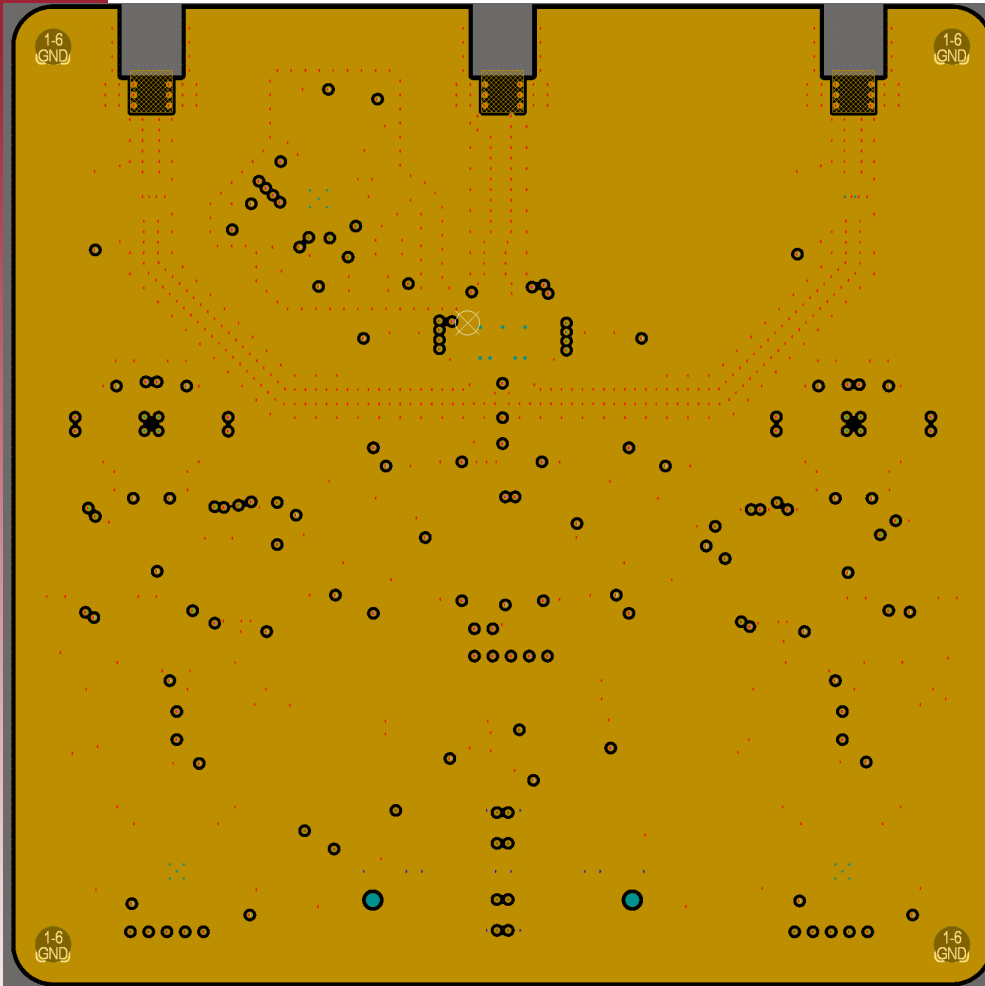
Ansys
2024 R1

0 0.035 0.07 (in)

000160

Layer 2: Ground layer

PCB



Replace plain metallization by grid pattern over large surface to avoid cracks during the firing process as the tape tends to shrink; Keep plain metallization under the coplanar waveguide for RF performance

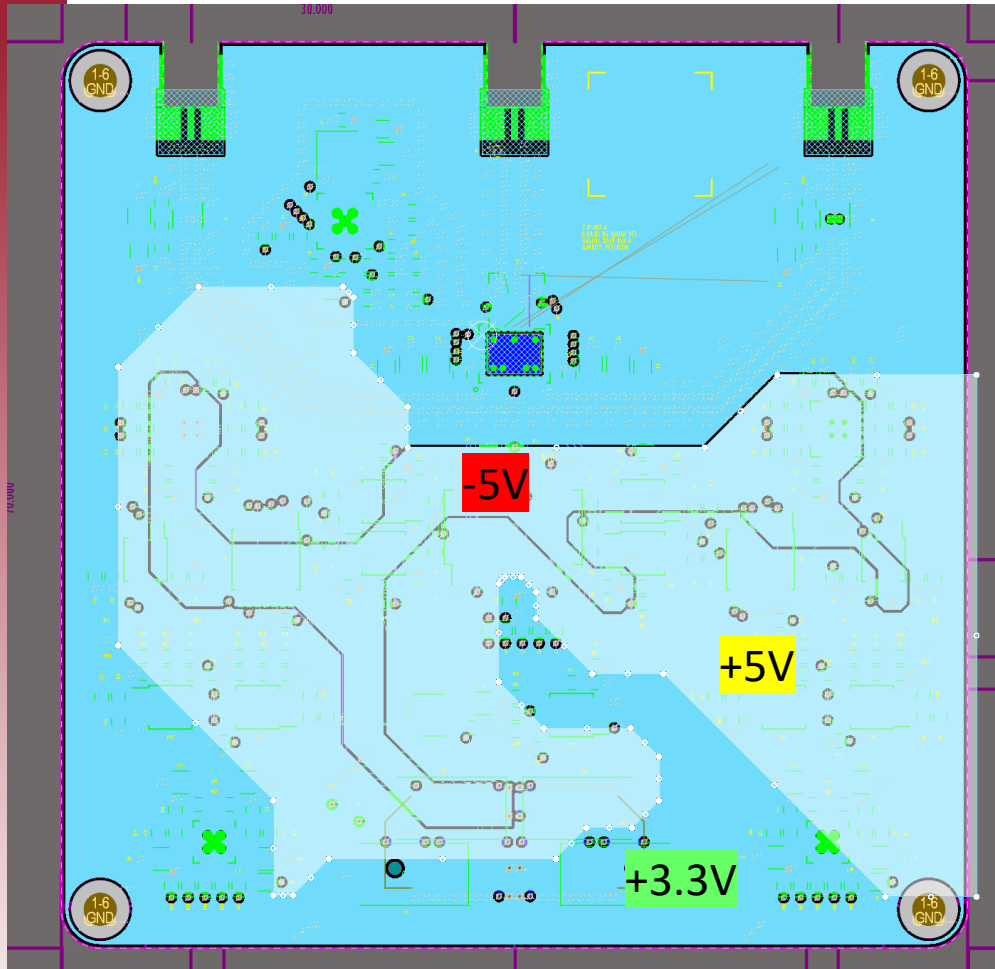


LTCC



Layer 3: Power network layer

PCB

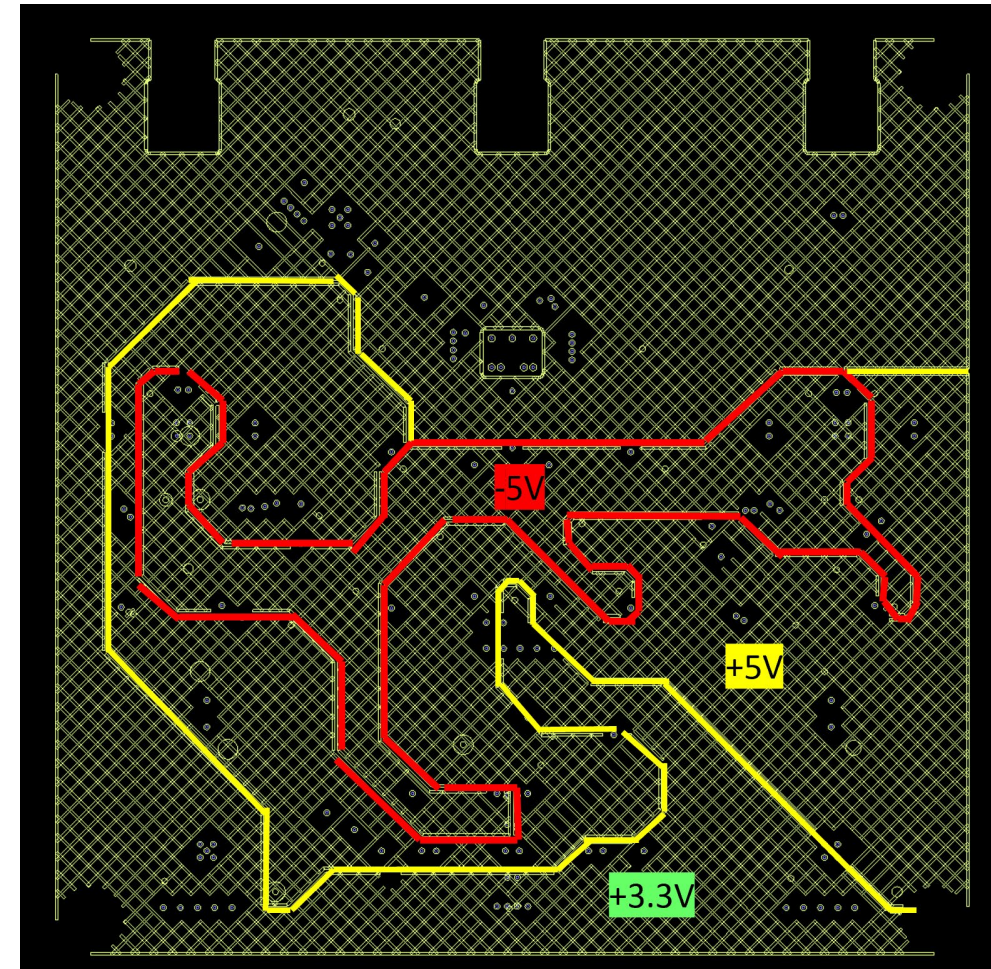


Replace plain
metallization
by grid pattern



Rotate the grid
pattern 45°
between vertical
layers to help
with board
warpage

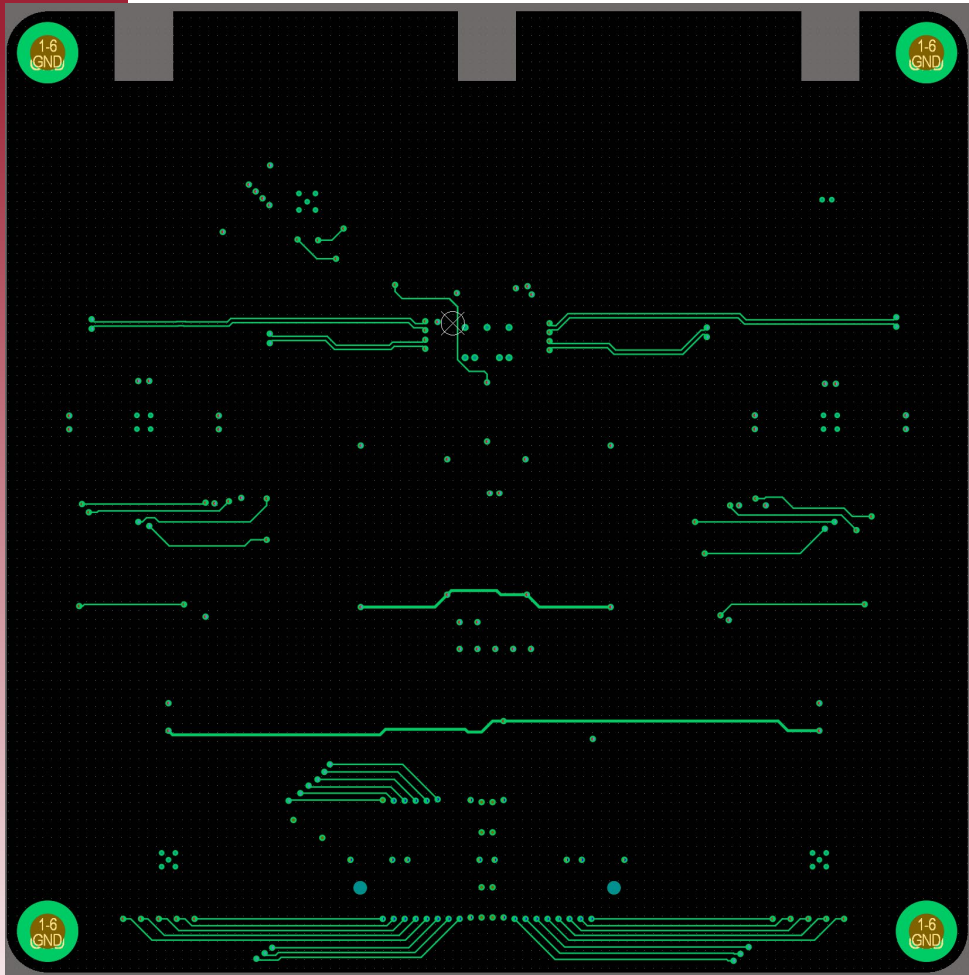
LTCC





Layer 4: Horizontal digital signal

PCB

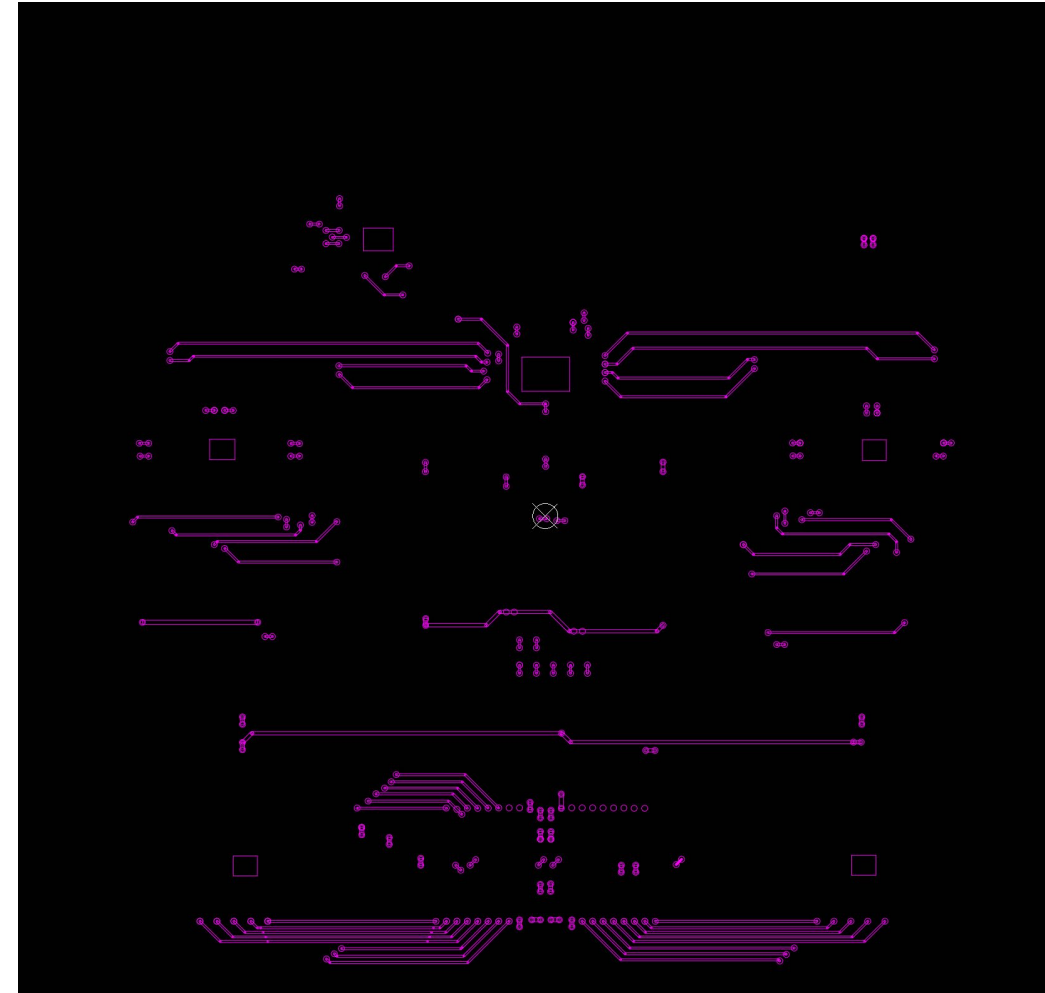


Reroute the signal lines to allow more spacing and increase the width of the signal lines; Simplify the fabrication process



No need to include grid metallization since this the center layer

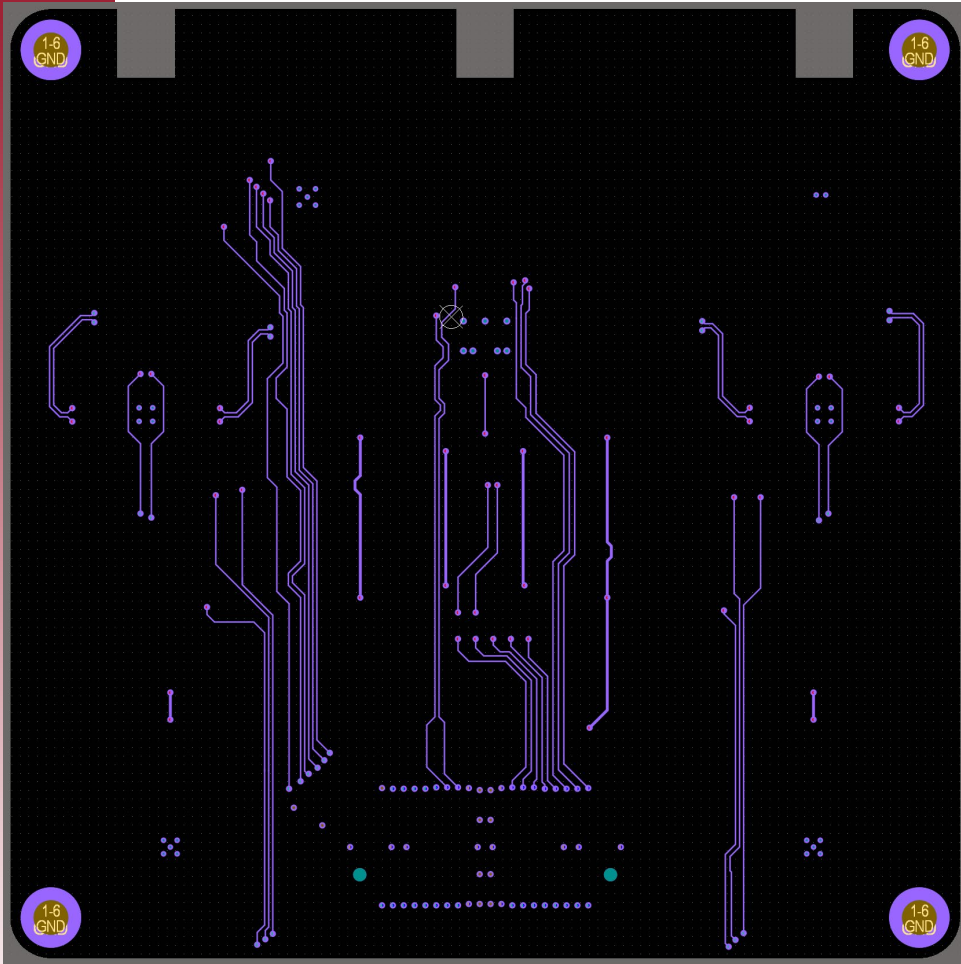
LTCC





Layer 5: Vertical digital signal

PCB

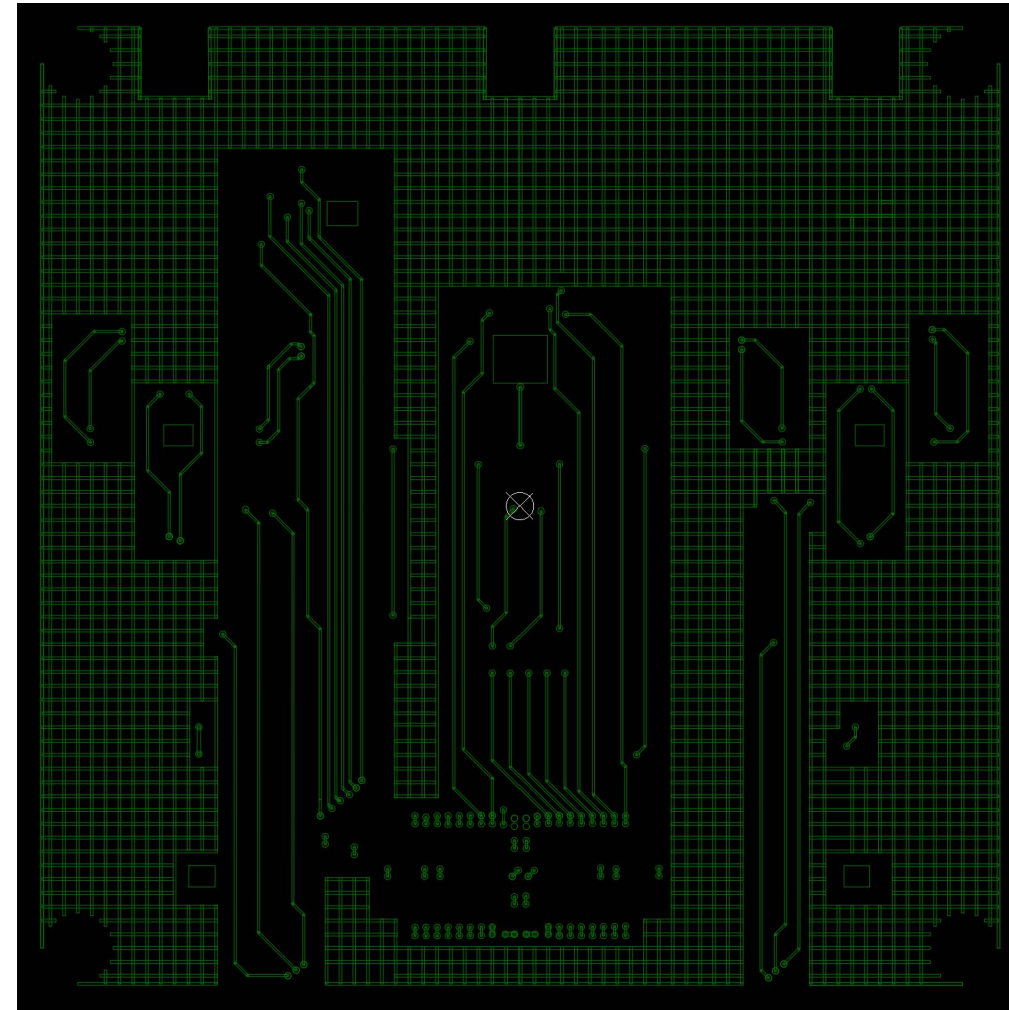


Reroute the signal lines to allow more spacing and increase the width of the signal lines; Simplify the fabrication process



Add the metallization grid to match layer 2 filling for vertical symmetry; This helps with the board warpage

LTCC

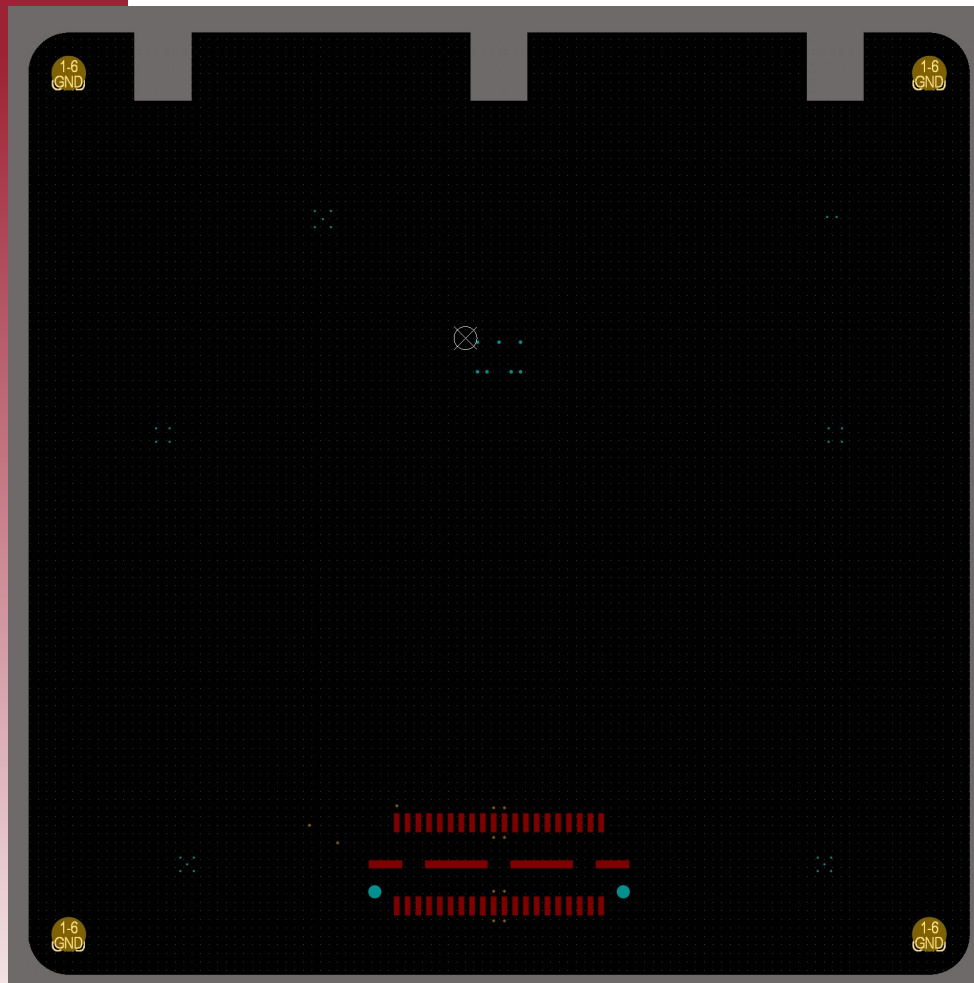




UNIVERSITY OF
ARKANSAS

Bottom layer: Soldering pad

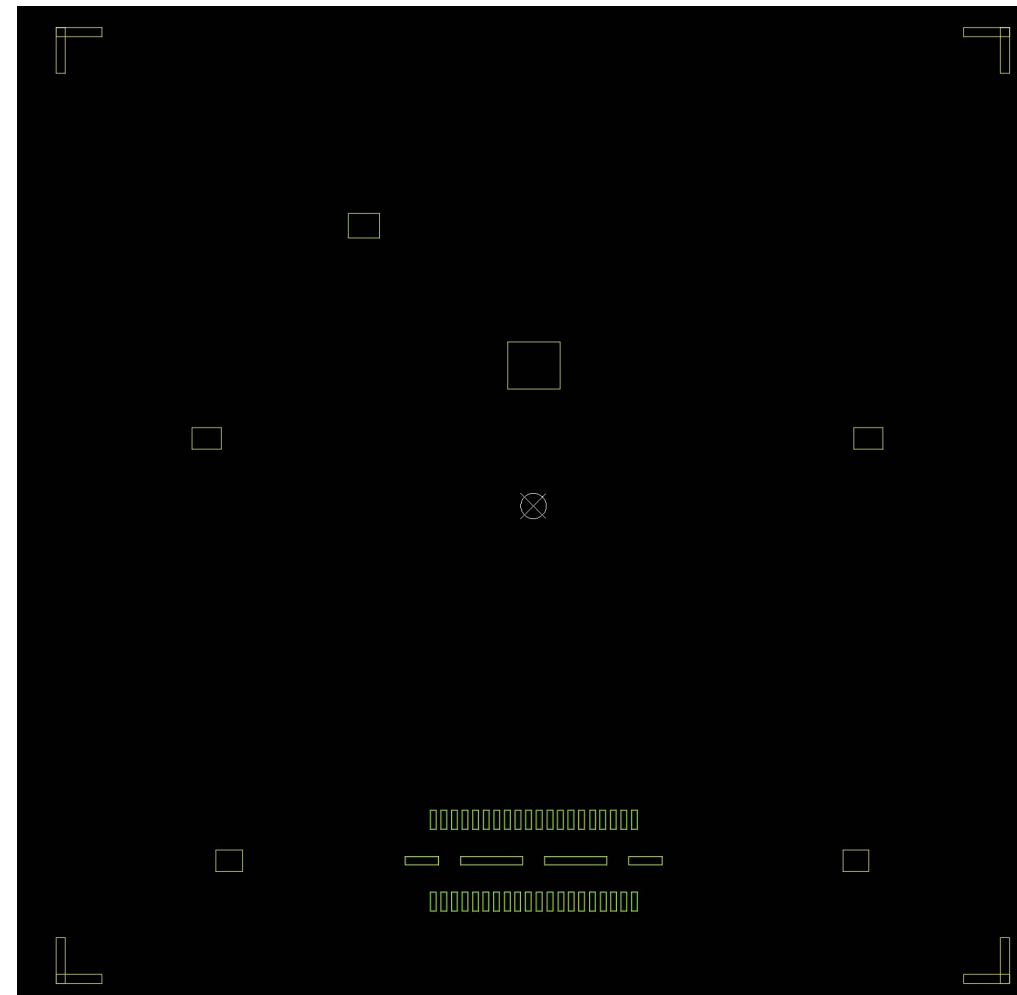
PCB



Location of the
connectors are
unchanged to
match the current
electronic chassis



LTCC





HiDEC Fabrication Cycle

- **Designer (responsible for the design): ~ 1-2 week**
 - **Software needed: Altium Designer, HFSS/CST/FEKO/ADS**
 - 1. Receive Altium or equivalent PCB design
 - 2. Understand the different network and layers (power, analog/digital signal, RF)
 - 3. Determine the RF transmission line dimensions
 - 4. Update the routing trace and via guidelines (gap, width, etc...) for LTCC
 - 5. Update plain metallized layer with grid
 - 6. Send the final layer design to manufacturing engineer. (gerber/drill files)
- **Manufacturer (responsible for the fabrication): ~3 weeks**
 - **Software needed: CAM350**
 - 1. Verify that the LTCC guidelines are respected through the gerber and drill files
 - 2. Scale CAD design to compensate for the shrinkage of the tape during fabrication
 - 3. Order the masks and fabricate the board
- **Tester (responsible for the test): ~ 1-2 days**
 - **Equipment needed: flying probe, network analyzer**
 - 1. Verify all networks before populating the board using flying probe
 - 2. Perform continuity check on the board if something is wrong, identify the problem and send back to designer for an iteration or populate the board

It usually requires one or two iterations to get the optimal performance on a board.

Depending on the complexity of the board, sometimes coupon tests are required to determine feasibility.





UNIVERSITY OF
ARKANSAS

High Density Electronics Center (HiDEC) - LTCC

HiDEC Ceramics Division

The ceramics integration laboratory houses punch, print, in-situ metrology, collate, laminate and firing tooling. The facility also holds wet etch, dry etch and dicing capabilities. This segment of the core facility aims to provide services of thick film circuitry for high frequency and extreme environment applications at both an R&D as well as pilot production volume.



Microtec 550TVC

Thick film screen printer for 200 mm square GreenTape. System utilizes a desktop PC with image capture software to help with screen to tape alignment.



Avure Technologies IL142005_SS

A batch-mode, semi-automatic isostatic lamination tool is currently configured to handle a batch of GreenTape assemblies hand packaged in water-tight and vacuum-sealed bags.

Thick Film Processing Tools:

UHT 8200Z SL punch

Baccini punch

Microtec 550TVC screenprinter

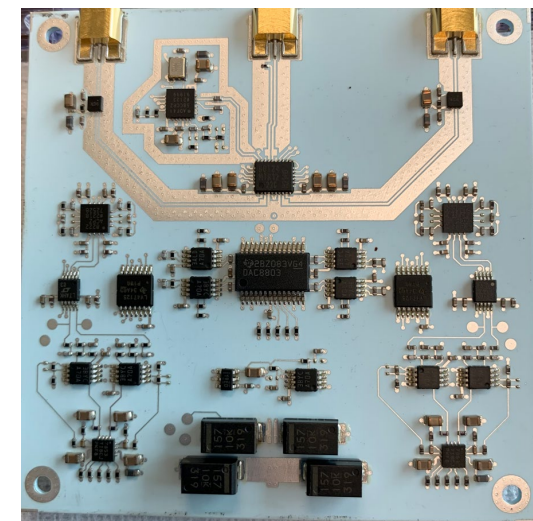
Stratus III Automated optical inspection tool

Baccini collator

Avure Technologies IL142005_SS laminator

Thermo Scientific 55.3L furnace

SierraTherm LTCC furnace



Fabricated LTCC board
fully populated



Conclusion & Discussion

- Transferring system from PCB to LTCC requires a certain amount of NRE and R&D work depending on the complexity of the board
 - List of software: Altium (PCB design) , HFSS/CST/FEKO (full-wave simulator), CAM350
- RF traces need to be redesigned to tune each impedance line in respect to the thickness of the tape and material used
- Embedded RF transmission lines improve signal integrity of the system
 - **Talk on Thursday 10.45am. : *Micro-Faraday Cage Within Low Temperature Co-Fired Ceramic Board Fabrication and Measurement***, Roberto Quezada, Emmanuel Decrossas, Alan Mantooth, Samir El-Ghazaly, University of Arkansas
- LTCC technology can improve RF high frequency performance quite significantly and provide more thermal stability than PCB technology
- Vertical integration of LTCC allows more integration flexibility and reduced footprint of the board



UNIVERSITY OF
ARKANSAS

Thank you

Questions?