LPKF Laser & Electronics

AWR A National Instruments Company™

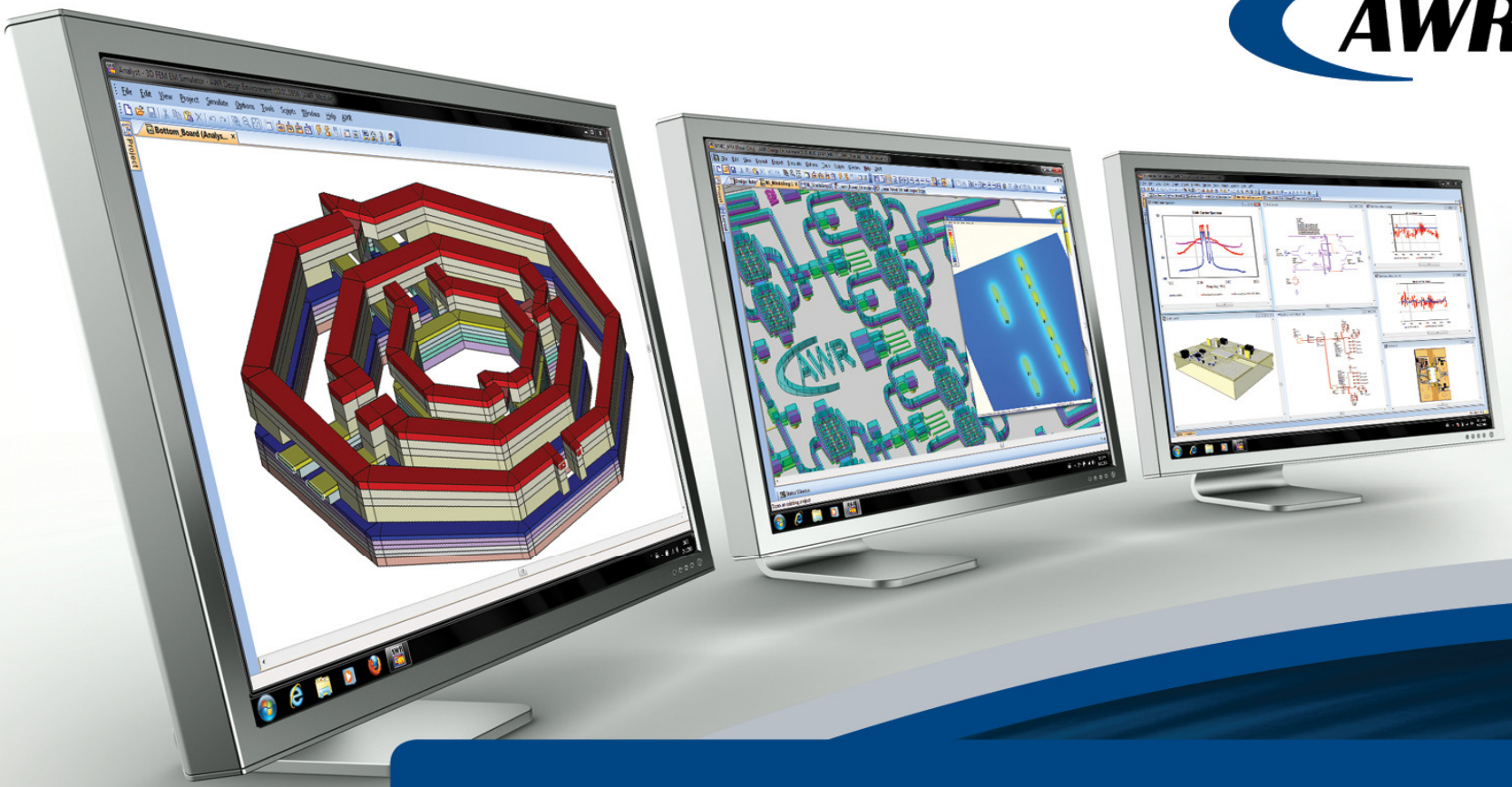
RF PCB Design

Henry Lau

Lexiwave Technology, Inc

December 4, 2012

AWR | A National Instruments Company™



AWR Corporation – Overview & Introduction

www.awrcorp.com

AWR - At a Glance

Advancing the wireless revolution®

www.awrcorp.com

The Innovation Leader in High-Frequency EDA

Product Portfolio:

- Microwave Office™ - MMIC, RF PCB and module
- Visual System Simulator™ - Wireless comms/radar
- AXIEM® - 3D planar EM
- Analog Office® - RFIC

Global Presence (direct offices)

- Los Angeles, California (headquarters)
- California, Wisconsin, Colorado
- United Kingdom and Finland
- Japan, Korea and China

AWR | A National Instruments Company™

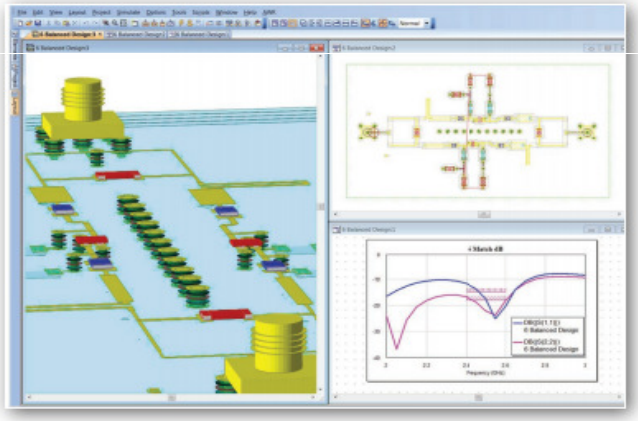
Microwave Office

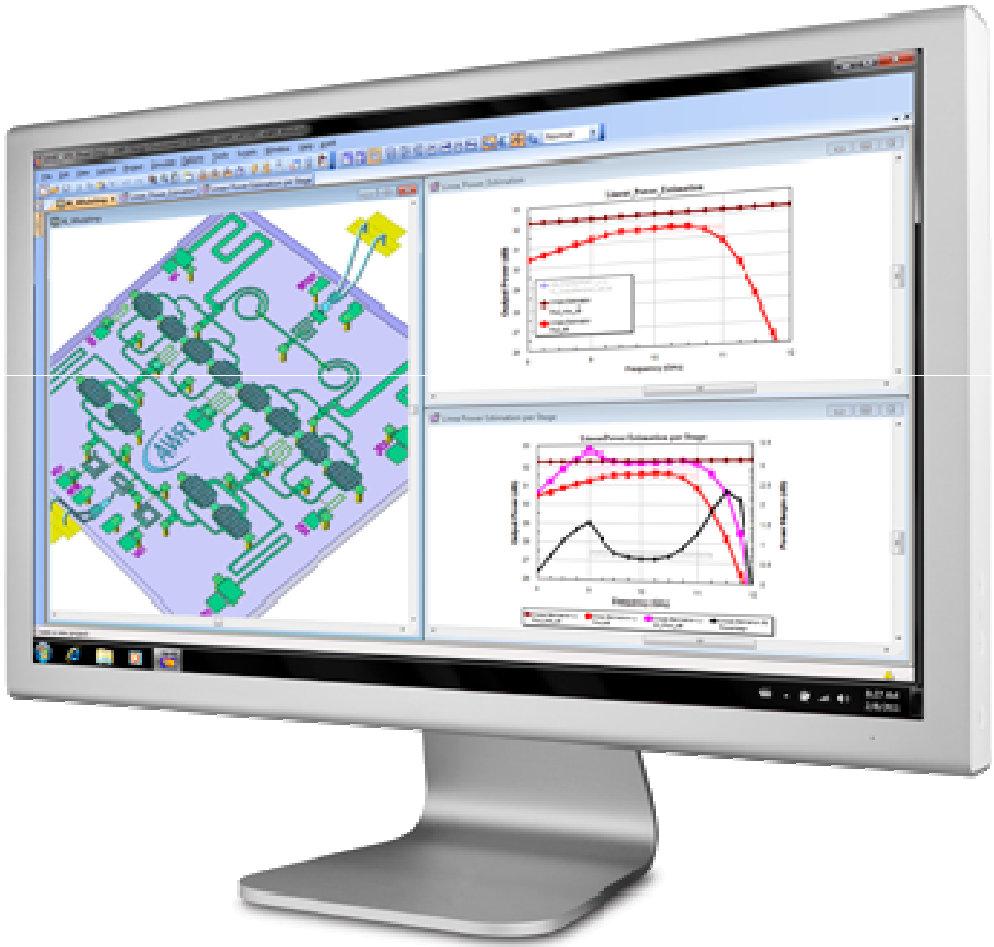
Advancing the wireless revolution®

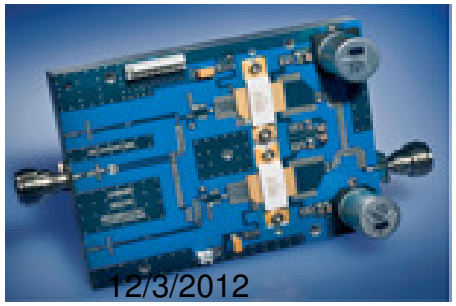
www.awrcorp.com

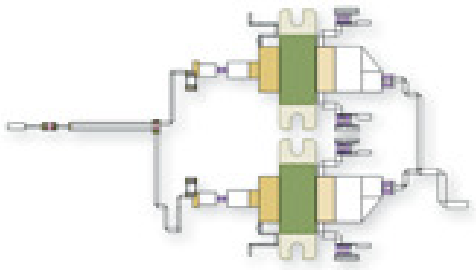
RF and Microwave Design Software

- MMIC
- RF PCB
- Modules









AWR | A National Instruments Company™

Microwave Office for PCB Design

Advancing the wireless revolution®

www.awrcorp.com

- Unified schematic & layout for easy design entry, simulation, optimization, tuning and export to manufacturing
- Versatile simulation technology:
  - APLAC® - Harmonic Balance
  - ACE™ - Automated Circuit Extraction
  - AXIEM® – 3D Planar Electromagnetics
  - Analyst™ – 3D FEM Electromagnetics
- PCB links to third-party tools for post-layout verification
  - Cadence Allegro,
  - Mentor Expedition,
  - Zuken CR-5000/8000
  - Altium, Intercept, and more…

Alcatel-Lucent Reduces Design Time by Eliminating File translation Issues Between PCB and High-Frequency Design

”The integrated flow between the mentor and AWR tools has enabled us to significantly cut our design times. By concurrently designing the RF circuits in the context of the rest of the PCB, we can also reduce our design and manufacturing respins, which helps us meet aggressive time-to-market goals..“

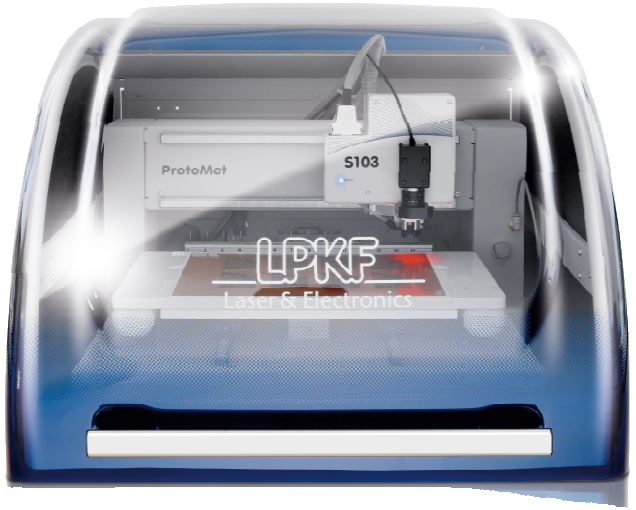
Xavier Leblanc,
Harware Tools Manager
Alcatel-Lucent

AWR | A National Instruments Company™

12/3/2012

PCB prototyping with LPKF

- *Benchtop milling machines*
- *Lab-ready laser systems*
- *SMT assembly equipment*



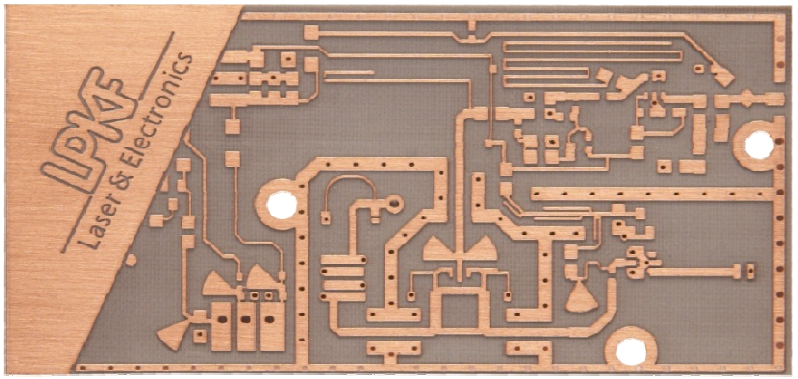
LPKF
Laser & Electronics

PCBs on demand

- *On the fly revisions*
- *Multiple iterations in a single day*
- *Beat your deadlines*

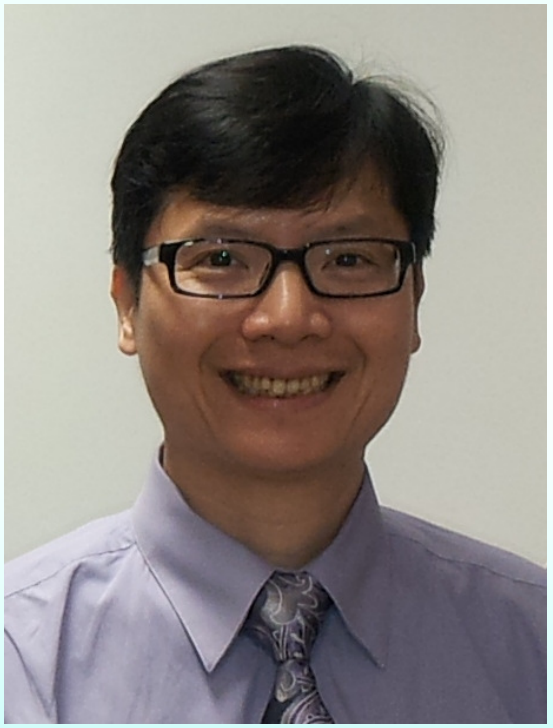
Next level prototyping

- *Visit www.LPKFUSA.com to learn more*



RF PCB Design

Henry Lau
Lexiwave Technology, Inc



LPKF Laser & Electronics

AWR A National Instruments Company™

Aims

- To acquire technical insights and design techniques on RF printed circuit board design for Wireless Networks, Products and Telecommunication
  * PCB of RF circuits
  * PCB of digital, analog and audio circuits
  * Design issues for EMI/EMC
  * Design for mass production

LPKF Laser & Electronics

AWR A National Instruments Company™

Contents

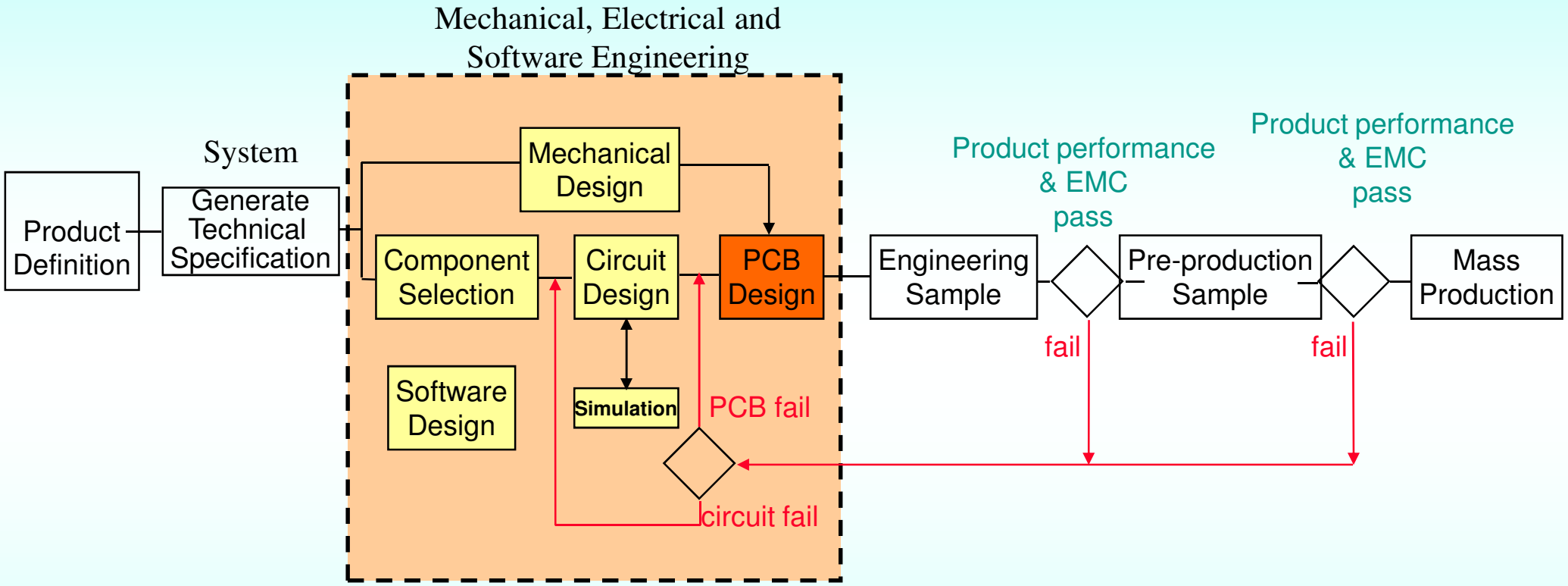
Printed Circuit Board design of RF circuits

- From product idea to mass production
- Design flow
- Layer stack assignment
- Board size and area
- Component placement
- Grounding Method
- Power routing
- Decoupling
- Trace routing
- Via holes : location, size and quantity
- Shielding

LPKF Laser & Electronics

AWR A National Instruments Company™

From Product Idea to Mass Production



Long cycle time process

LPKF Laser & Electronics

AWR A National Instruments Company™

Cooperation Between Mechanical & Electronic Design

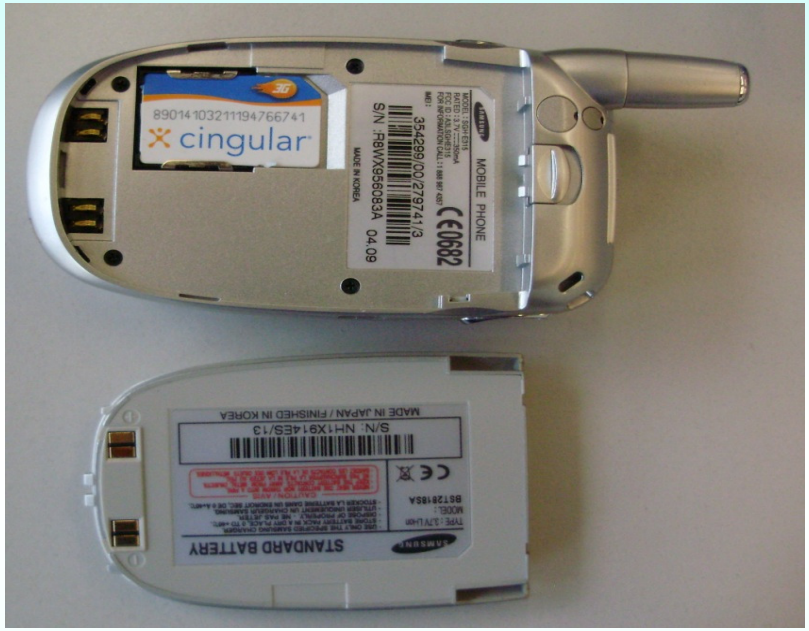
Case Study : Samsung Cellphone

- Marketing concerns
  - Outlook, features
  - ***Cost***
- Electrical performance concerns
  - Reception reliability
  - Sensitivity
  - Talk time
  - Stand-by time
- EMC concerns
  - Transmit powers and duration
  - ESD
  - Immunity tests

LPKF Laser & Electronics
AWR A National Instruments Company™

Cooperation Between Mechanical & Electronic Design

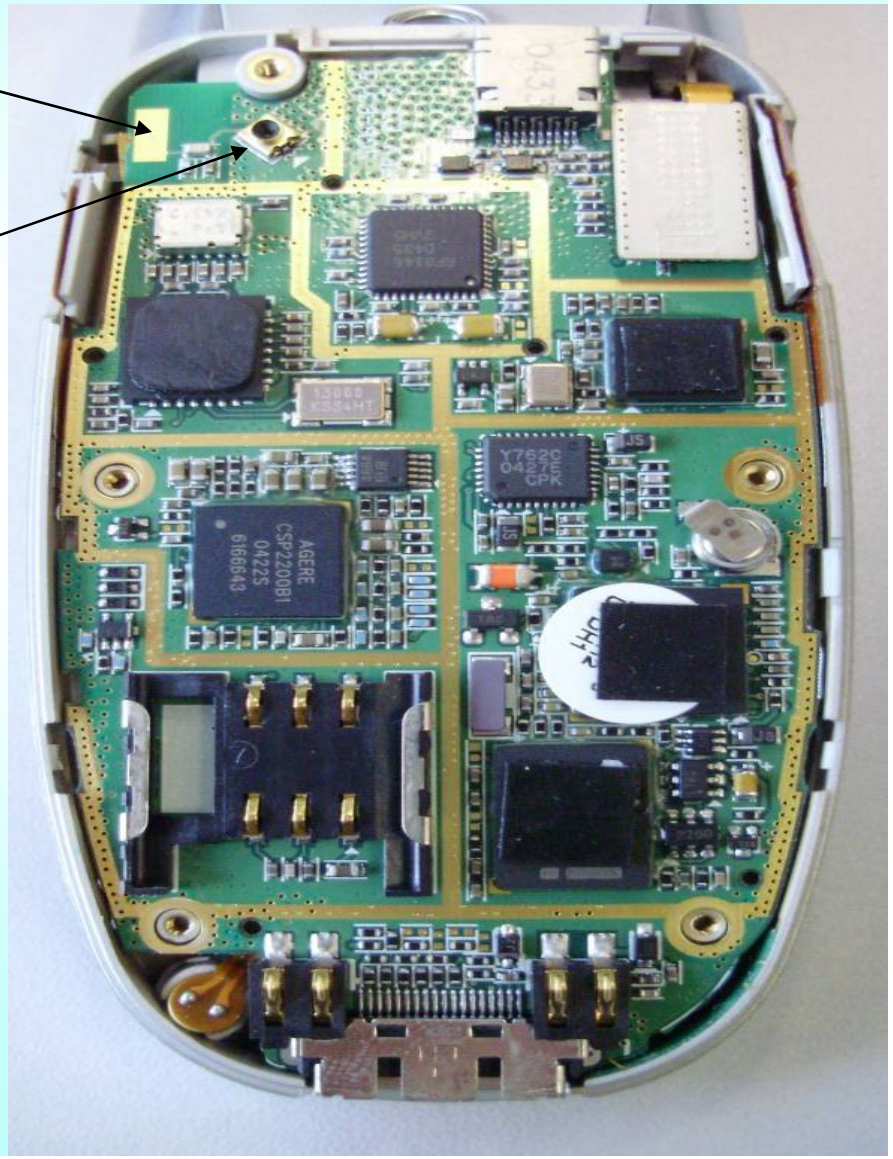
- Type and location of loudspeaker, microphone, display, keypad, switch
- Type of battery
- Location of I/O
  - antenna, power, analog, audio, digital . . . .
- Mounting method
  - screw and mounting holes, support poles
  - mechanical reliability and drop test



LPKF Laser & Electronics
AWR A National Instruments Company™

Cooperation Between Mechanical & Electronic Design

- Maximum thickness
- Maximum board size and optimal shape
  - maximum space utilization
- Power supply and large current connections
- Mass production concerns
  - easy assembly, alignment and repair

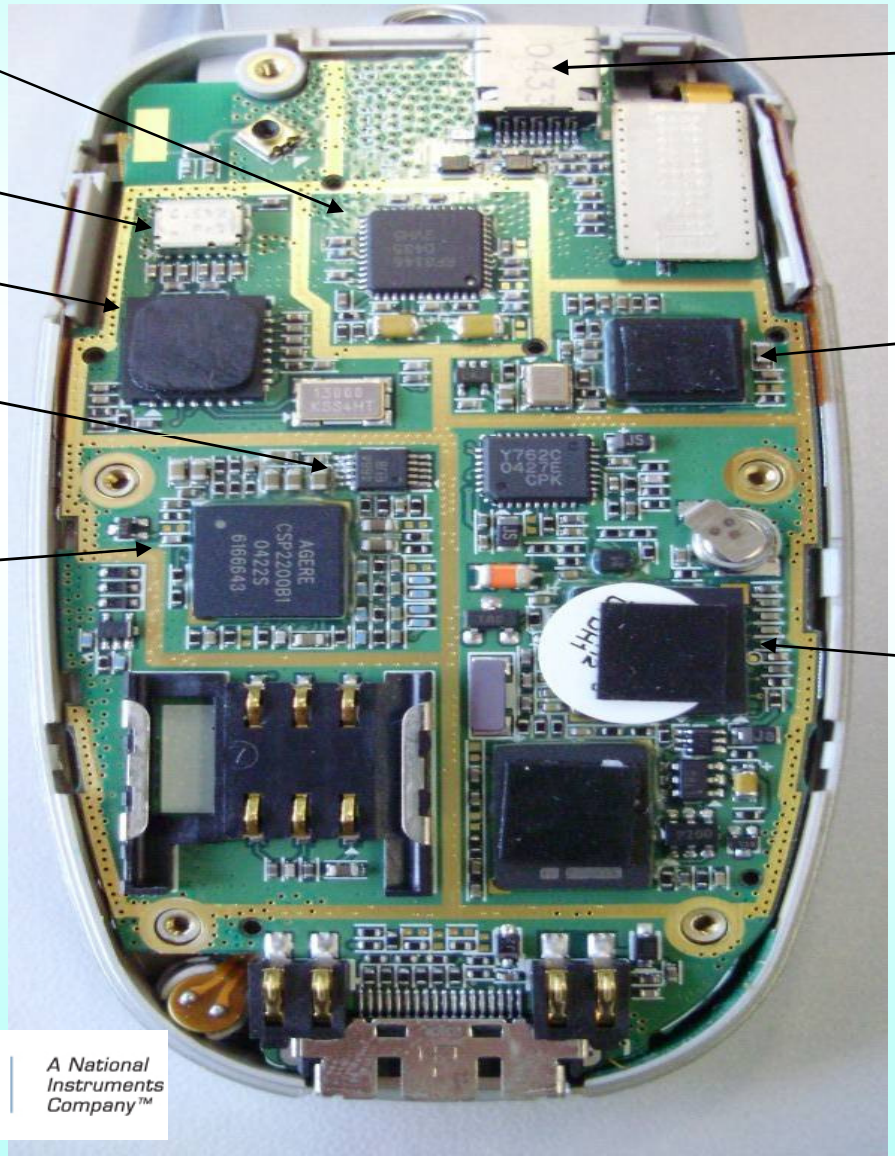


LPKF Laser & Electronics

AWR A National Instruments Company™

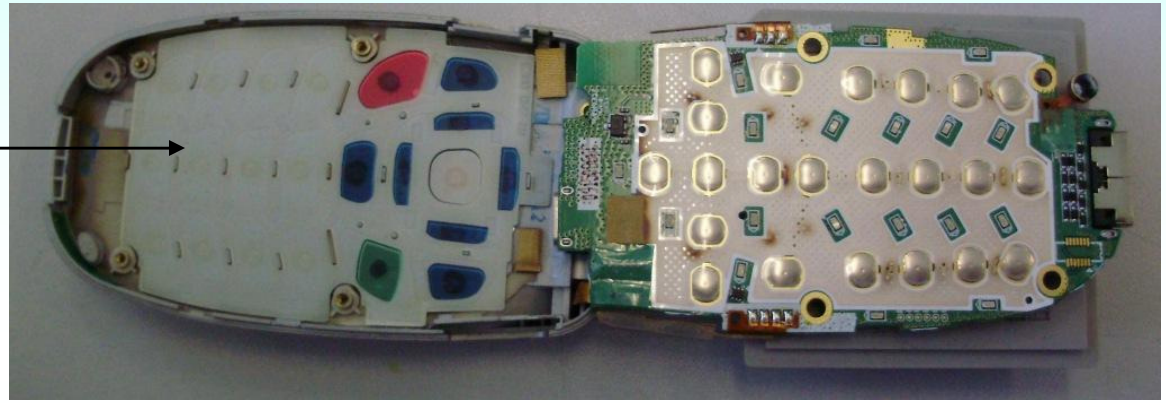
Cooperation Between Mechanical & Electronic Design

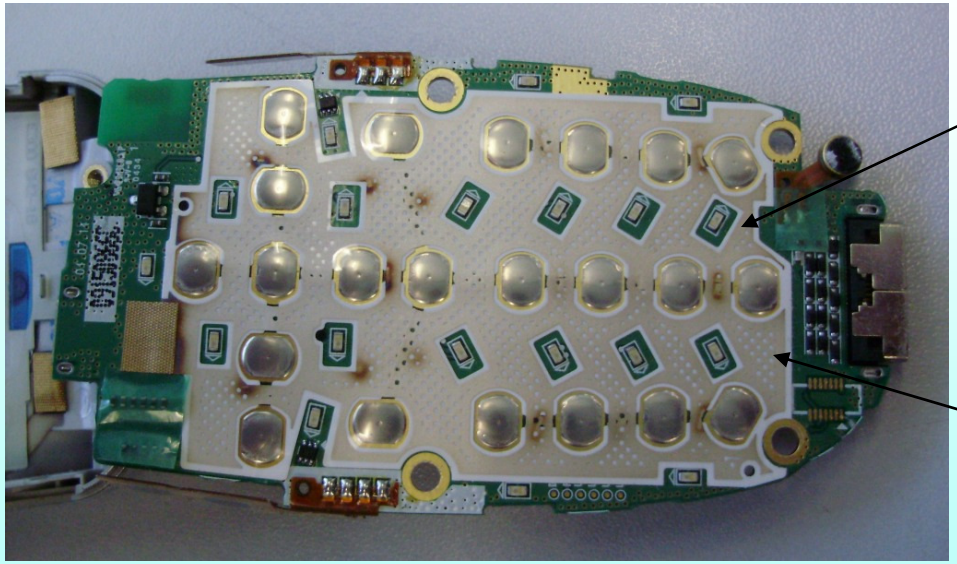
- Circuit grouping and partitioning
- Audio, video, digital, RF, analog
- Board mounting and assembly



Cooperation Between Mechanical & Electronic Design

Key Pad

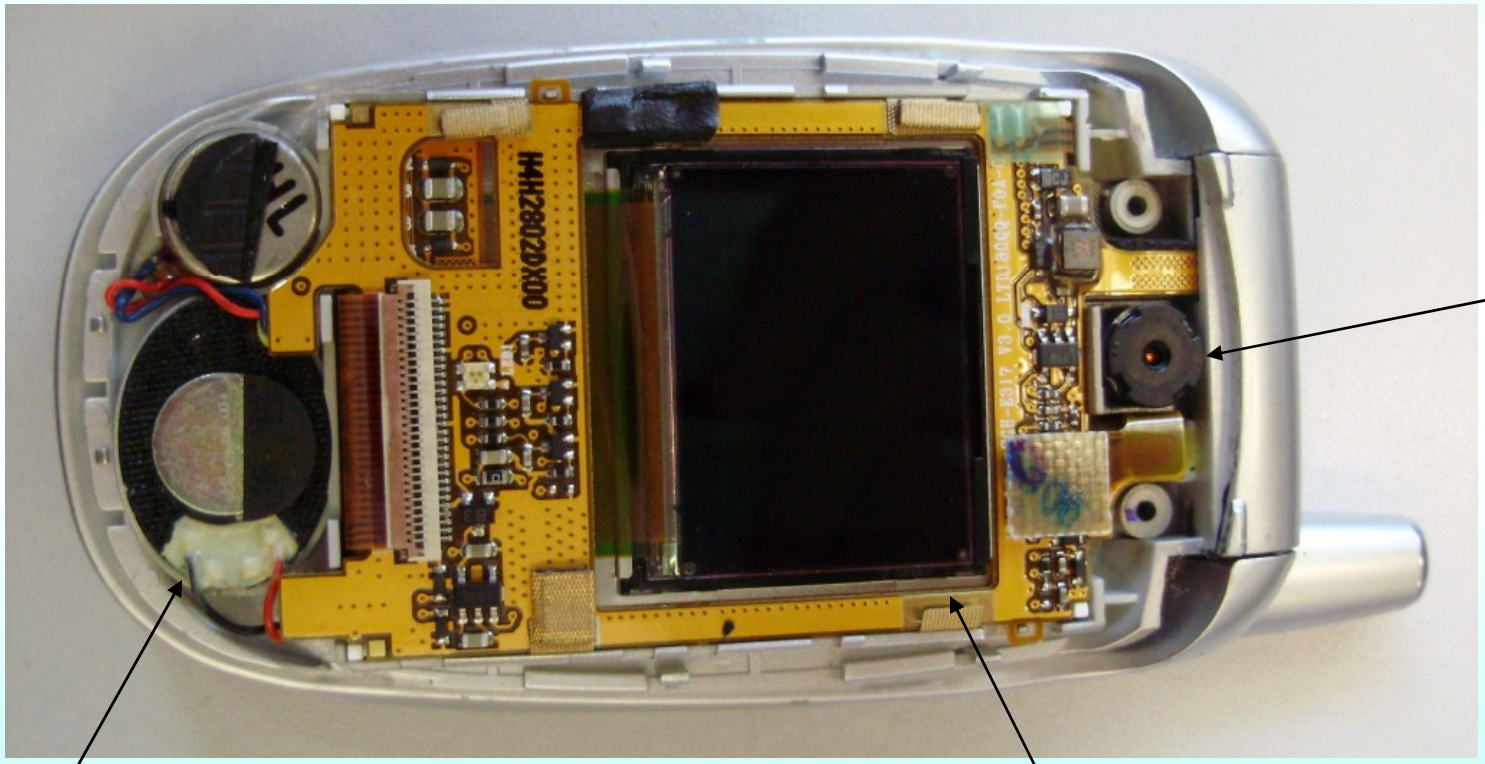




Membrane

Very few components on bottom layer

Cooperation Between Mechanical & Electronic Design



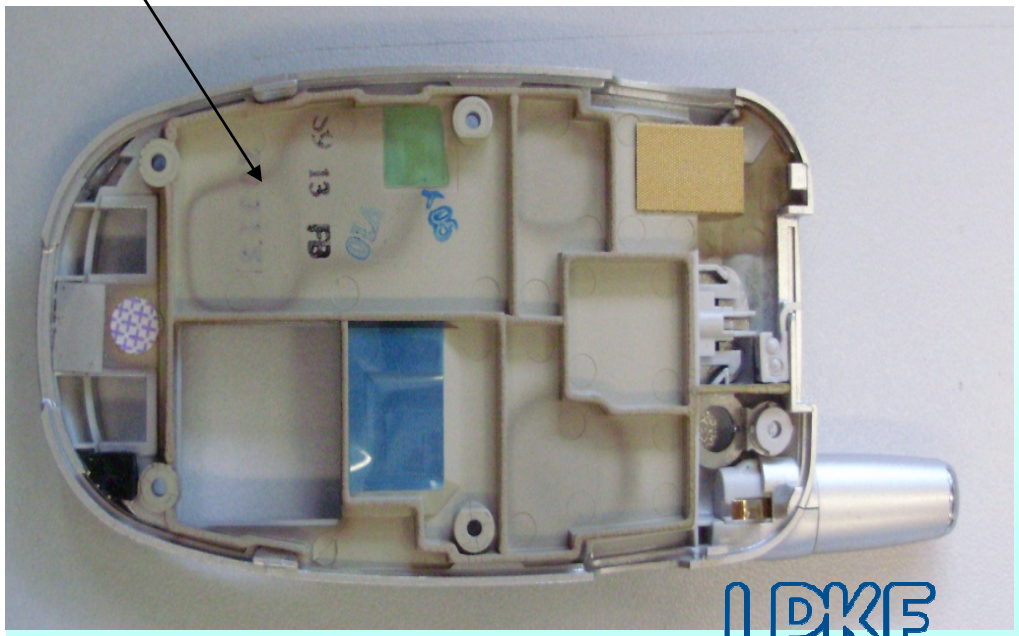
LPKF Laser & Electronics

AWR A National Instruments Company™

Cooperation Between Mechanical & Electronic Design

- Shielding and isolation
  - Method, material
- EMI/EMC/ESD issues

Metallization on plastic

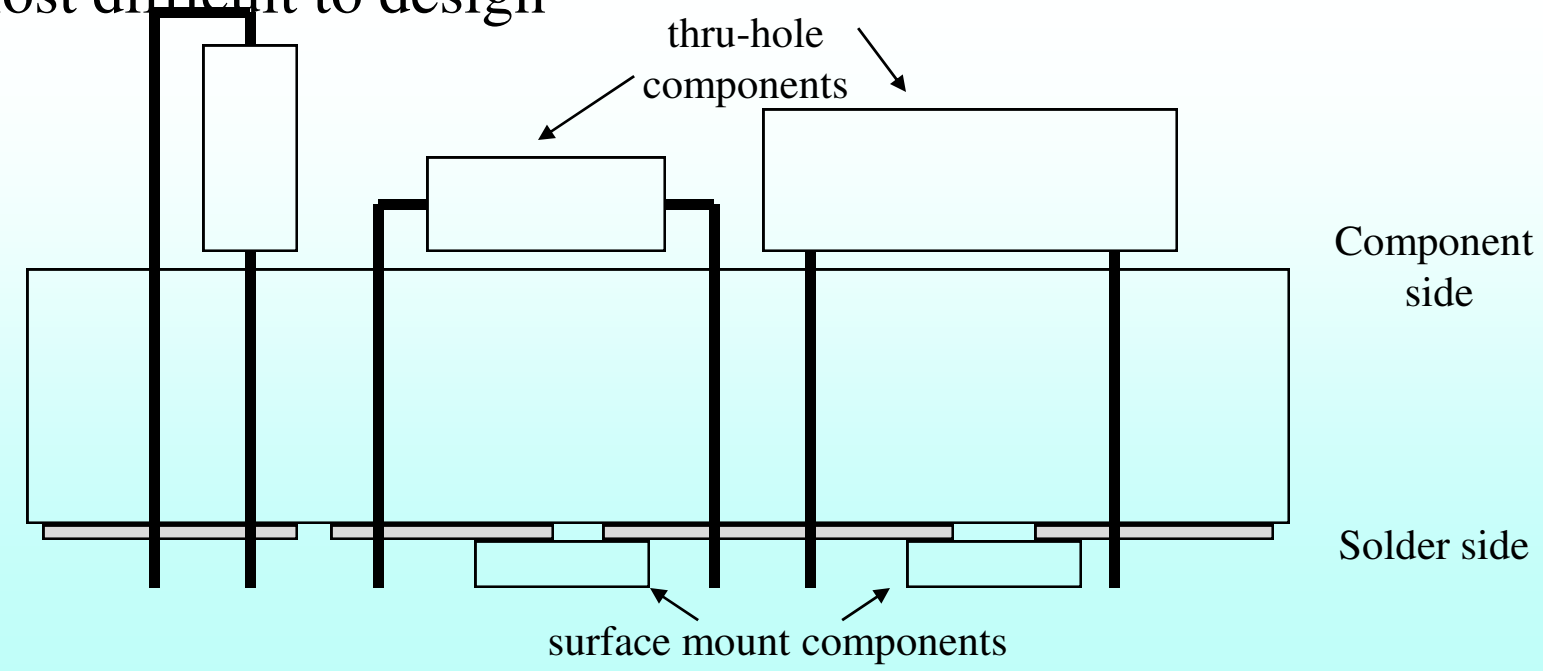


LPKF Laser & Electronics

AWR A National Instruments Company™

Layer Stackup Assignment

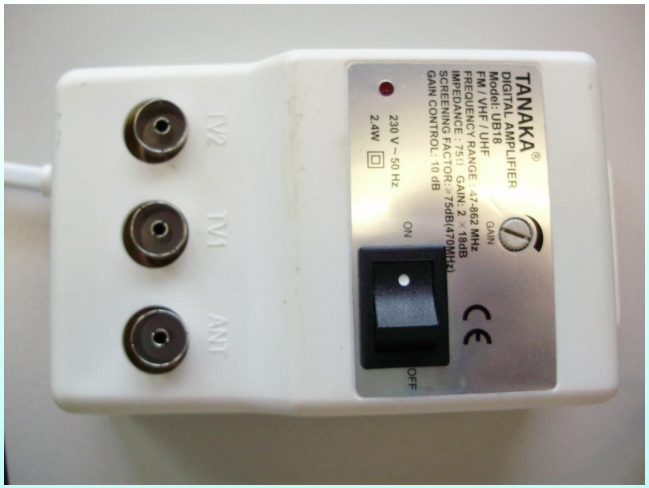
- Single - layer PCB
  * Typical thickness : 1.6mm, 1.2mm, 0.8mm, 0.6mm
  * Cheapest
  * Sample Turn-around time - about 3 days
  * Component mounting occupies most area
  * Most difficult to design



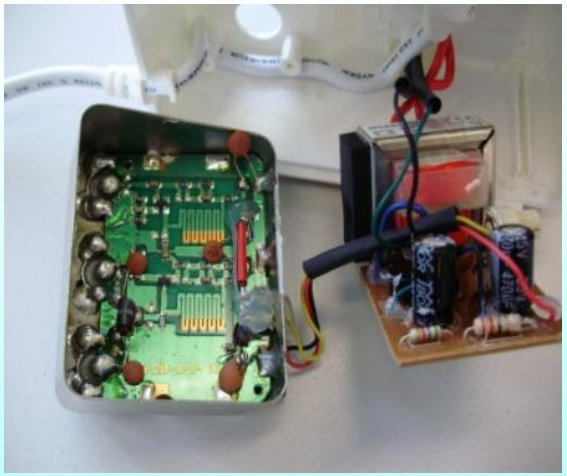
Layer Stackup Assignment

- Single - side PCB
  * Ground and power routing is very critical
  * Larger current circuits - closer to power source; low noise circuits - far from power source
  * Metal shield serves as auxiliary ground

TV signal booster



RF amplifier + Power Supply

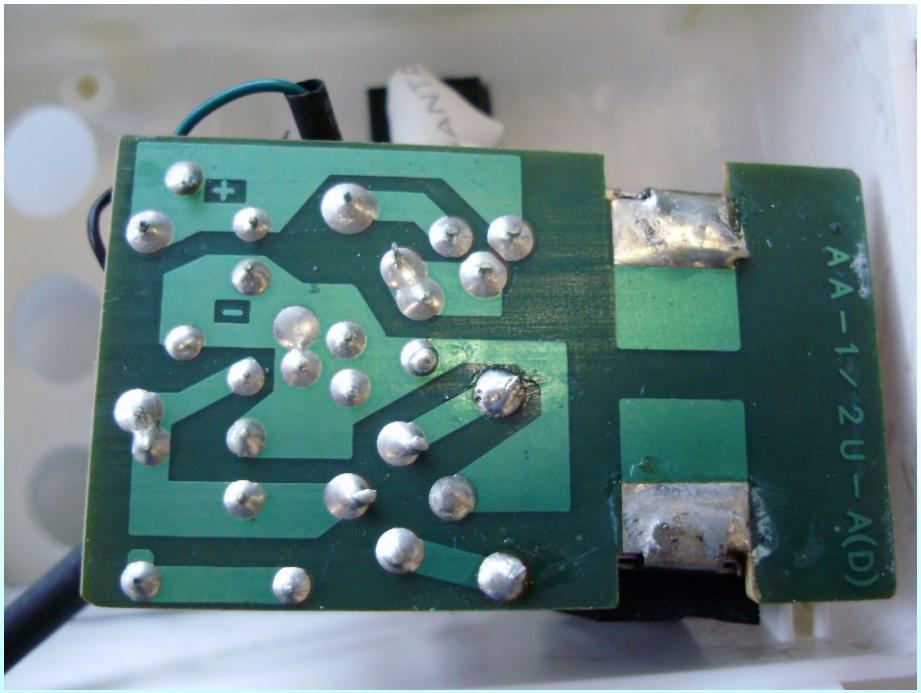


RF amplifier in a shield box

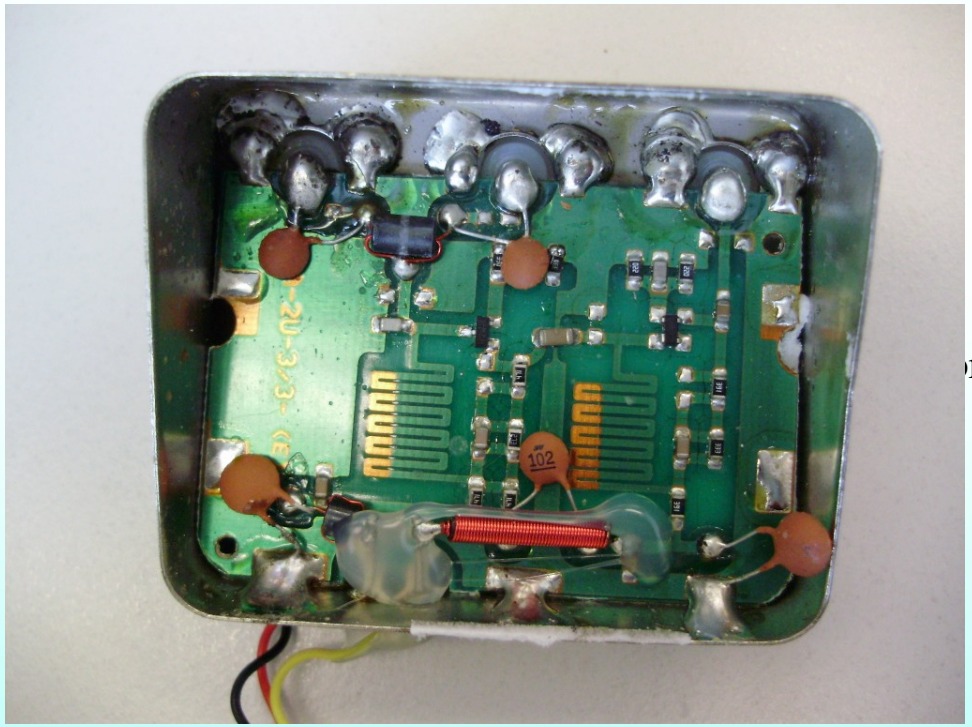
Layer Stackup Assignment

- Single - side PCB

Safety issue
on AC board

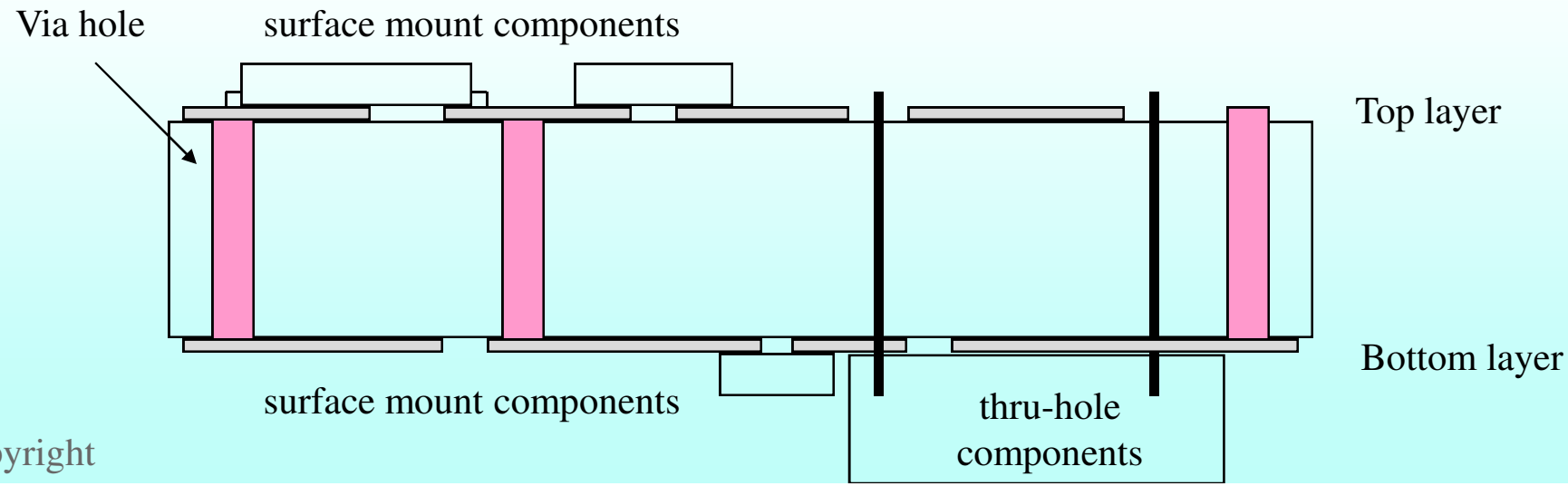


SMT + Lead
type components



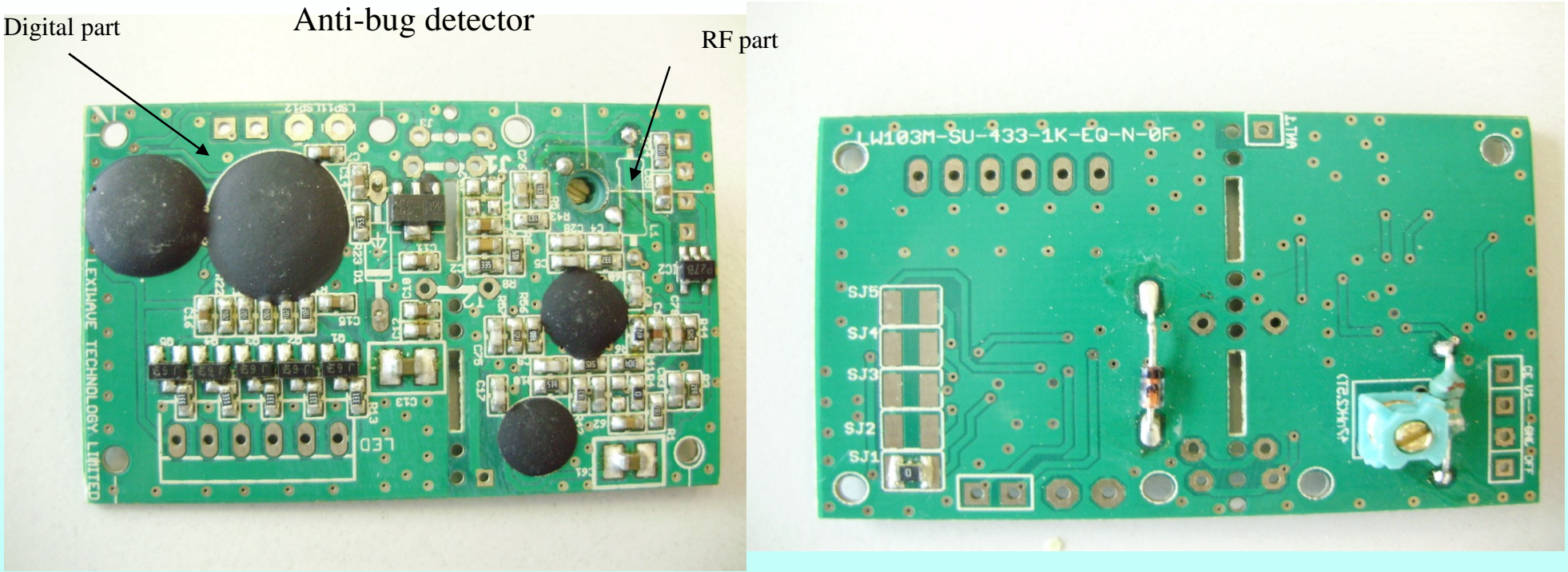
Layer Stackup Assignment

- Double - side PCB
  * Price competitive
  * Sample Turn-around time : 1 week
  * Top layer : component mounting and major signal tracings
  * Bottom layer : primarily with ground plane
    power trace
  * Put SMD / TH mixed component design on one side to save production cost



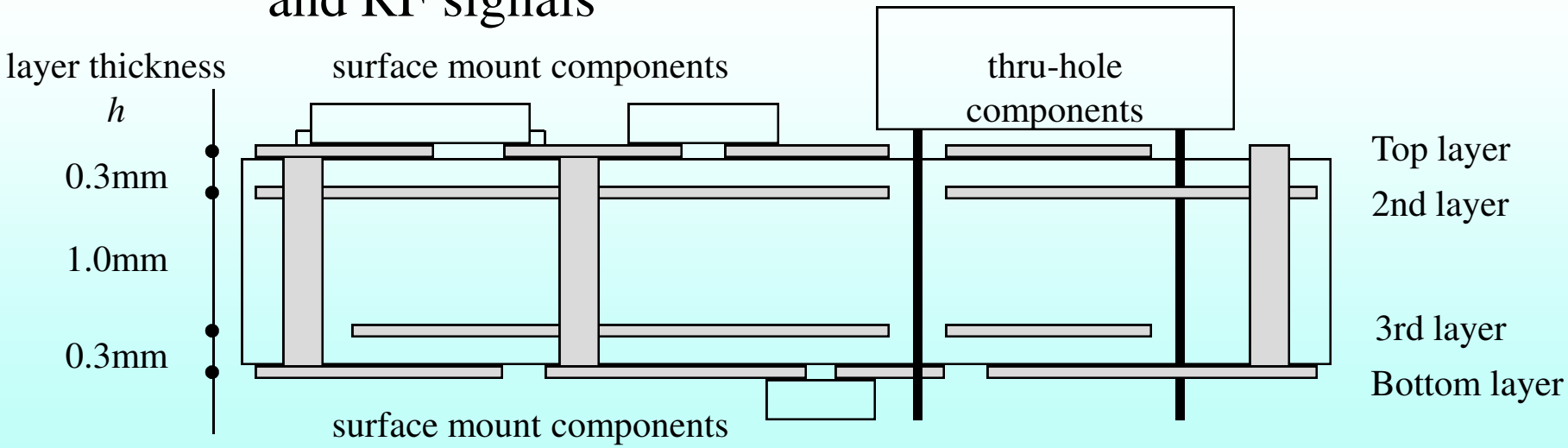
Layer Stackup Assignment

- Double - side PCB
  * Put component and route traces on one side
  * leave a good, big ground plane on the other side
  * Divide into sub-circuits



Layer Stackup Assignment

- 4 - layer PCB
  * Top layer : major component, major signal routing
  * 2nd-layer : main ground plane
  * 3rd-layer : power plane & minor signal routing
  * Bottom layer : minor component, auxiliary ground and signal routing
  * Commonly used for most applications with digital, analog and RF signals



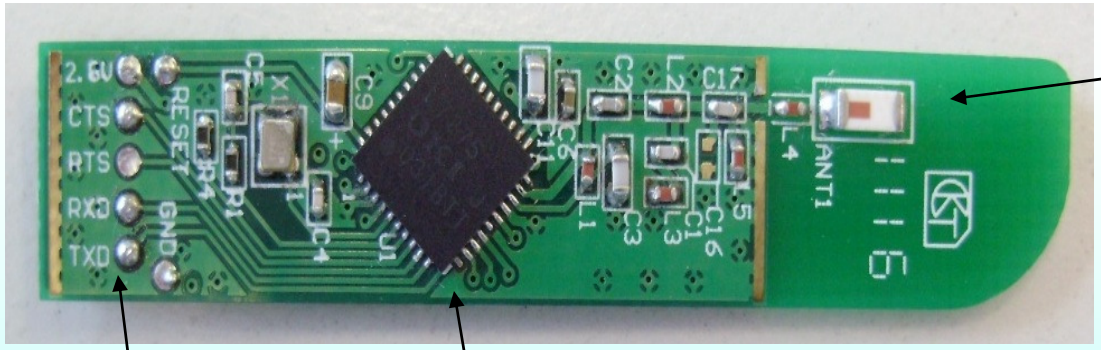
Performance comparison

Type	Price	Performance	Application
Single - side PCB	X1	Poor	Single circuit type
Double - side PCB	X2	Reasonable	Analog, Digital, RF
4 - layer PCB	X4	Good	Optimal for RF
6 - layer PCB	X6	Good	Mixer-mode with higher complexity, microwave striplines

Component Placement

Priority of RF PCB design

1. Antenna
2. Partitioning of different circuits
3. Vdd and ground placement
4. Trace minimization and board area utilization

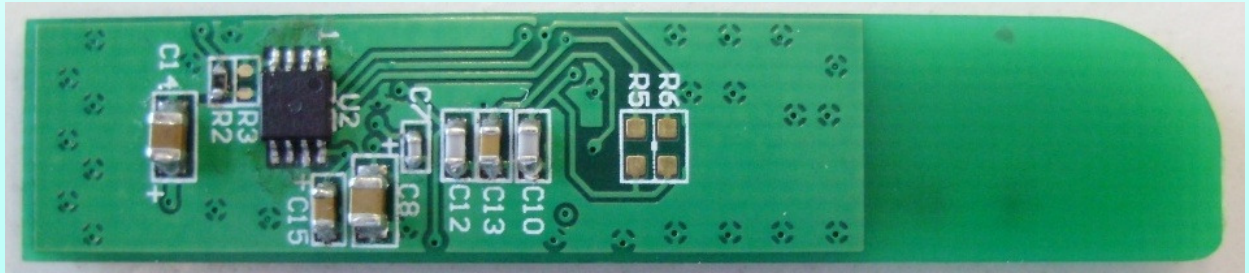


Chip Antenna

2.4GHz Zigbee Wireless Module

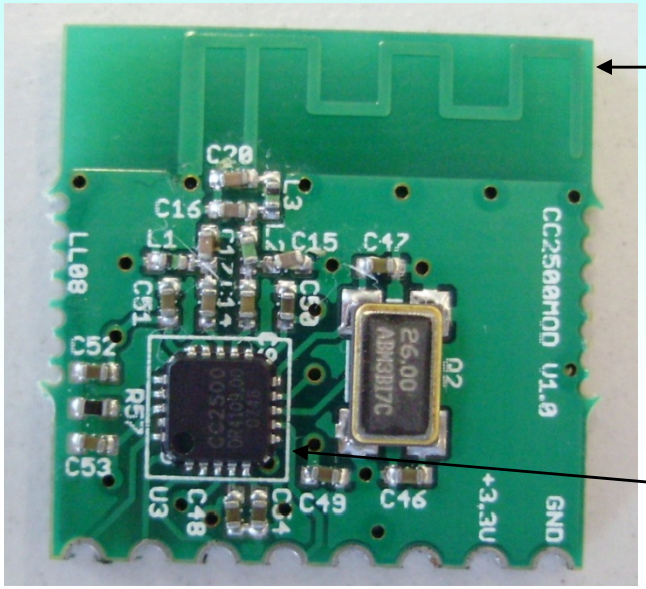
Host MCU interface

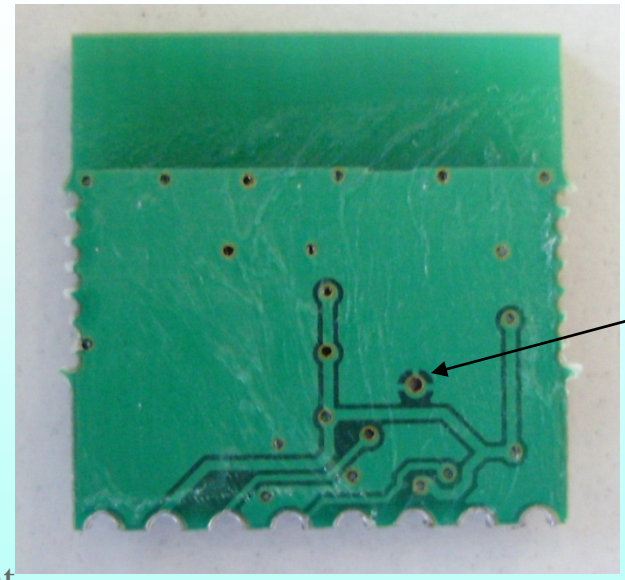
Transceiver



Component Placement

- Identify and segment groups of circuits
  - antenna, analog, digital, switching, audio. . . . . .
- Identify critical components
- Maximize grounding plane
- Optimize power routings
- Minimize traces and their lengths
  - Rotate components with different angles
  - Good I/O assignment
  - Optimize PCB shapes or mounting holes
  - use daughter board





Tips of Component Placement

- Place components as close to Integrated Circuits as possible with the priority of RF, IF and audio components
- Put the components with more interconnections close to each other
- Proper bus / ports assignment to shorten trace length and avoid cross-over

Tips of Component Placement

- Signal Isolation - in any amplifier circuit, the input and output should be separated as much as possible to avoid any oscillation due to signal coupling.
- Do not put inductors / transformers too close
- Put neighboring inductors orthogonally
- Good component placement will ease routing effort

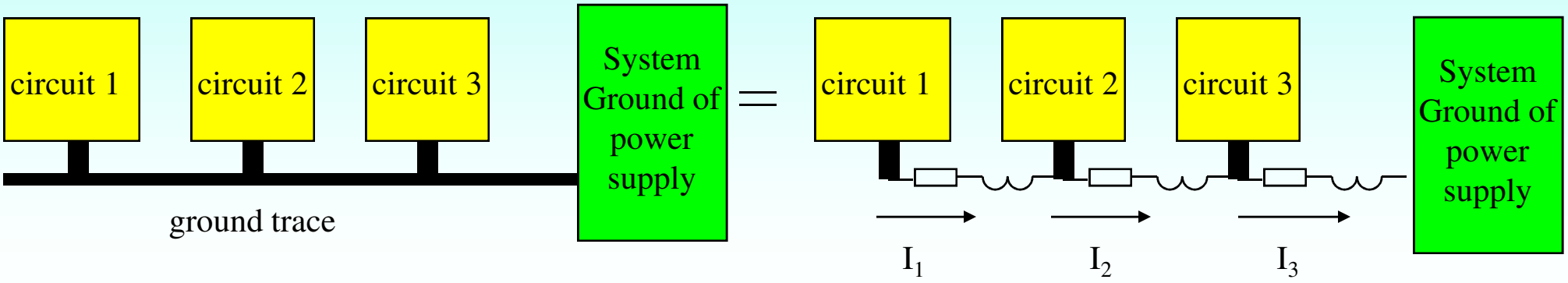
Grounding

- Types of Grounds
- Safety ground
  - A low-impedance path to earth
  - Minimize voltage difference between exposed conducting surfaces
  - Avoid electric shock
  - Protection against lightning and ESD
- Signal voltage referencing ground
  - zero voltage reference of a circuit
  - current return path

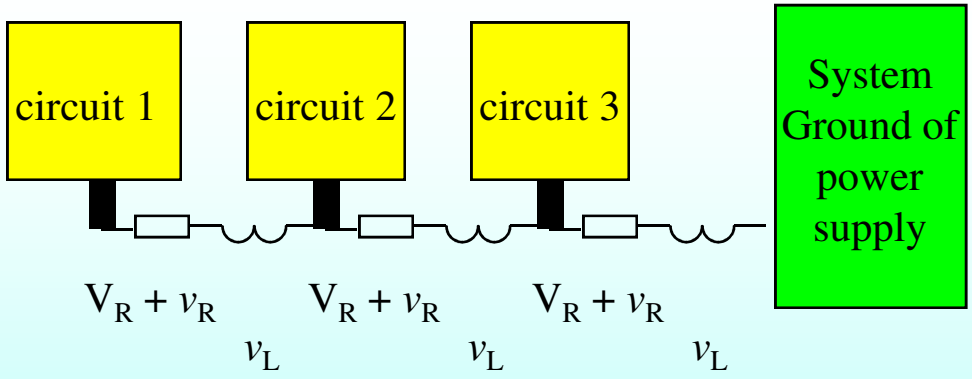
Grounding

- Good grounding:
  - Prerequisite of good RF and EMC performance
  - ground trace
    - as short and wide as possible
  - ground plane :
    - as large as possible
    - far away from antenna
  - ground via holes : 0.8mm board : 0.4mm diameter
    1.6mm board : 0.6mm diameter
  - Try to be a complete plane
    - avoid interruption from via, signal traces
  - avoid excessive copper pour and unused copper

Grounding Method



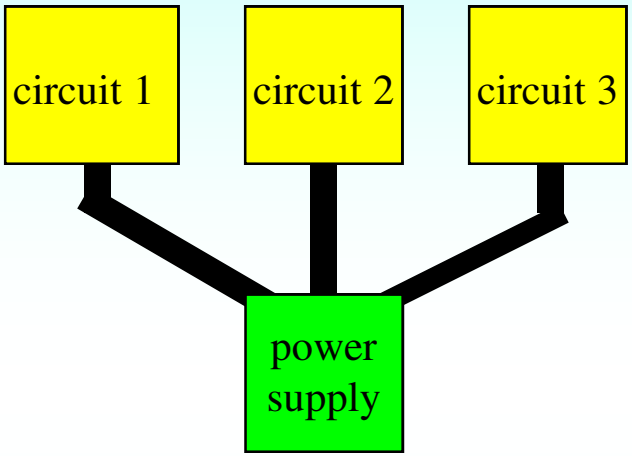
Equivalent circuit of ground trace (series connection)

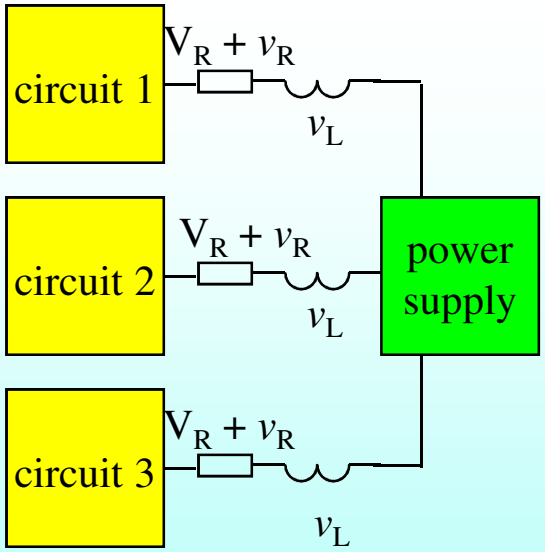


Noise and signal voltage induced by
ground current and imperfect ground connection,
additive noise and signal voltage affects all circuit blocks

Grounding Method

Star Connection

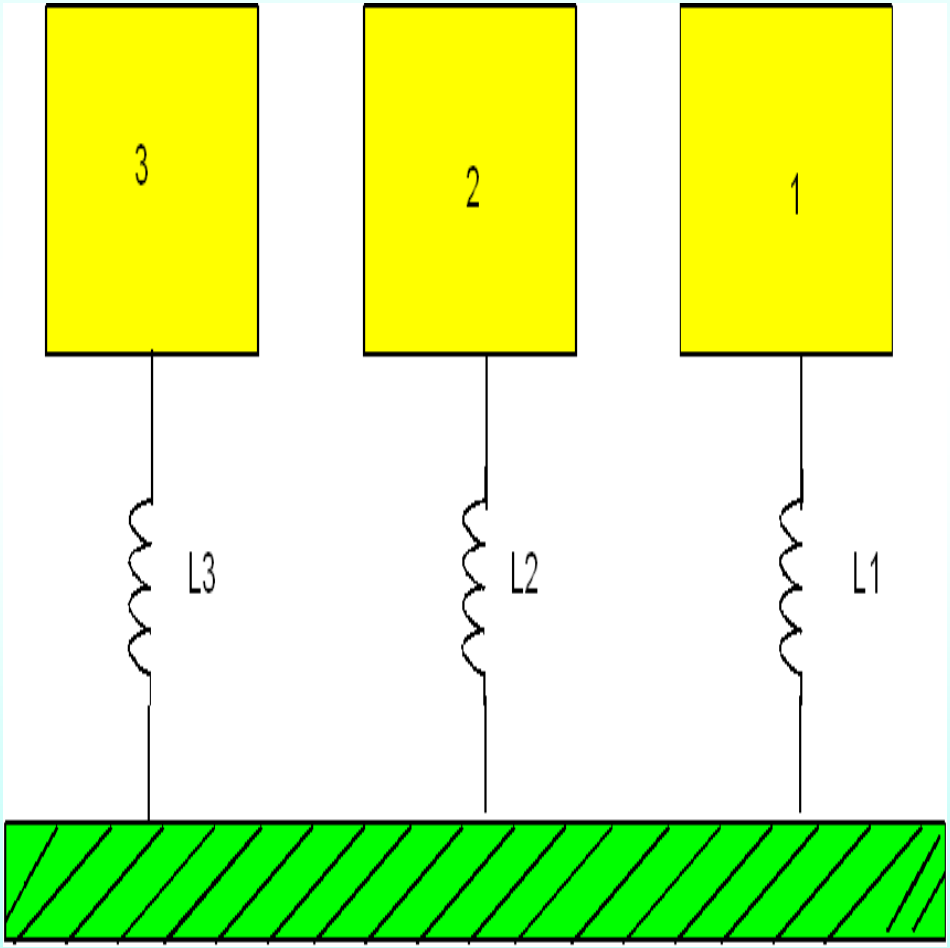




Minimize ground inductance and resistance,
Reduce induced ground noise voltage,
Minimize additive ground noise voltage

Grounding Method

Multipoint Grounding Connection



LPKF Laser & Electronics

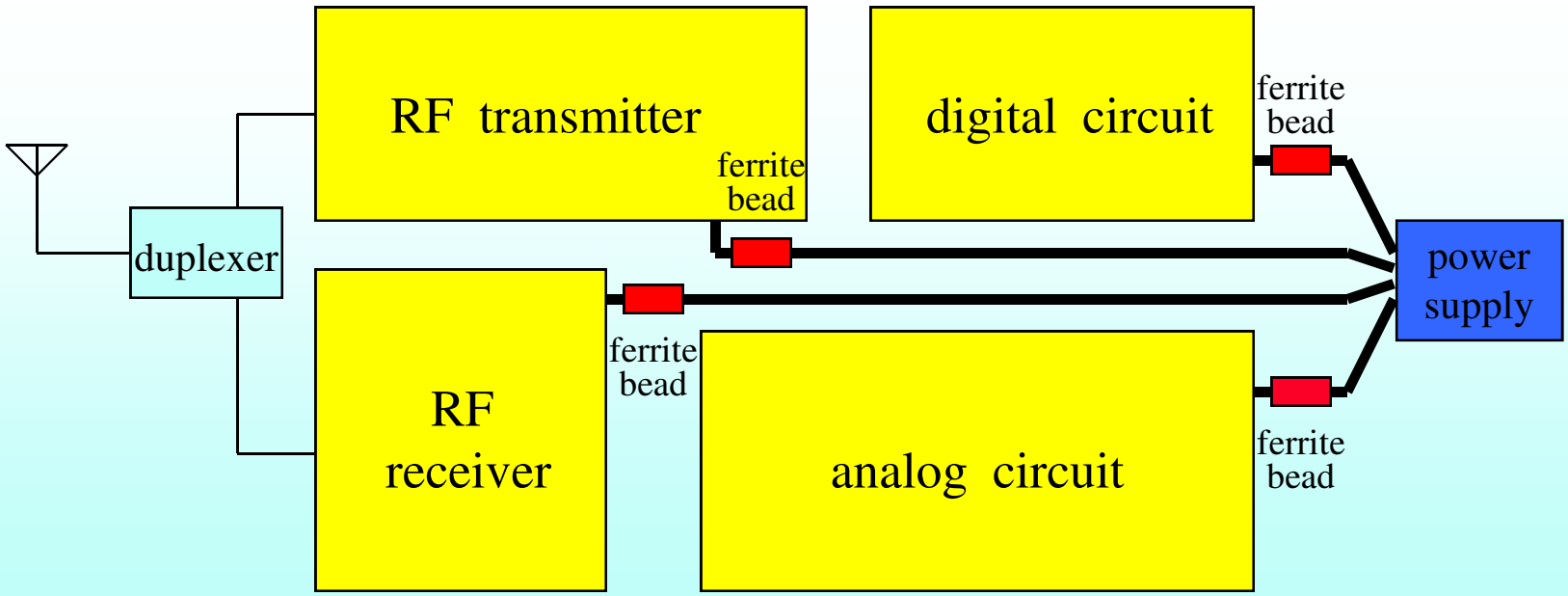
AWR A National Instruments Company™

Power Routing and Power Plane

- Power plane
  * treat the power plane the same as ground plane
  * Use ferrite beads for decoupling
- Power routing
  * Decoupling of power lines is a must
  * Place higher current or high switching circuit closed to the power supply
  * Separate power trace for separate sib-circuit

Power Routing and Power Plane

- " Star " type connection , work with **GOOD** ground plane
- Put ferrite bead near the sub-circuit
- Printed inductors and printed capacitors can be used above 1 GHz



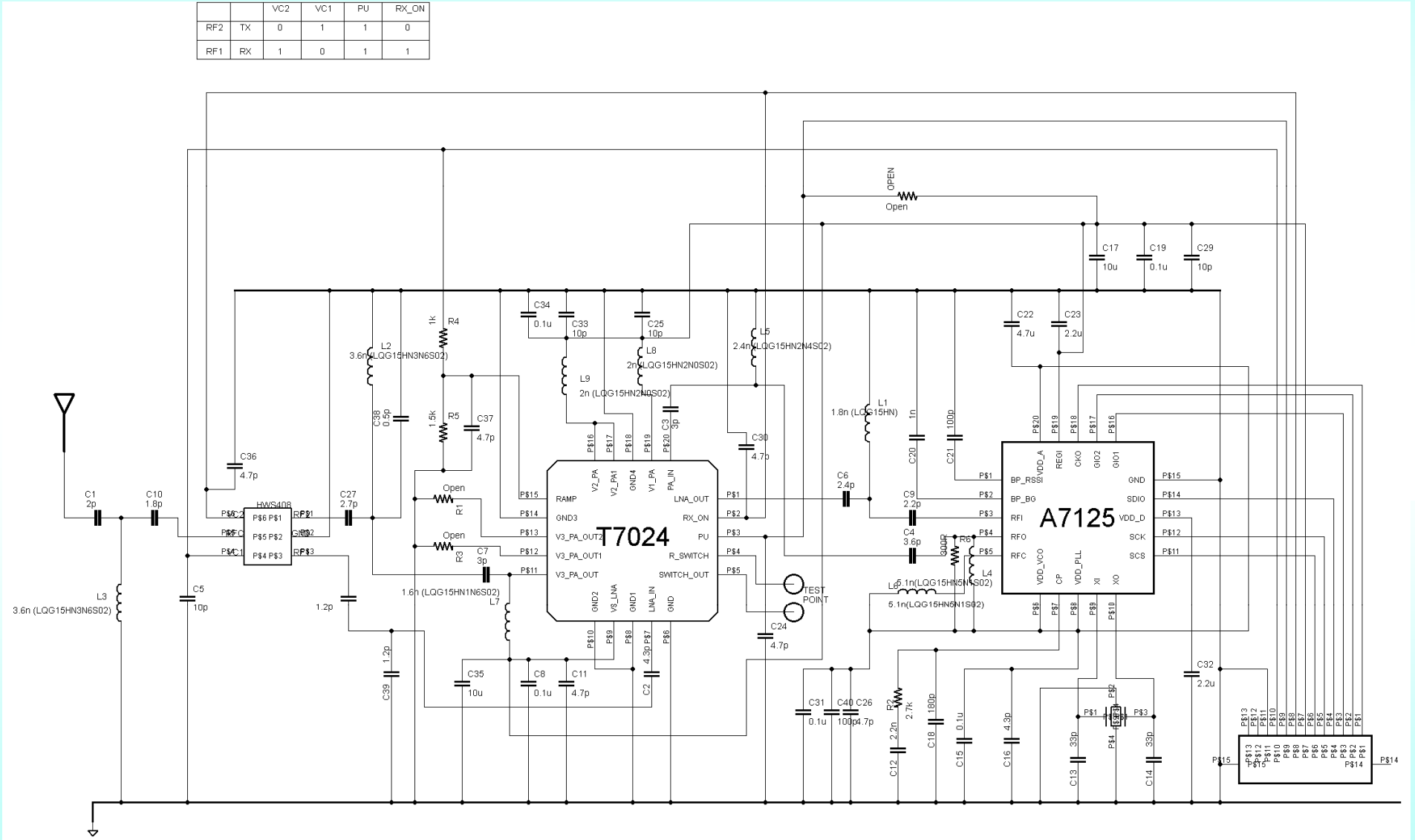
Bypassing & Decoupling

- Prevent energy transfer from one circuit to another
- Decoupling capacitors provide localized source of DC power and minimize switching voltage or current propagated throughout the PCB
- Location of decoupling components is critical
- Common mistakes
  - wrong component location on schematic diagram
  - Wrong component types
  - Lack of routing information between blocks
  - Un-necessary long traces

Bypassing & Decoupling

- Put decoupling components on optimal locations
- Decouple each circuit block individually
- Decouple each supply pin individually
- VCC decoupling capacitors
  - Require three types
    - 10~100uF for audio frequency
    - 0.01u to 0.1uF for IF frequency
    - 30~100p for RF frequency
  - Place the RF one as close as possible to the chip
- Use the right decoupling component for the right frequency

Bypassing & Decoupling



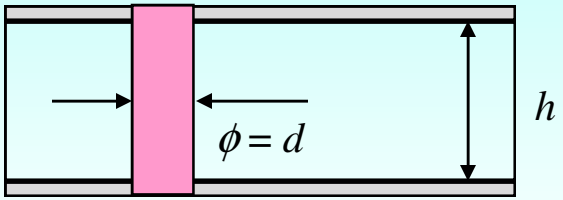
Via Holes

- Location
  - avoid cutting too much on the ground plane
  - do not put inside the SMD component pads
- Inner diameter :

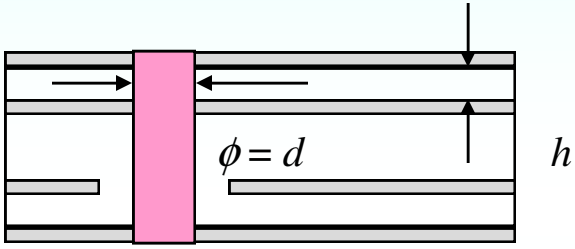
  Resistance proportional to $\pi \times d / h$

  0.8mm board : 0.4mm for grounding and power, 0.25mm for signal

  1.6mm board : 0.6mm for grounding and power, 0.35mm for signal
- Outer diameter : 0.2mm larger than inner diameter
- Quantity : depending on current and frequency
- Solder mask : cover via holes on both sides
- Use multiple via holes for critical ground



double layer



multi layer

Trace Routing

- Good component placement can shorten trace length and wider trace width
- Can minimize parasitic inductance, capacitance and resistance
  * proportional to trace length
  * inversely proportional to trace width
  * Avoid sharp corner on high frequency or ESD sensitive traces
- Minimize parasitic can achieve
  * higher circuit Q with higher performance
  * More controllable
  * wider tuning range and more stable

Tips of Trace Routing

- Routing on-grid
- Minimize stitches between layers
- Avoid sharp corner
  - Routing on 0, 90 degrees and prefer 45 degrees
- Maximize board space to leave space for trace routing
- If trace is long, line impedance will have to be controlled

Trace Routing

- Impedance-controlled trace
  - * High frequency input/output connection
  - * As a high frequency distributed circuit element
    - * Micro-stripline, stripline, coplanar stripline
  - * Input/output matching element
  - * Require information on PCB material and geometry
    - * Er (4.6 for FR-4 material)
    - * Copper thickness, board thickness
- PCB Antenna
  - * shorter trace, smaller effective antenna aperture

Shielding

- Effective solution for EMI and EMC
- Understand the sources and relation of interference
- Circuit partitioning :

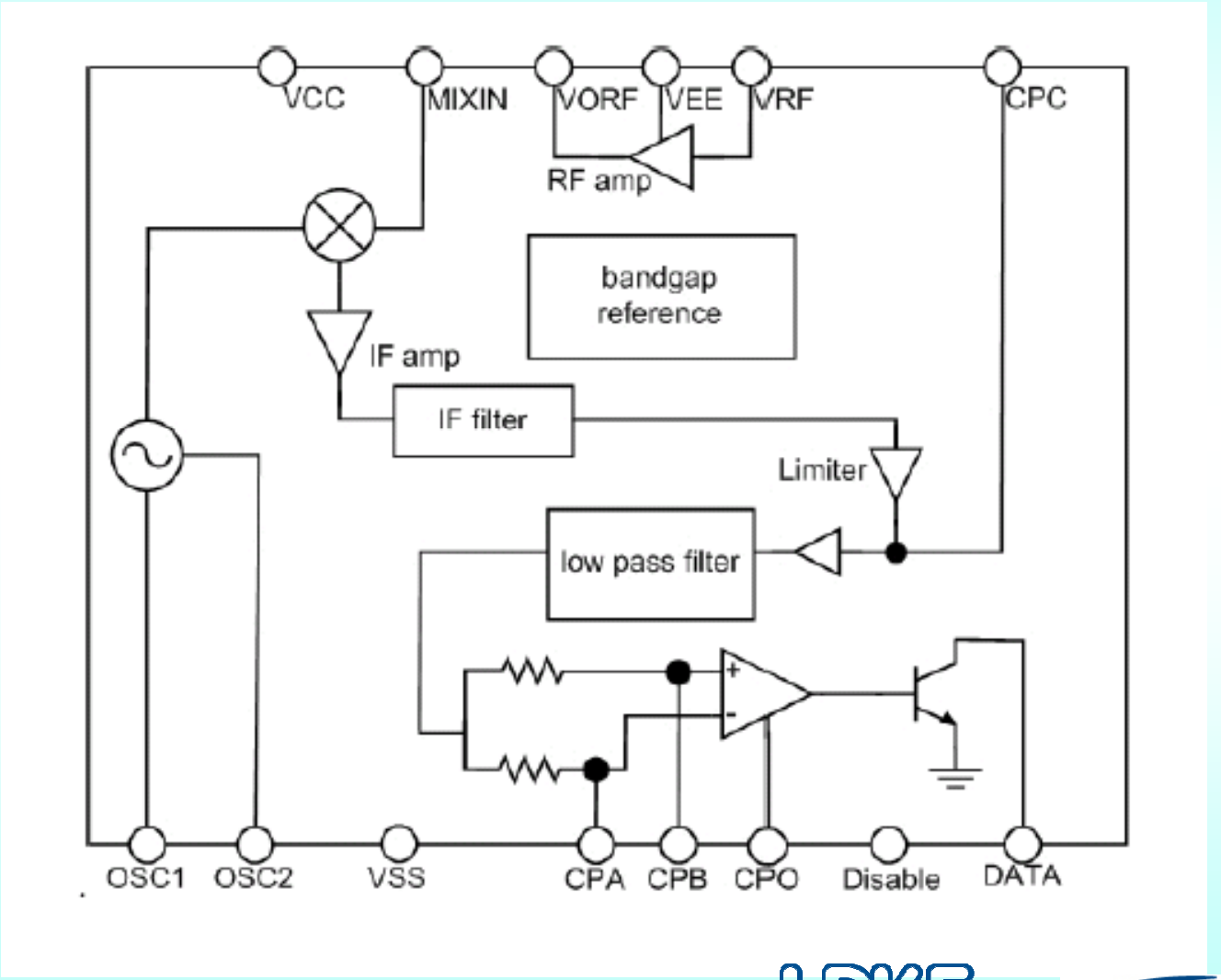
  RX : LNA, mixer PLL and IF amplifier

  TX : oscillator, PLL, buffer and power amplifier
- Material
  - Metal sheet
  - Conductive Coating
- Re-openable cover for repair
- Opening for Alignment and test points
- More contact surface for cover

PCB Design for LW106M

- LW106M from Lexiwave
  - 310MHz to 440MHz Receiver Module
    - Using LW106 RFIC receiver chip
    - Single-superheterodyne receiver
    - High sensitivity, -90dBm
    - RF (400MHz), IF (MHz) and Low frequency (KHz)
  - High selectivity
  - Applications
    - Remote controllers
    - Wireless door bells
    - Car alarm system

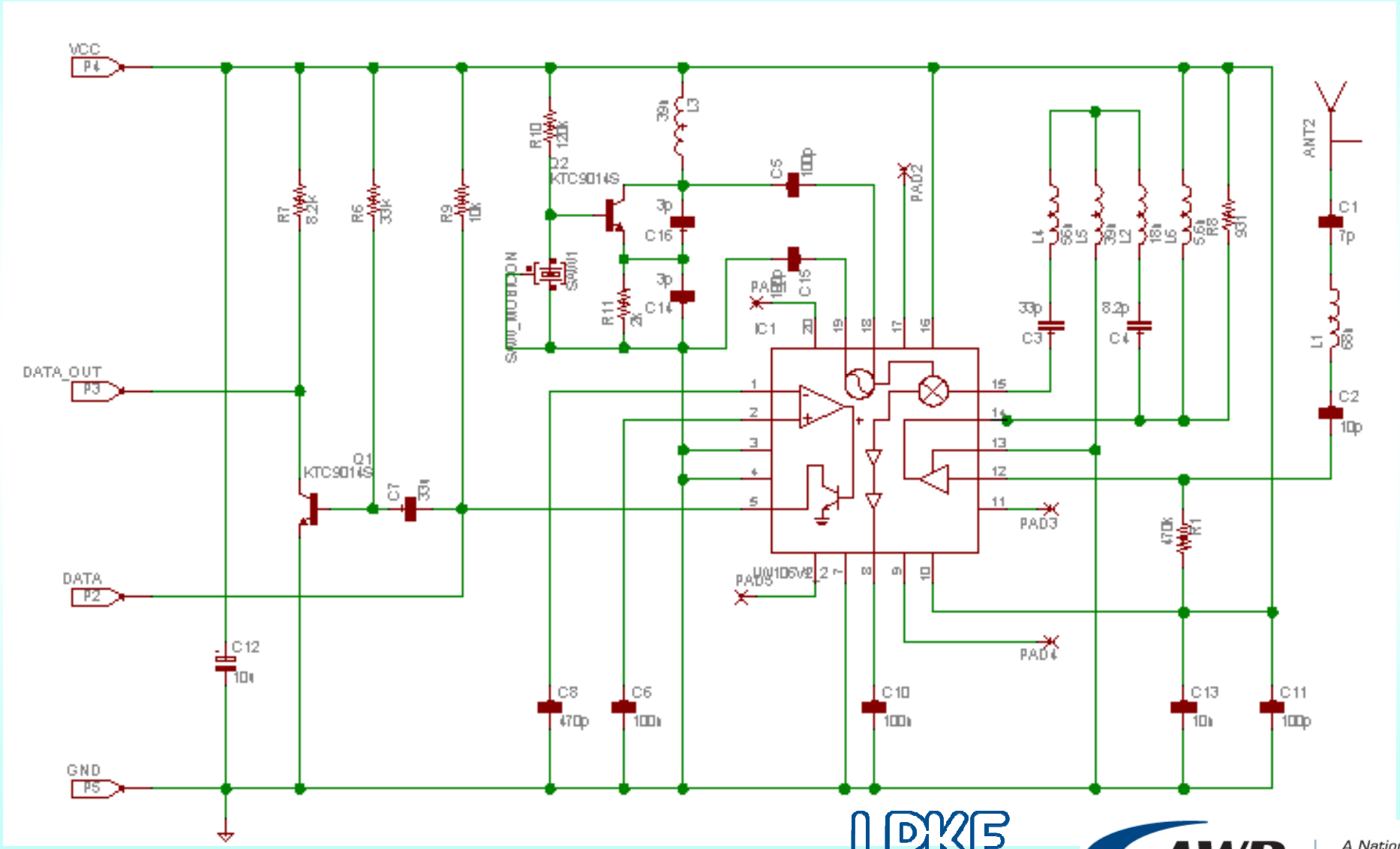
LW106 Block Diagram



LPKF Laser & Electronics

AWR A National Instruments Company™

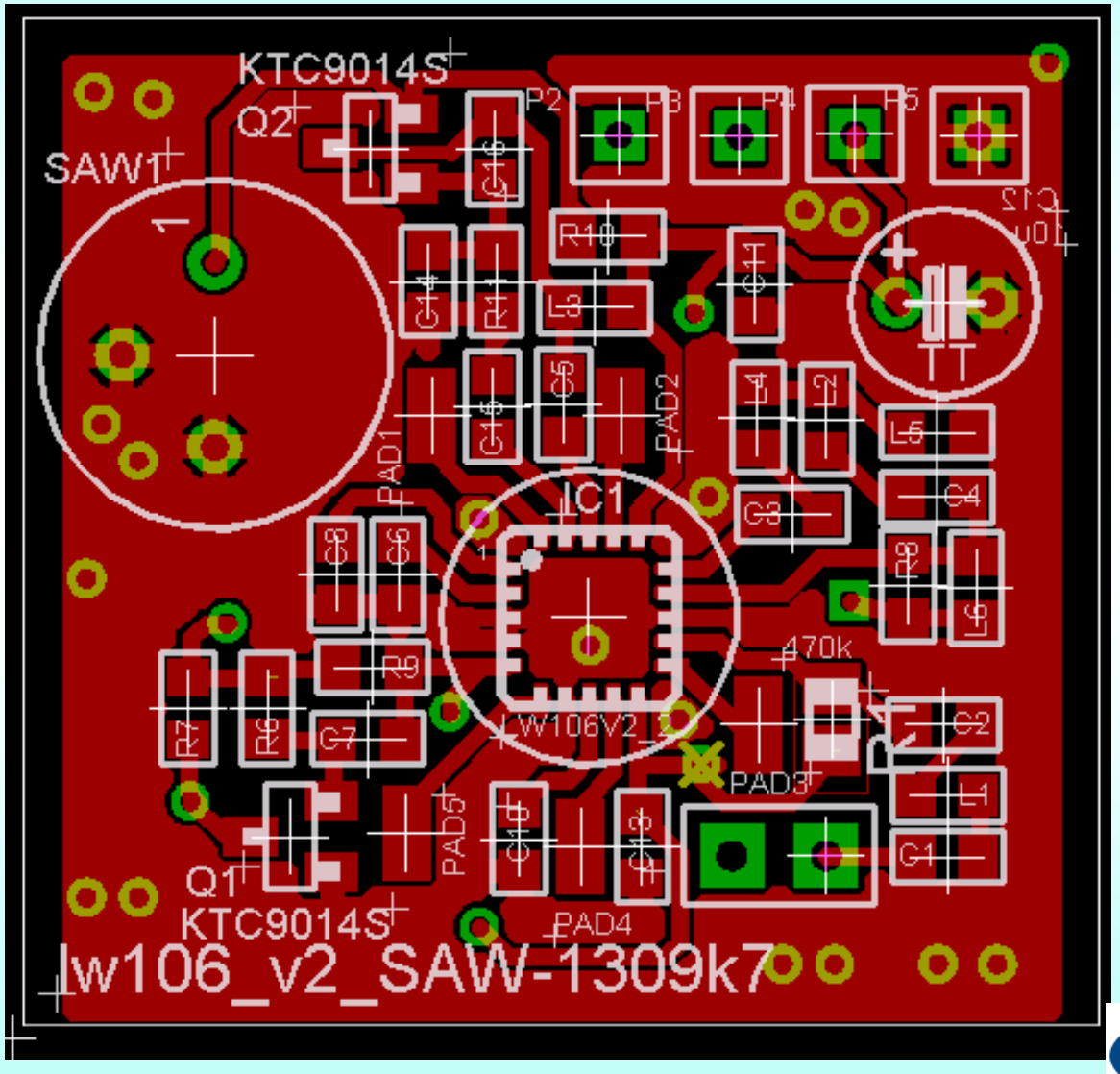
LW106M Schematic Diagram



LPKF Laser & Electronics

AWR A National Instruments Company™

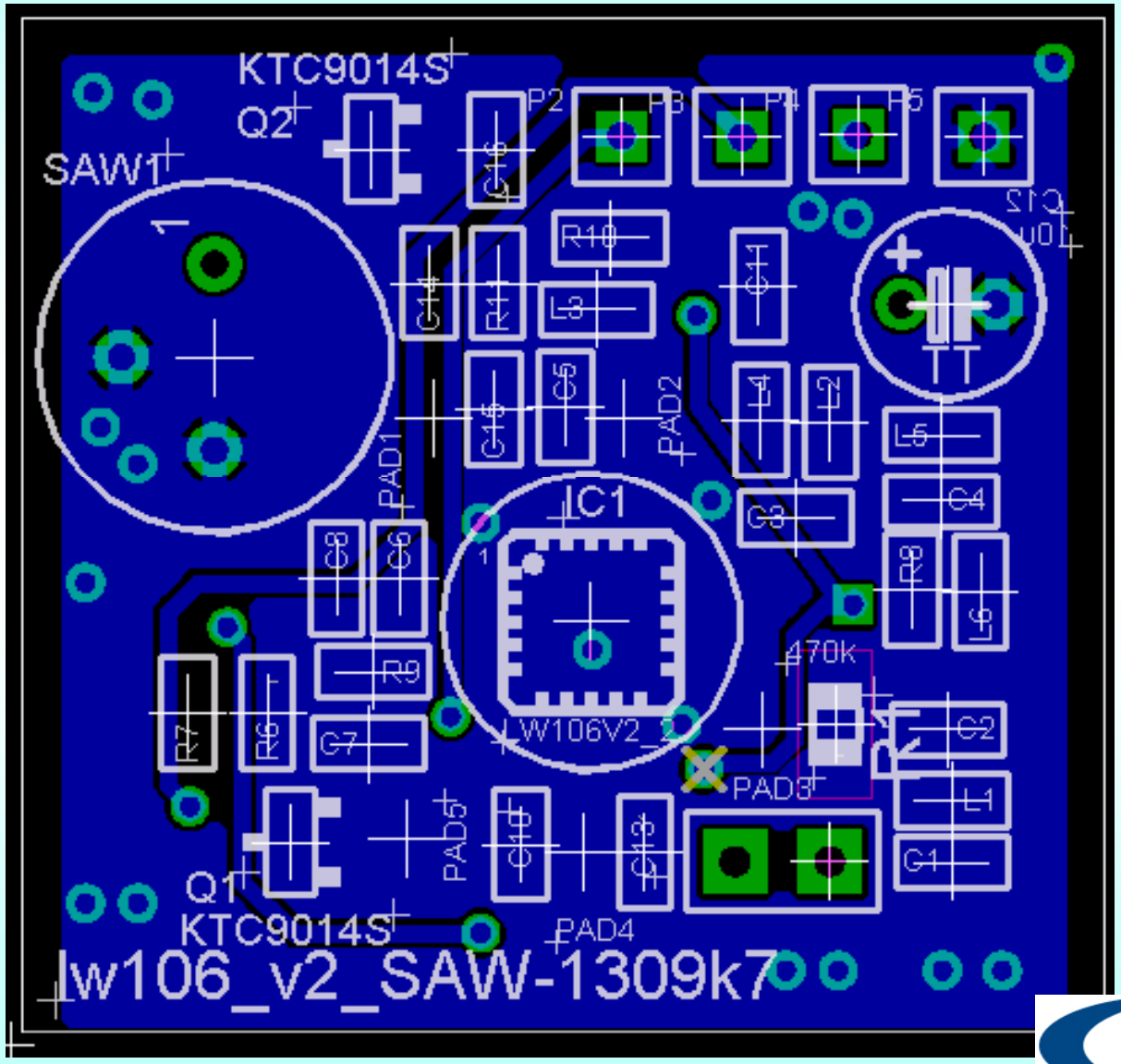
LW106M PCB Top Layer



LPKF Laser & Electronics

AWR A National Instruments Company™

LW106M PCB Bottom Layer

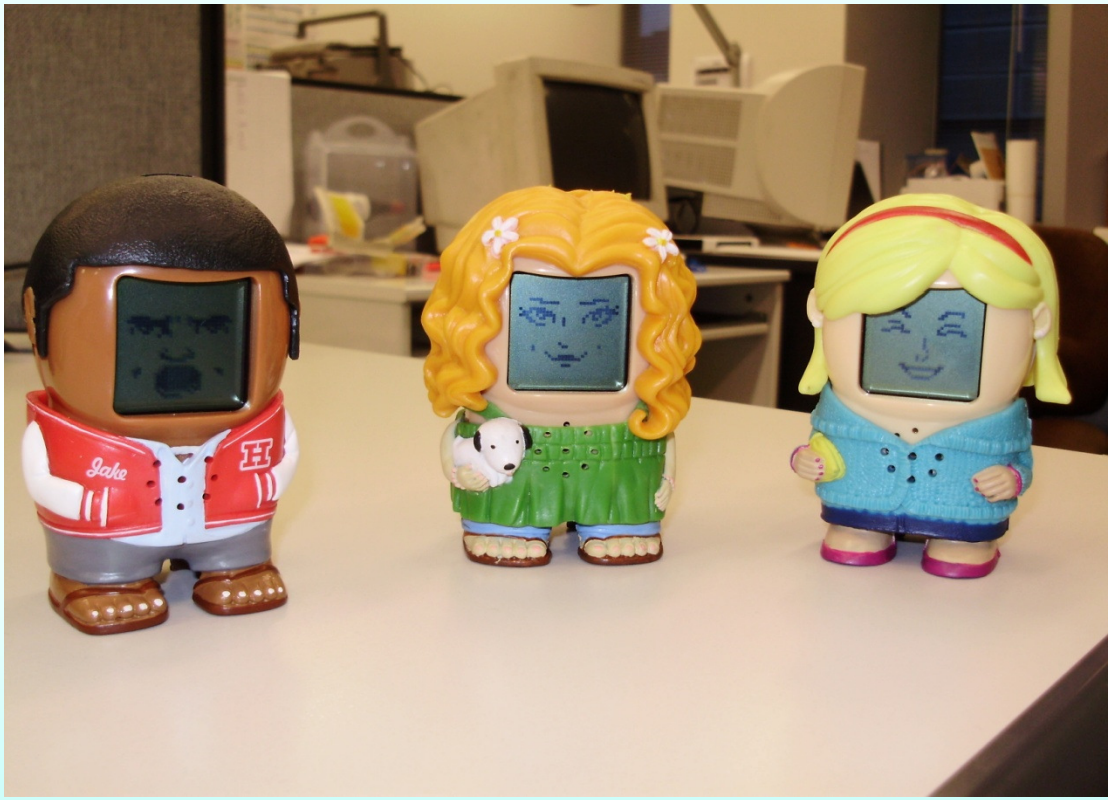


LPKF Laser & Electronics

AWR A National Instruments Company™

Case Study – Interactive Toy

- Interactive Doll – Huru-Humi
  - Bi-directional RF datalink
    - Communicate with each other
    - Voice recognition
    - Link up to 6 units
    - Short distance
  - On sale at
    - Wal-mart
    - Target
    - Toys “R” us



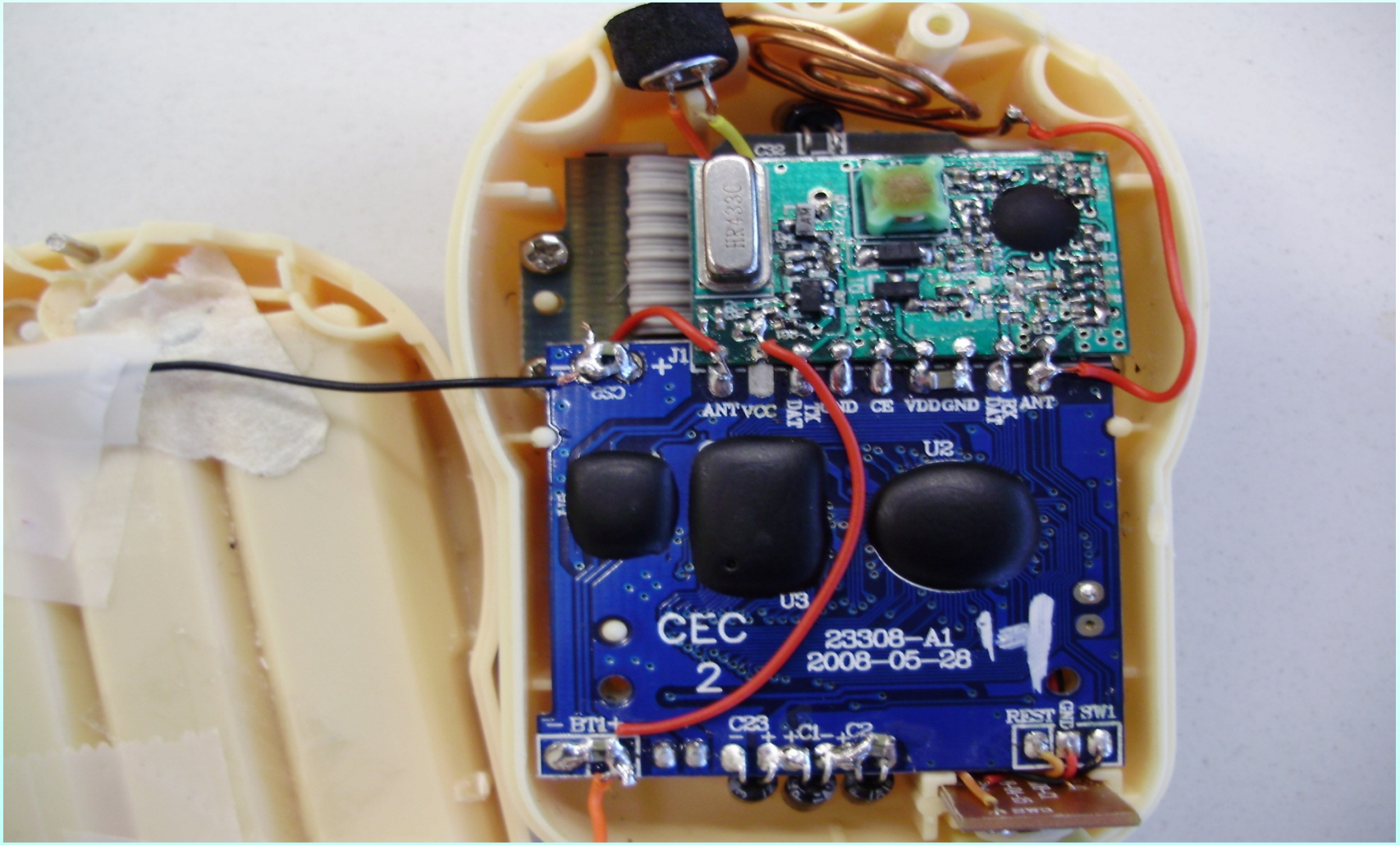
LPKF Laser & Electronics
AWR A National Instruments Company™

Case Study – Interactive Toy

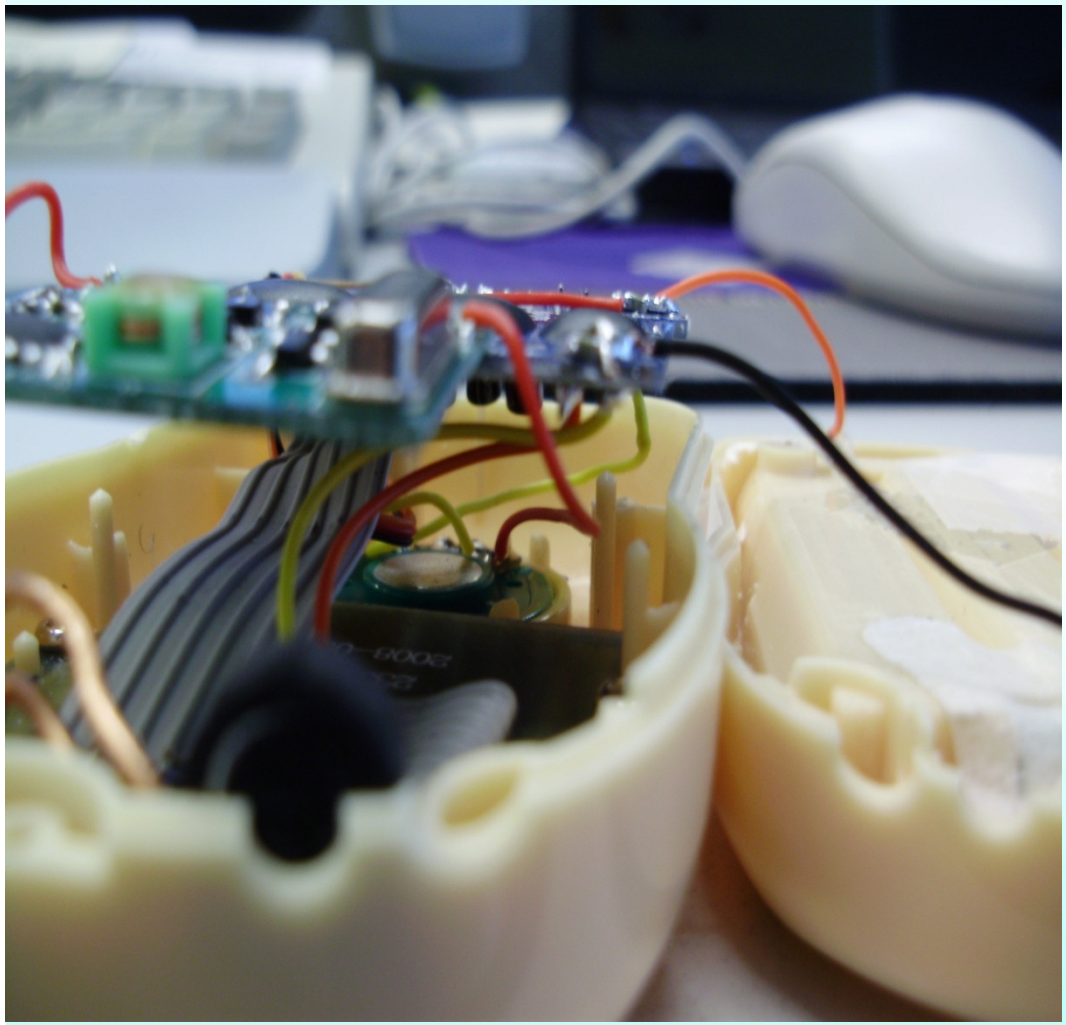
- Key Building Blocks
  - MCU
  - External ROM for speeches
  - MCU address extender
  - LCD driver and display
  - RF Transceiver Module
  - Audio amplifier
  - Microphone amplifier

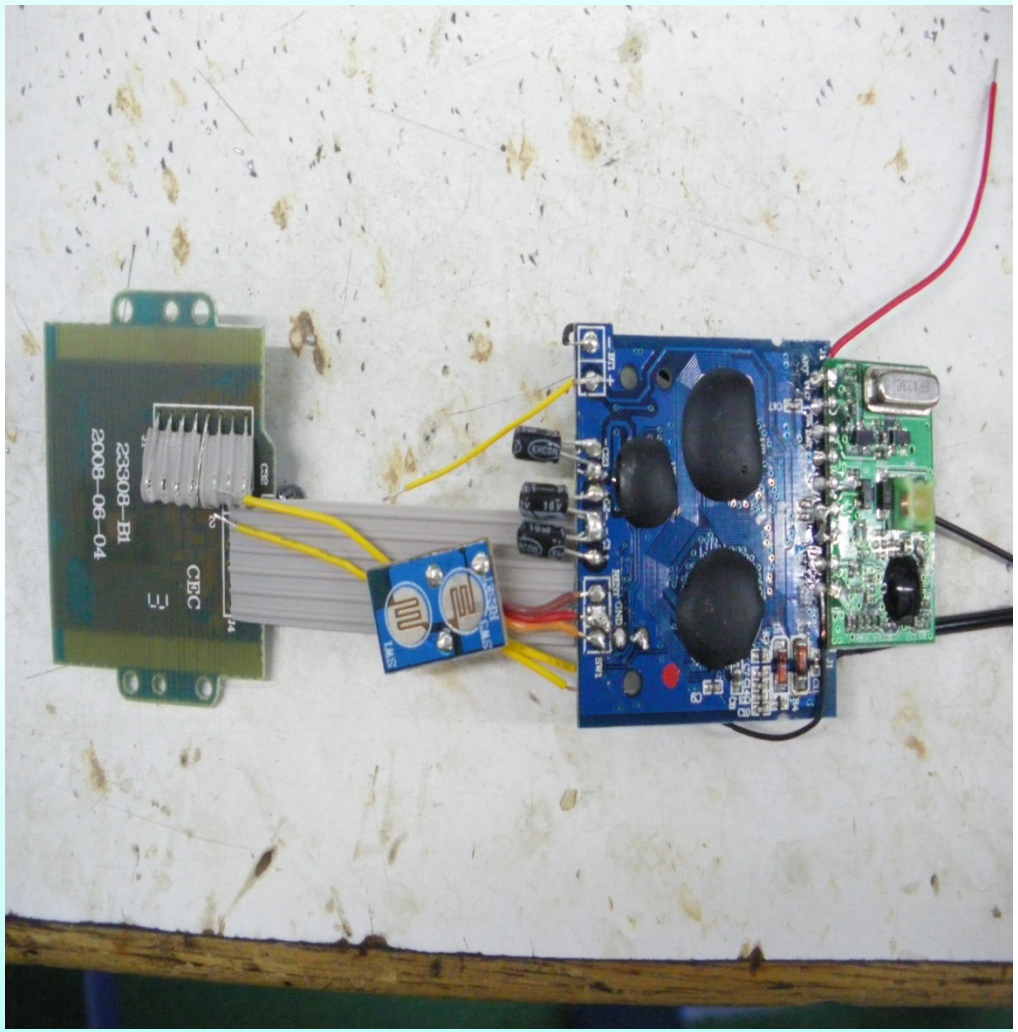
Case Study – Interactive Toy

- Original PCB – poor communication distance



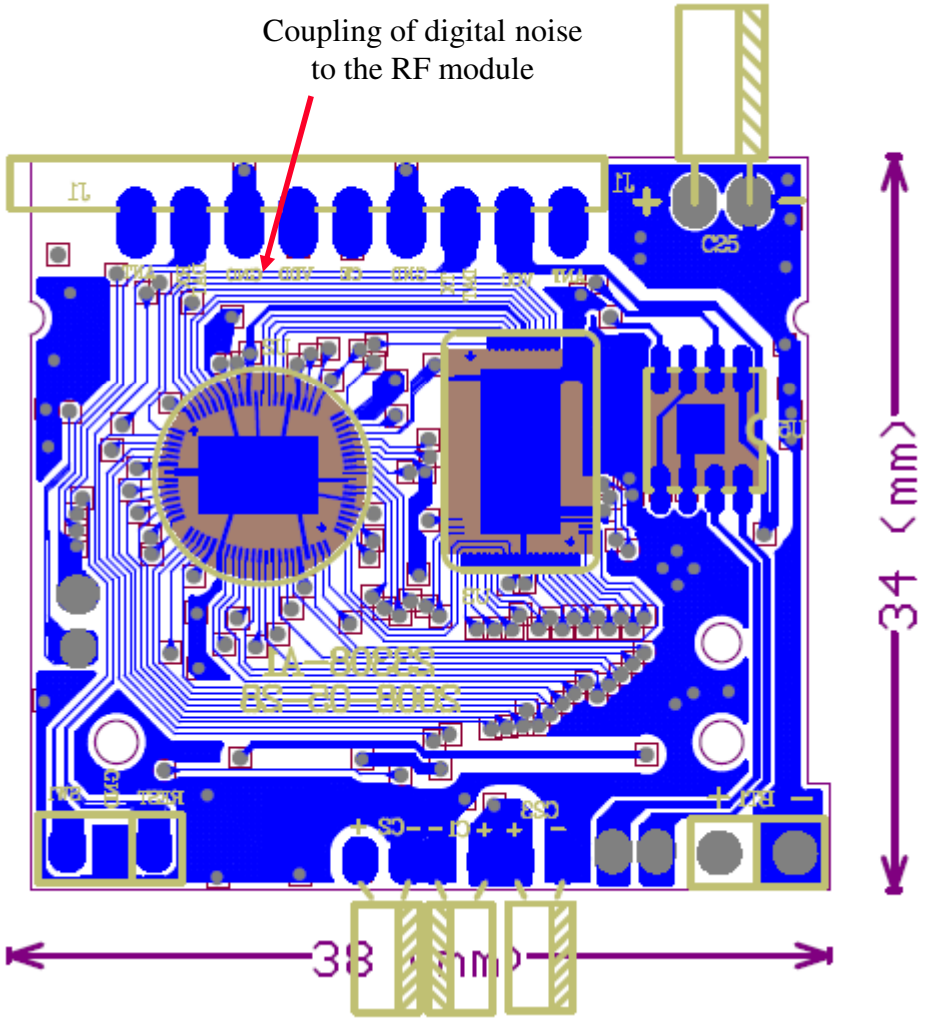
Case Study – Interactive Toy

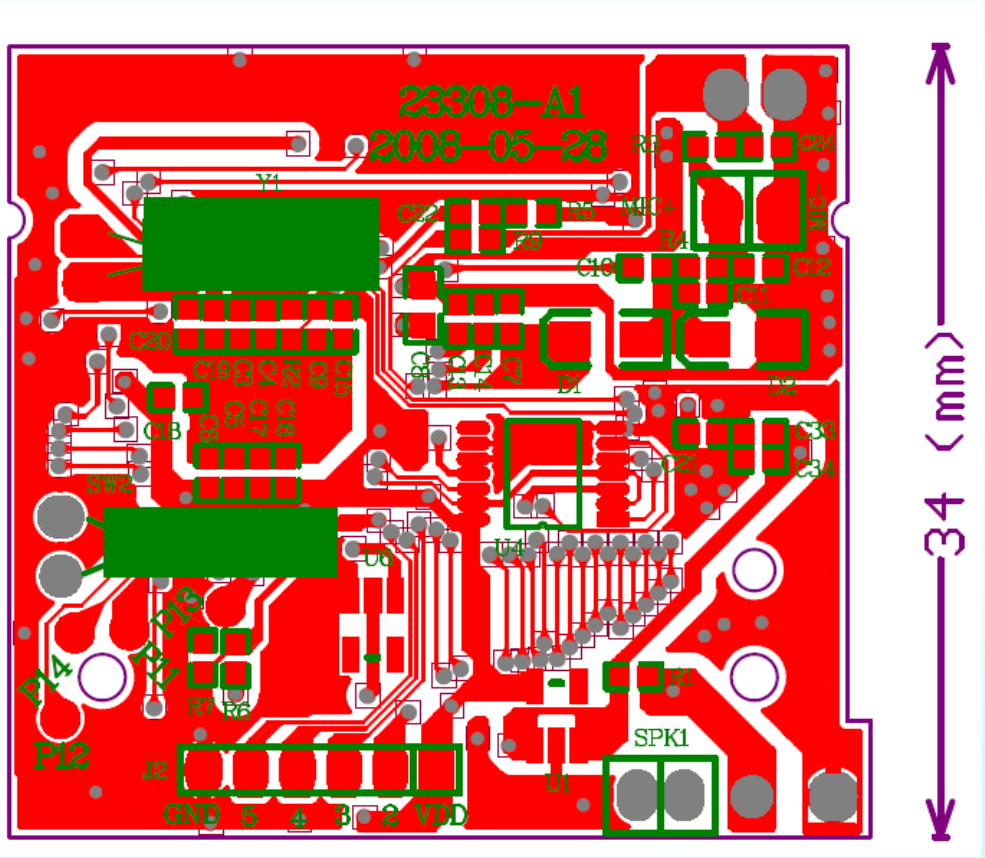




Case Study – Interactive Toy

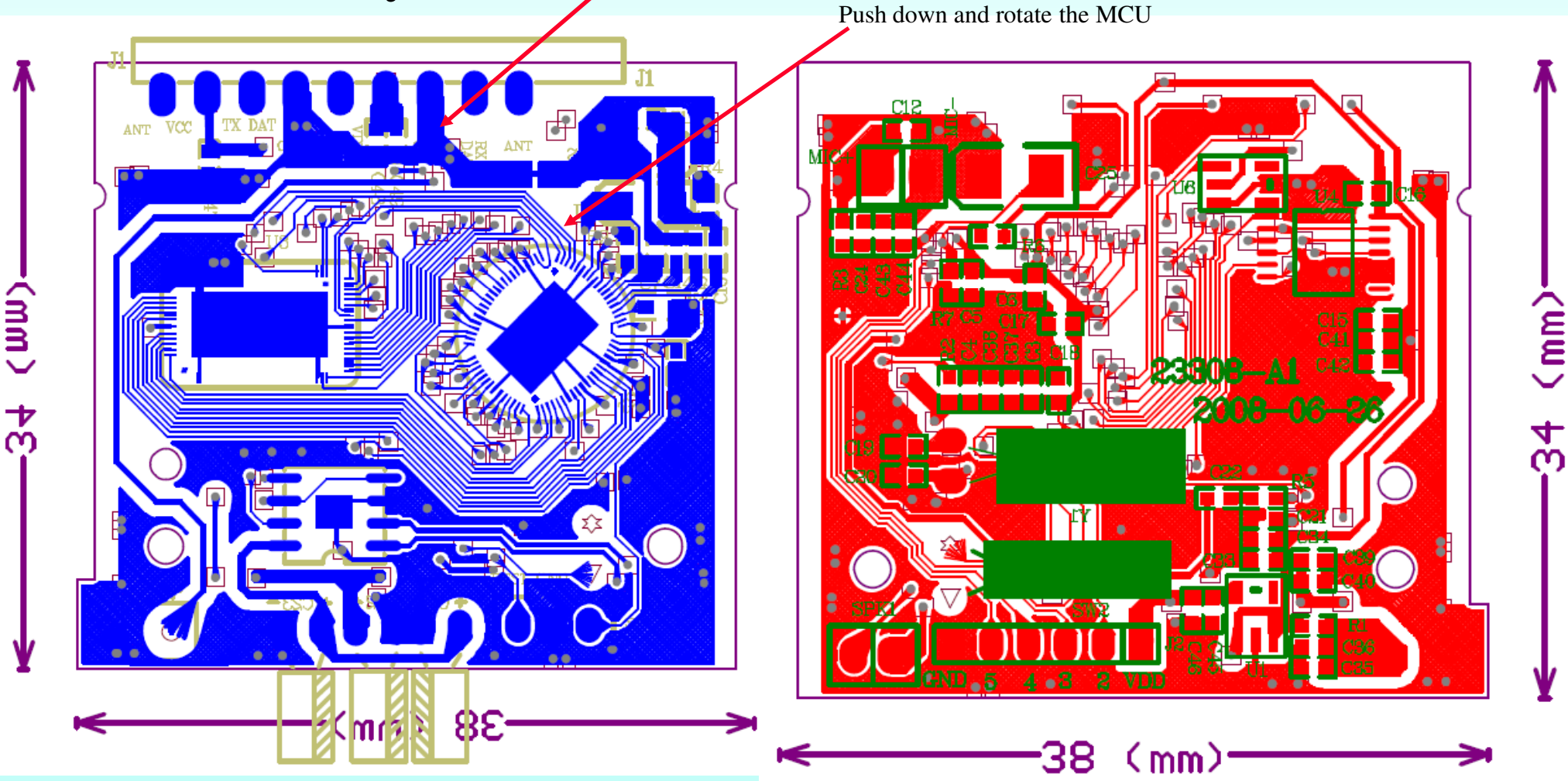
- Original Layout





Case Study – Interactive Toy

- Modified Layout



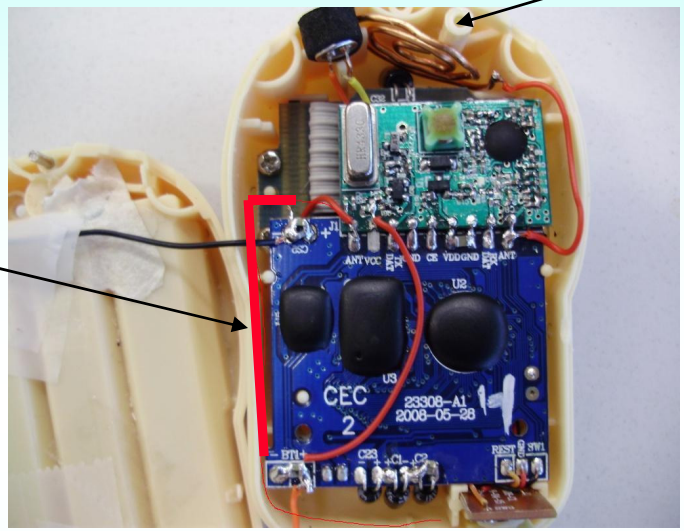
Case Study – Interactive Toy

- Antenna Structure

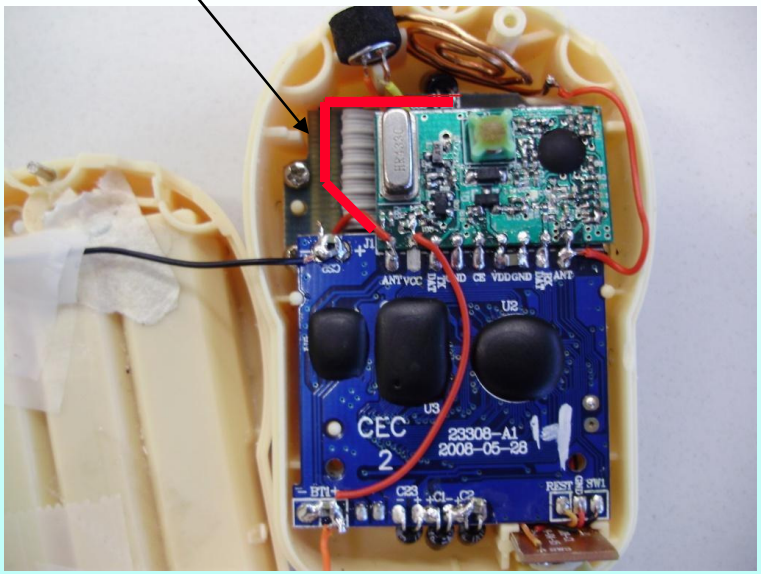
Improved version –

Spiral antenna

Original monopole antenna

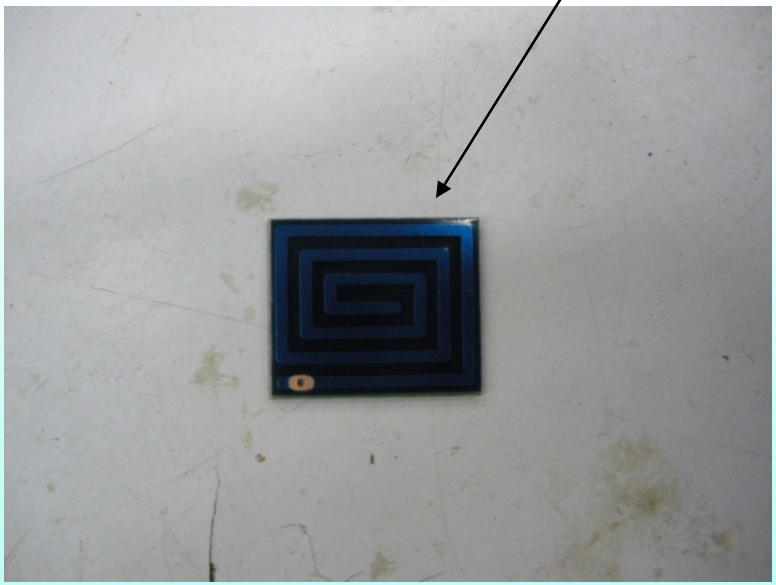


Another suggested antenna



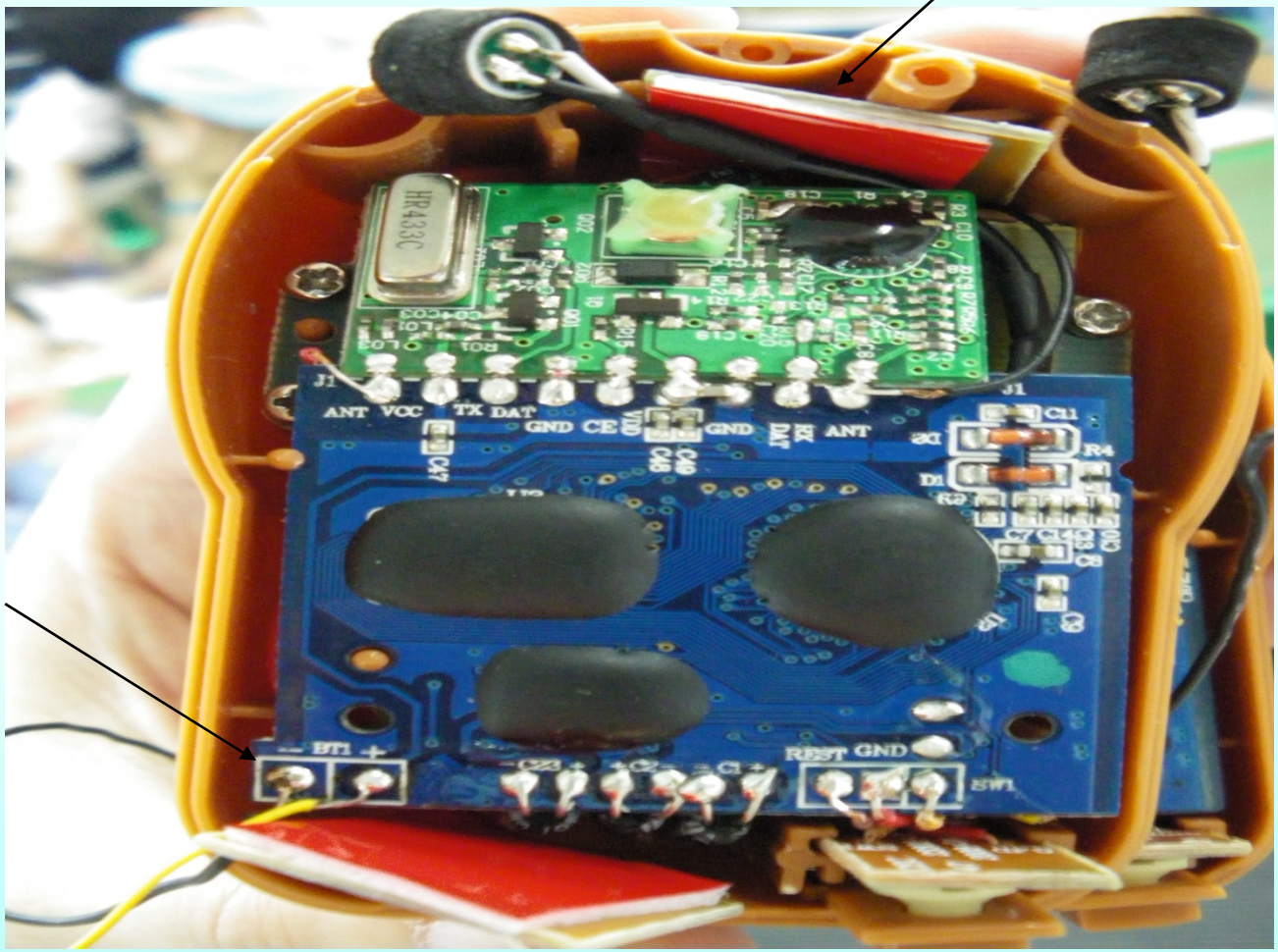
Final production version –

Spiral PCB antenna



Case Study – Interactive Toy

- Modified PCB



Conclusions

- RF PCB layout plays a crucial role on determining the success of the product
  * Electrical performance
  * EMI/EMC regulations
  * Stability and reliability
  * Design for mass production

Q & A

Thanks to our sponsors AWR and LPKF