RF PCB Design

Presented by: *Henry Lau, Lexiwave Technology, Inc.*
Sponsored by: *National Instruments (formerly AWR Corp.)*
October 15, 2015


NATIONAL
INSTRUMENTS™

NI AWR Software

Product Line Overview

NATIONAL INSTRUMENTS

NI AWR Design Environment - At a Glance

Software Product Portfolio

- Microwave Office - MMIC, RF PCB and module circuit design
- Visual System Simulator - Wireless communications/radar systems design
- AXIEM - 3D planar electromagnetic (EM) analysis
- Analyst - 3D finite element method (FEM) EM analysis
- Analog Office - Analog/RFIC circuit design

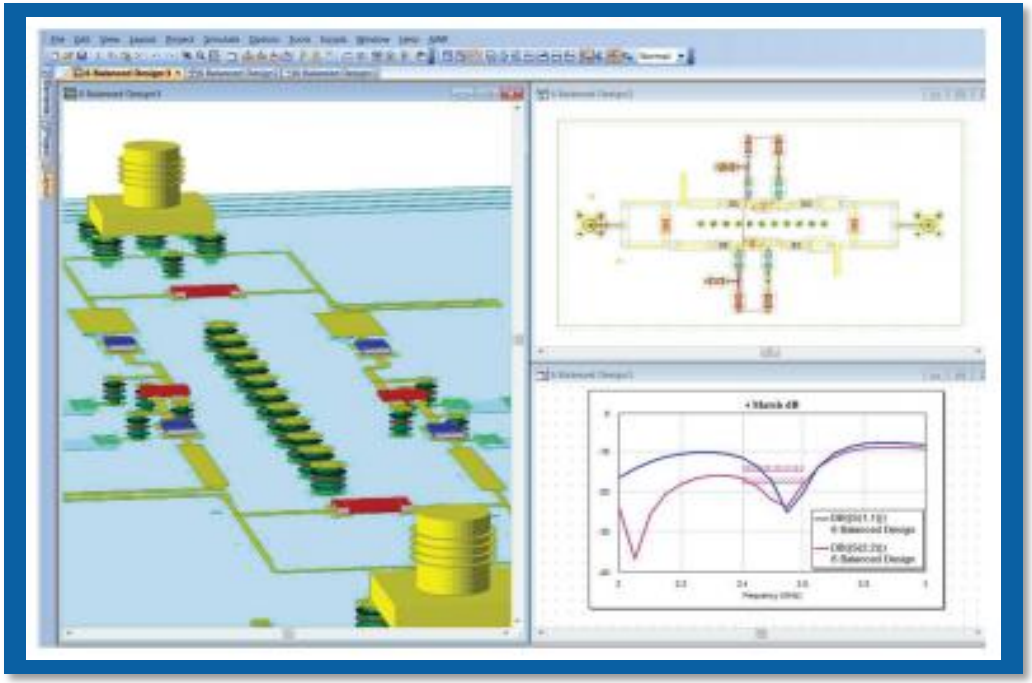
Global Presence (sales & support office locations)

- California, Wisconsin, Colorado
- United Kingdom, Finland, France and Germany
- Japan, Korea, Taiwan, China and Australia

Microwave Office

RF/Microwave Circuit Design Software

- MMIC
- RF PCB
- Modules



www.aavamobile.com

Aava Mobile Uses Microwave Office In The Design Of World's First Open Mobile Device Platform

"Because we are a young start-up, design time and cycles are critical and it is important for us to succeed on the first round. The ease-of-use of the software, simulation speed, and accuracy of models in Microwave Office gave us confidence for the first build.**"**

Sami Kolanen, RF Specialist
Aava Mobile

NATIONAL INSTRUMENTS

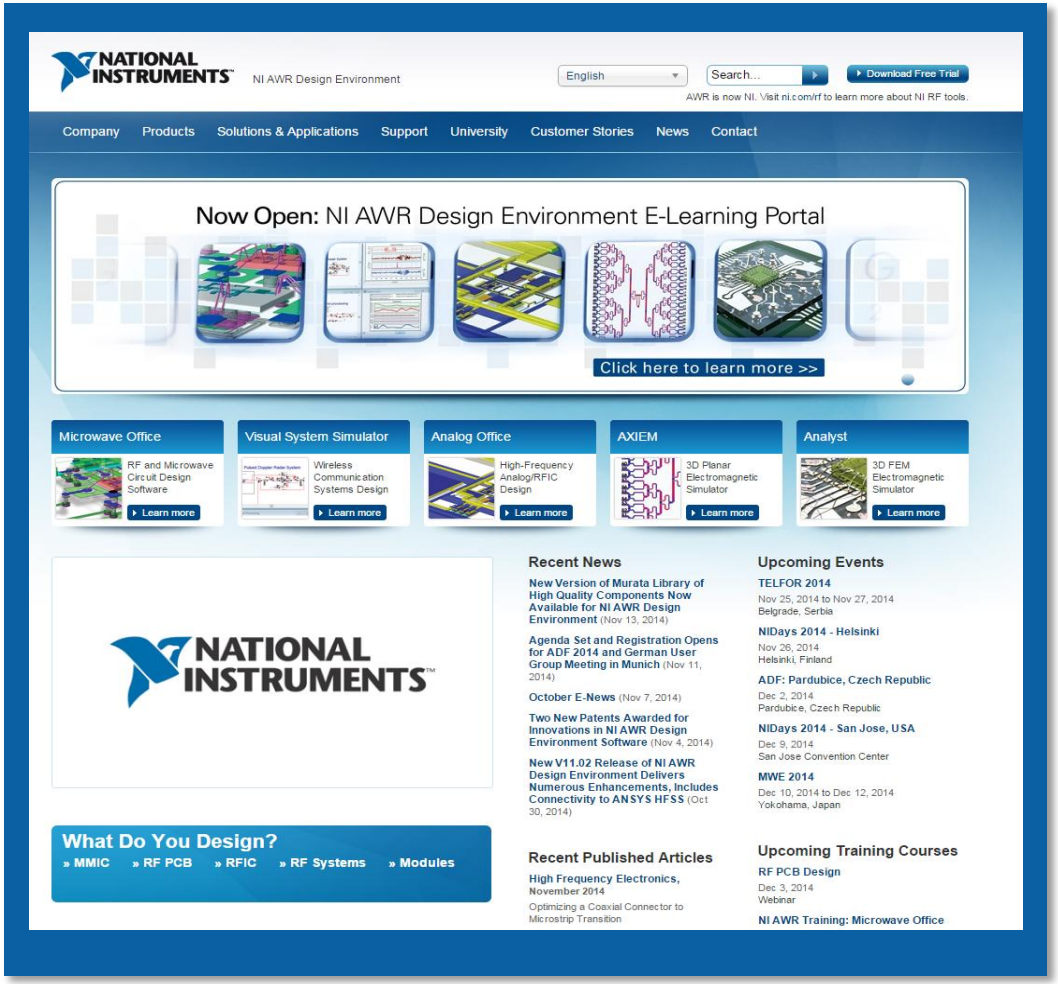
Learn More…

Online

- ni.com/awr
- awr.tv

Email

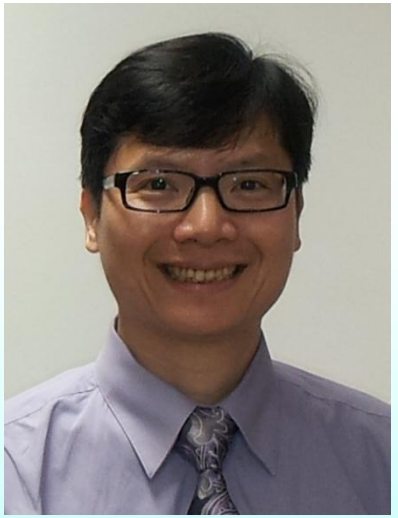
- info@awrcorp.com



RF PCB Design

Henry Lau

Lexiwave Technology, Inc.



Aims

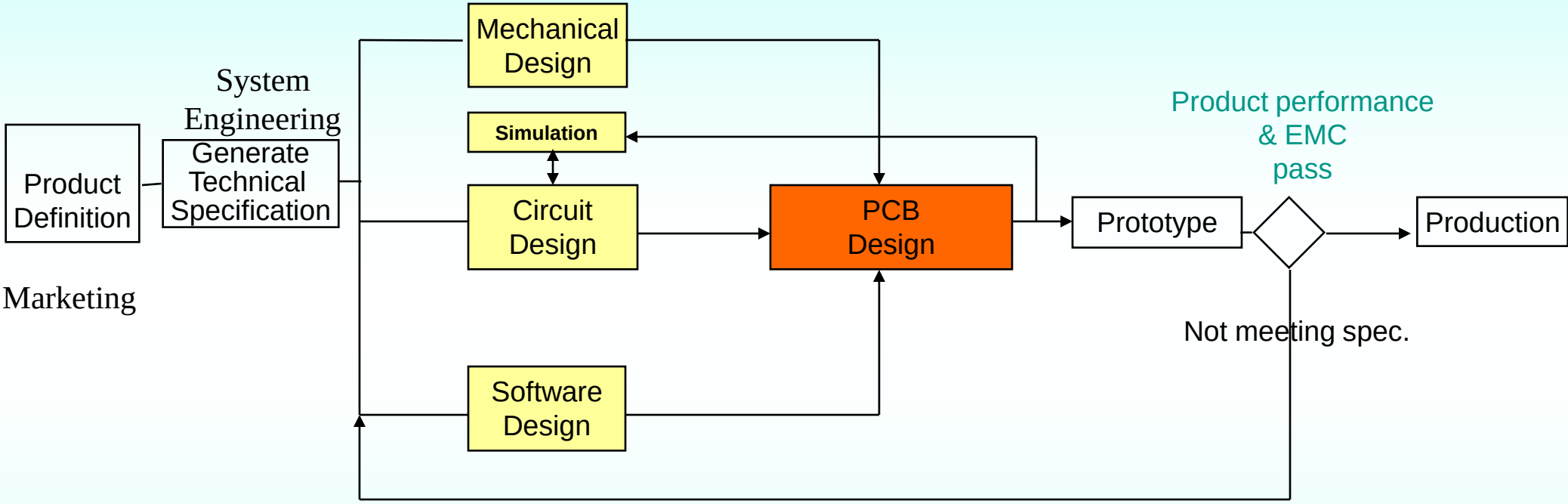
- To acquire technical insights and design techniques on RF printed circuit board design for Wireless Networks, Products and Telecommunication
  * PCB of RF circuits
  * PCB of digital, analog and audio circuits
  * Design issues for EMI/EMC
  * Design for mass production

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

Design Framework



Long cycle time

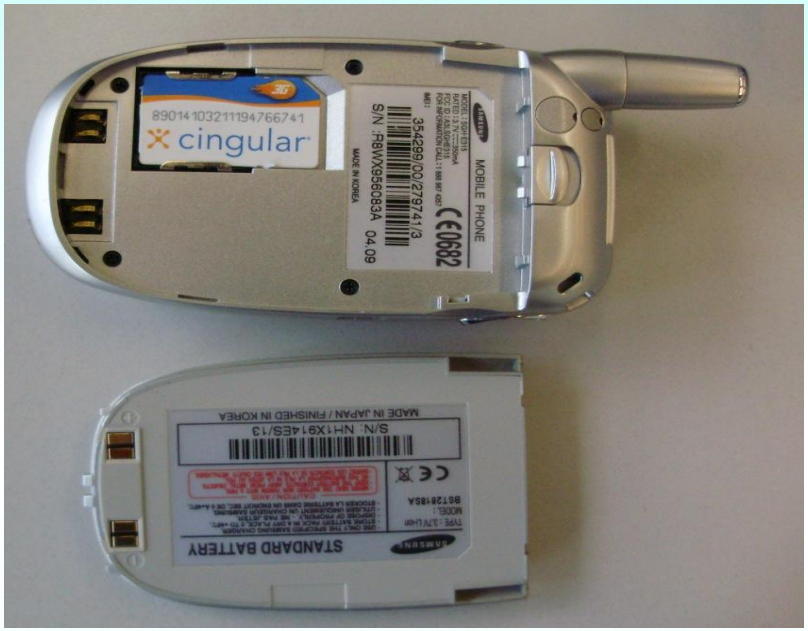
Cooperation Between Mechanical & Electronic Design

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

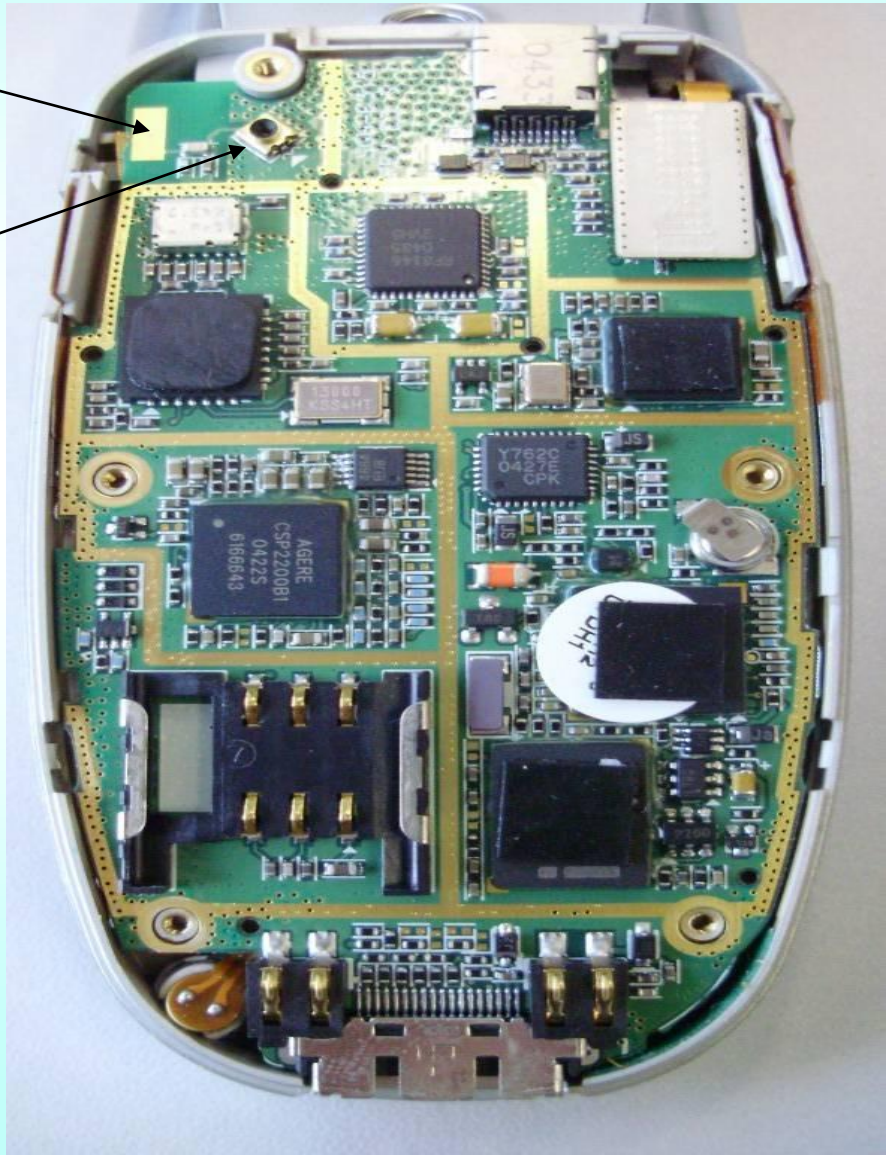
Cooperation Between Mechanical & Electronic Design

- 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



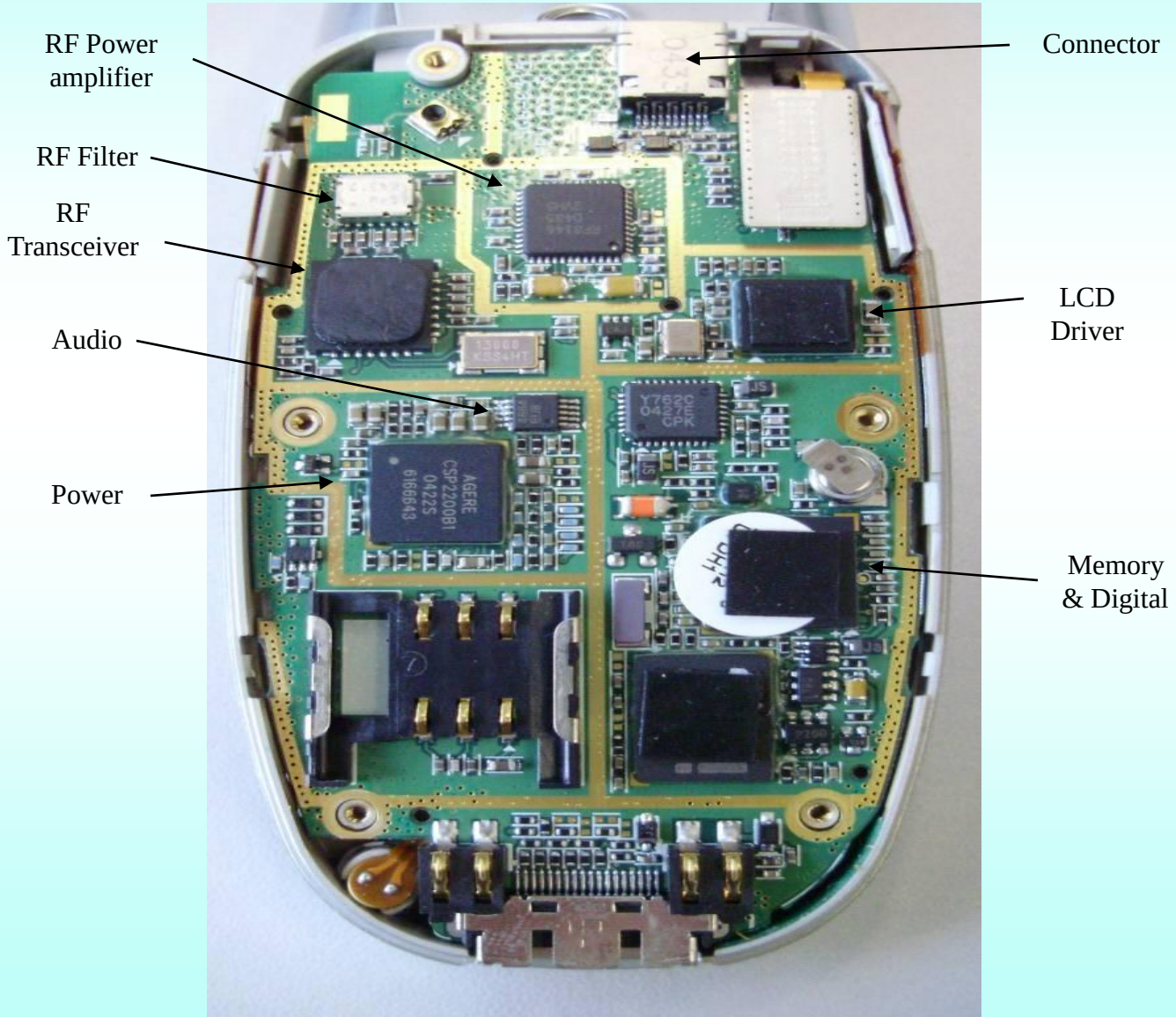
Cooperation Between Mechanical & Electronic Design

- 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



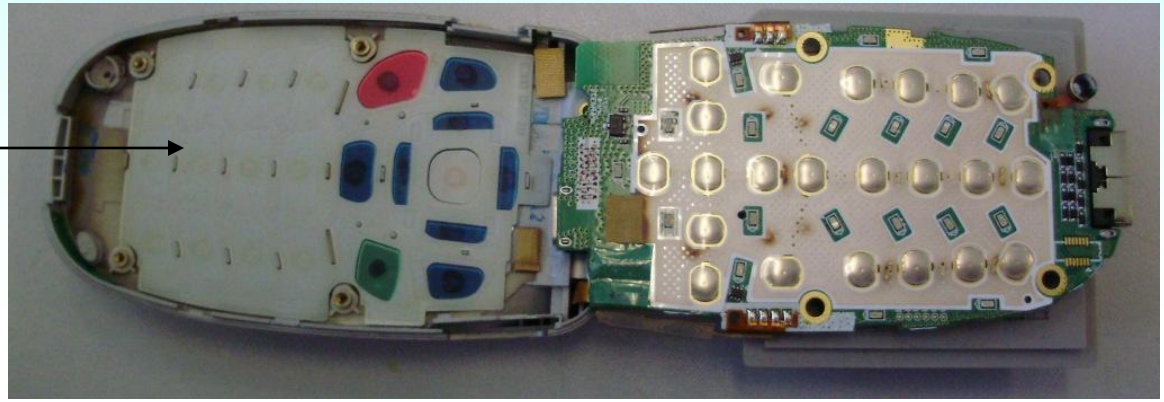
Cooperation Between Mechanical & Electronic Design

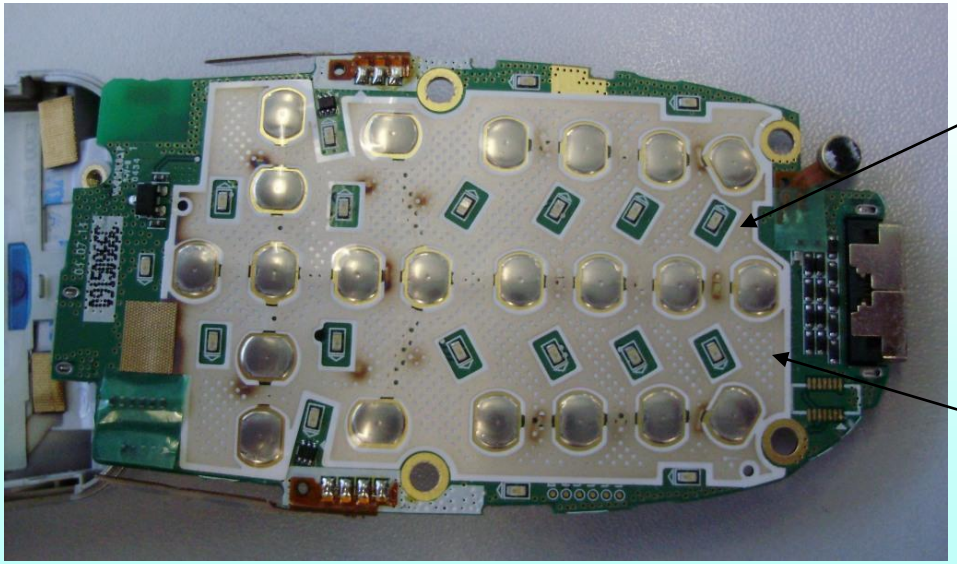
- Circuit grouping and partitioning
- Audio, video, digital, RF, analog
- Board mounting and assembly



Cooperation Between Mechanical & Electronic Design

Key Pad

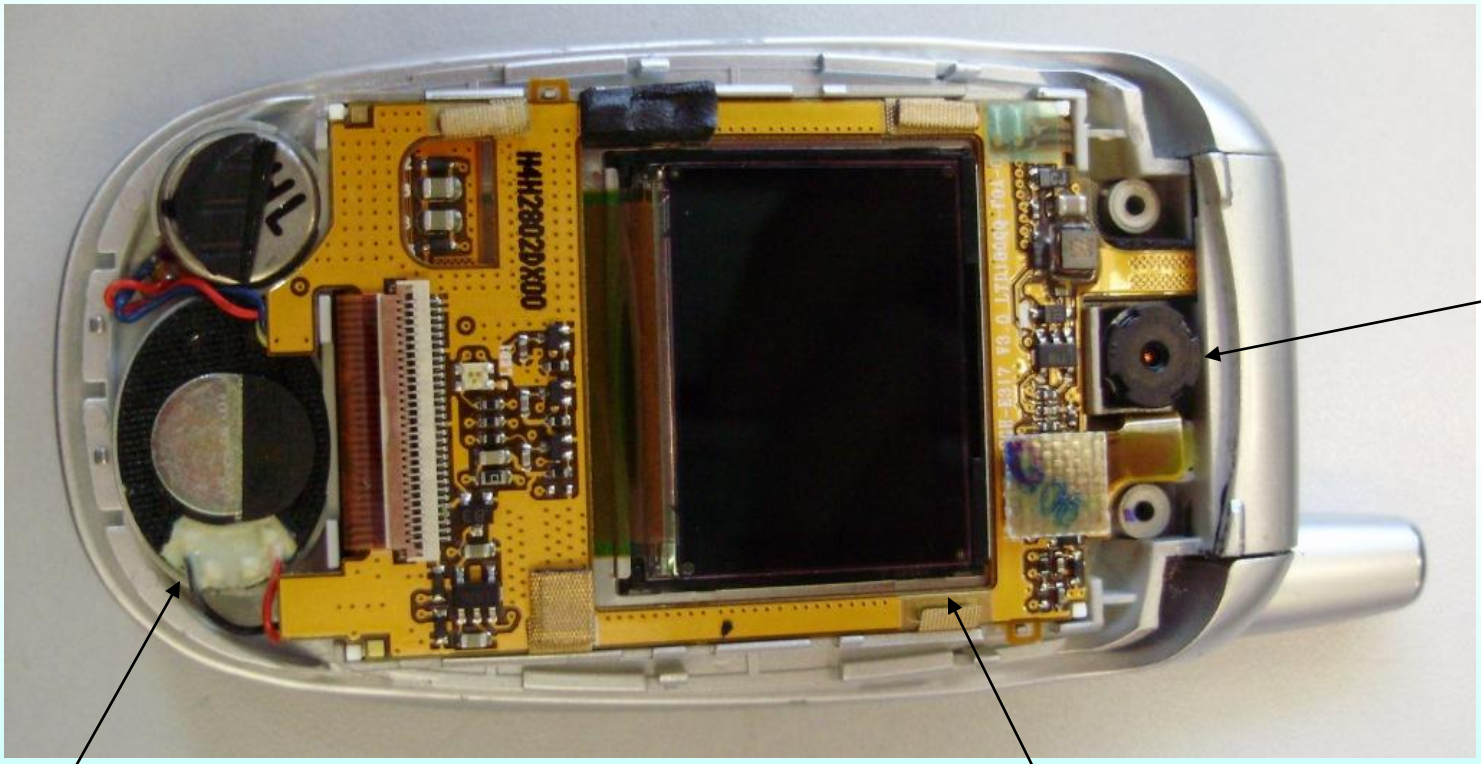




Membrane

Very few components on bottom layer

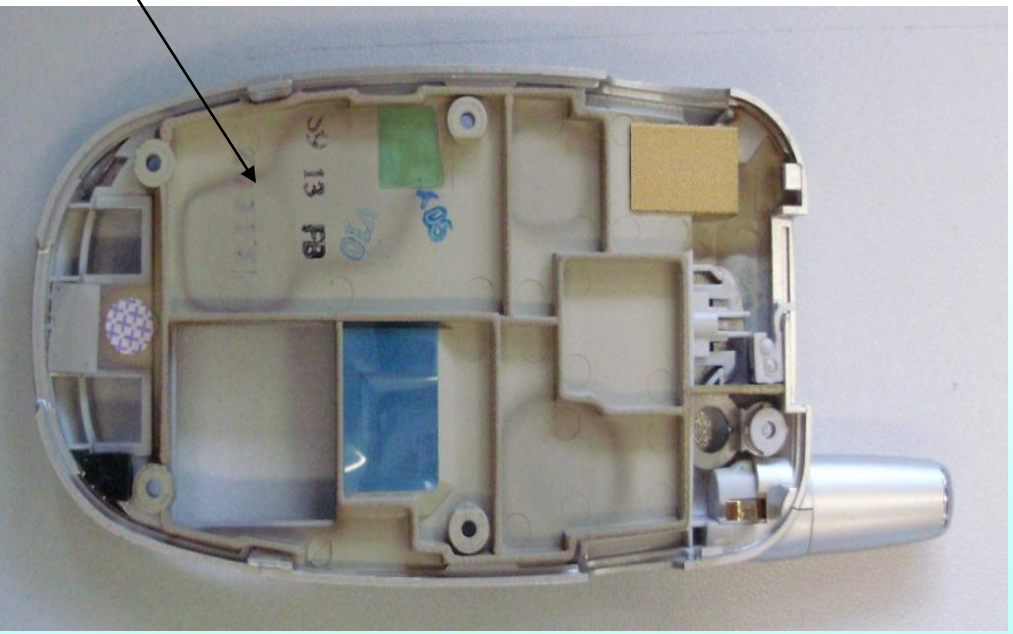
Cooperation Between Mechanical & Electronic Design



Cooperation Between Mechanical & Electronic Design

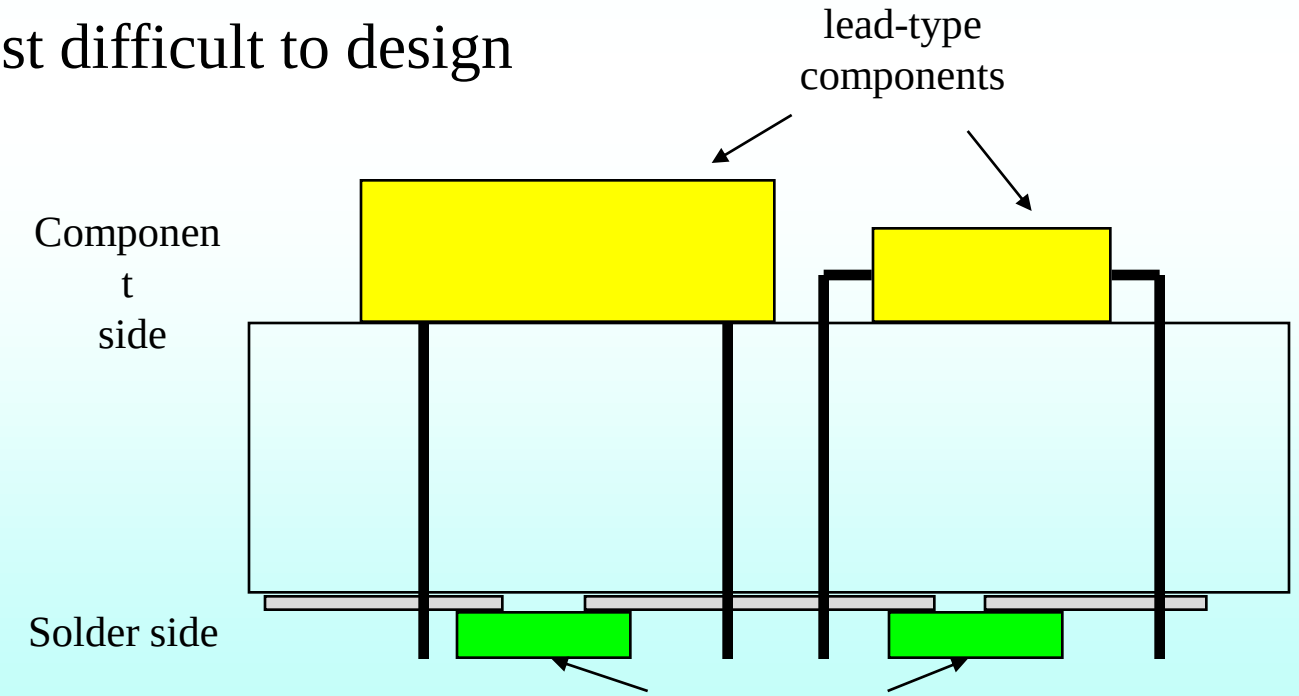
- Shielding and isolation
  - Method, material
- EMI/EMC/ESD issues

Metallization on plastic



Layer Stackup Assignment

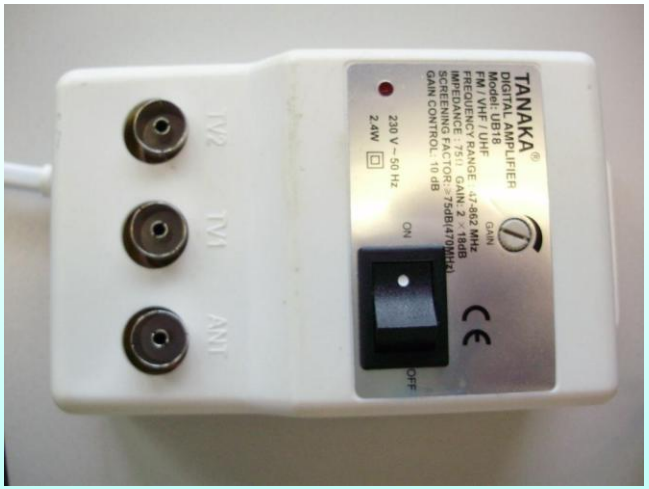
- Single - layer
  - Typical thickness : 1.6mm, 1.2mm, 1mm, 0.8mm
  - Cheapest
  - Prototype turn-around time - 2 days
  - Component mounting occupies most area
  - Most difficult to design



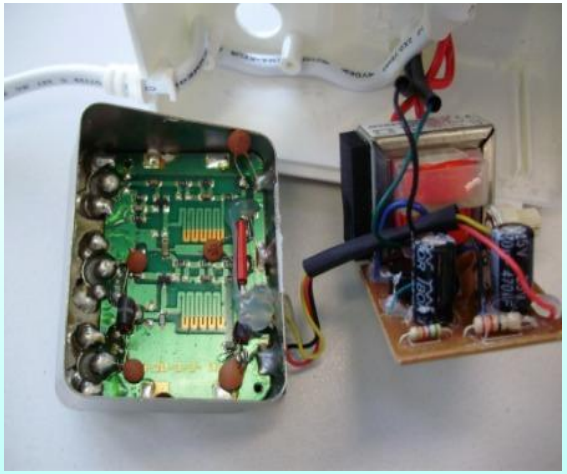
Layer Stackup Assignment

- 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

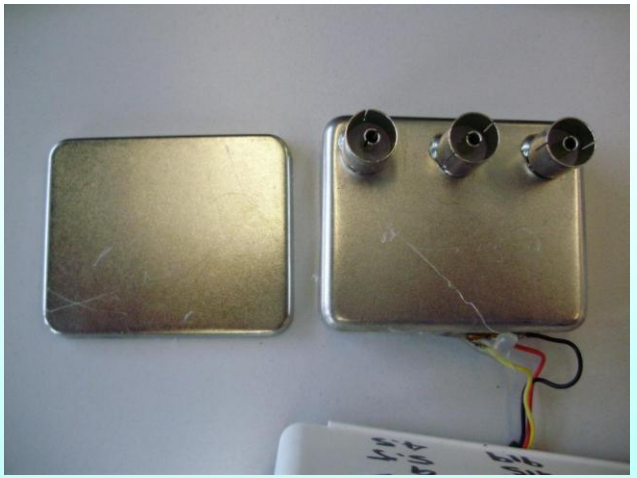
TV signal booster



RF amplifier + Power Supply



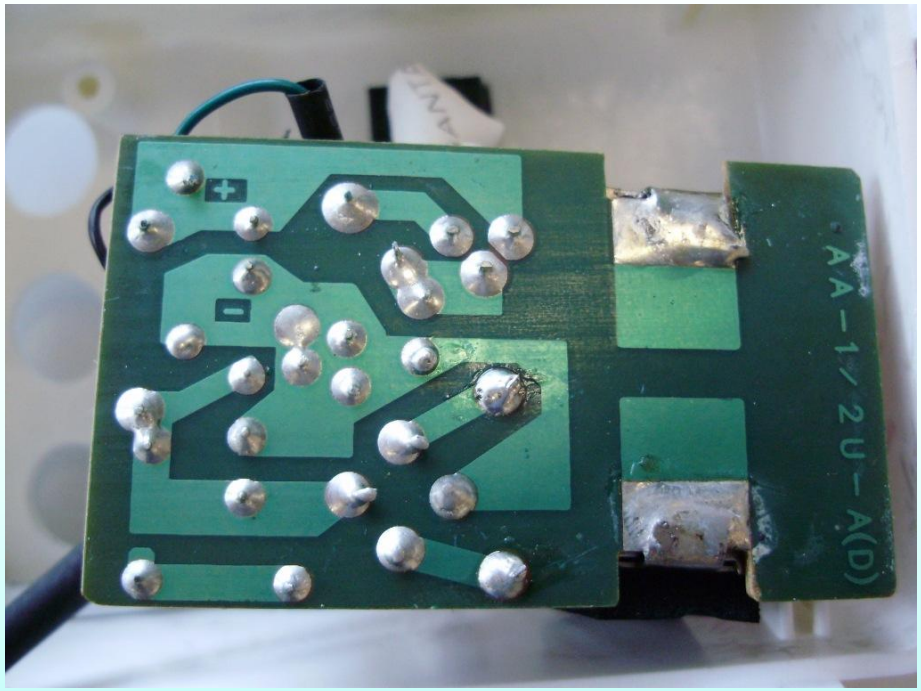
RF amplifier in a shield box



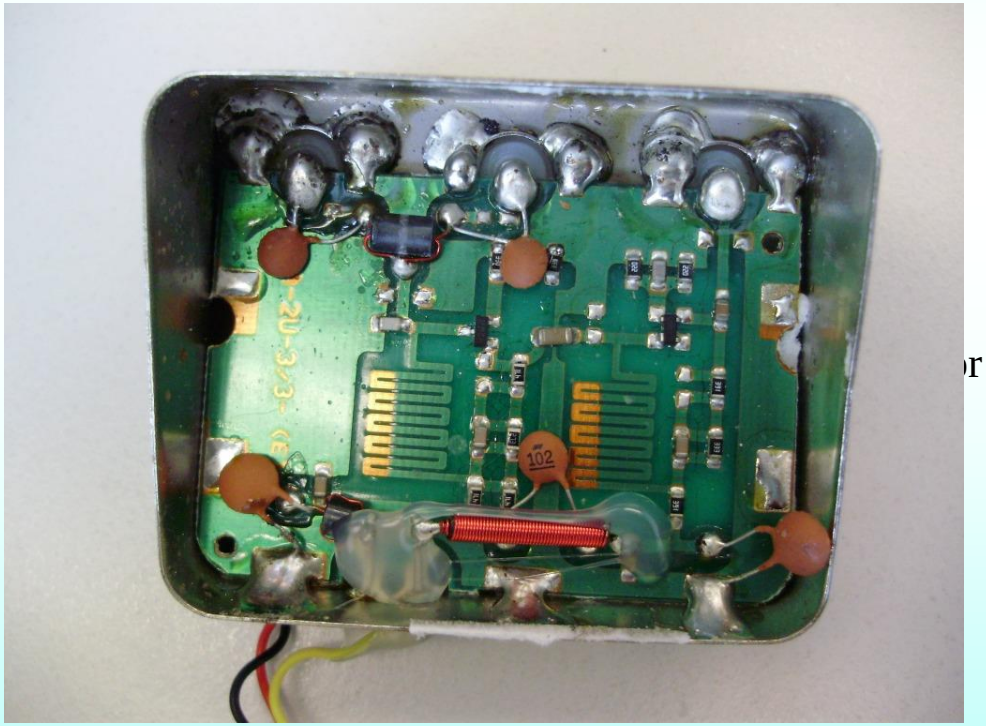
Layer Stackup Assignment

- Single - side PCB

Safety issue
on AC board

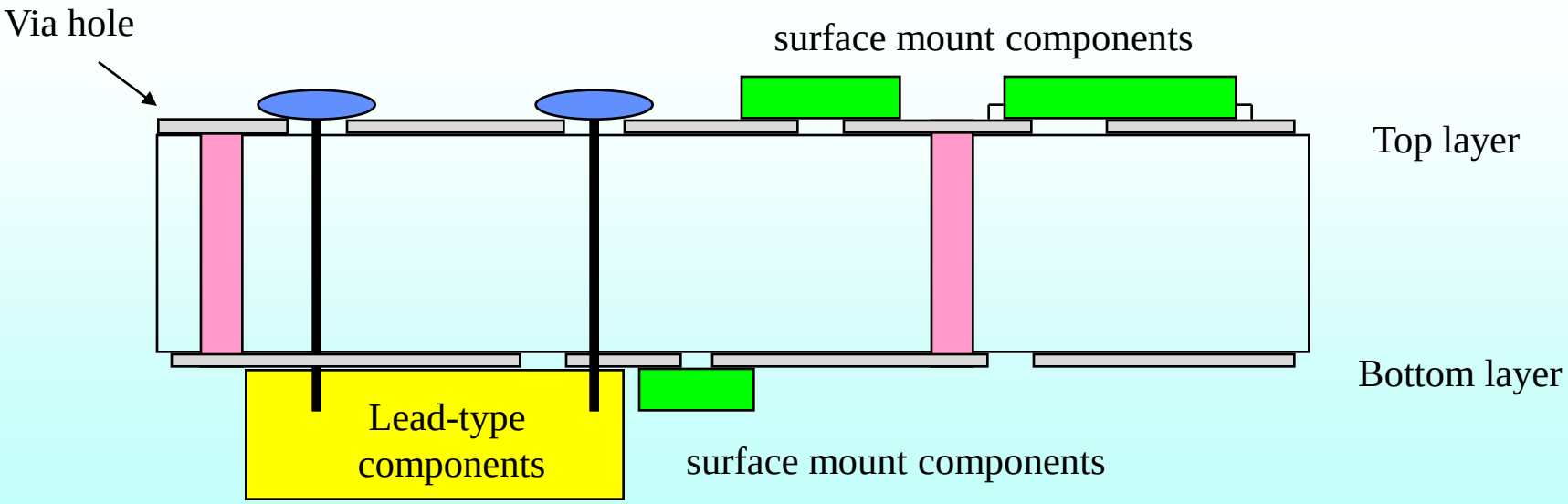


SMT + Lead
type components



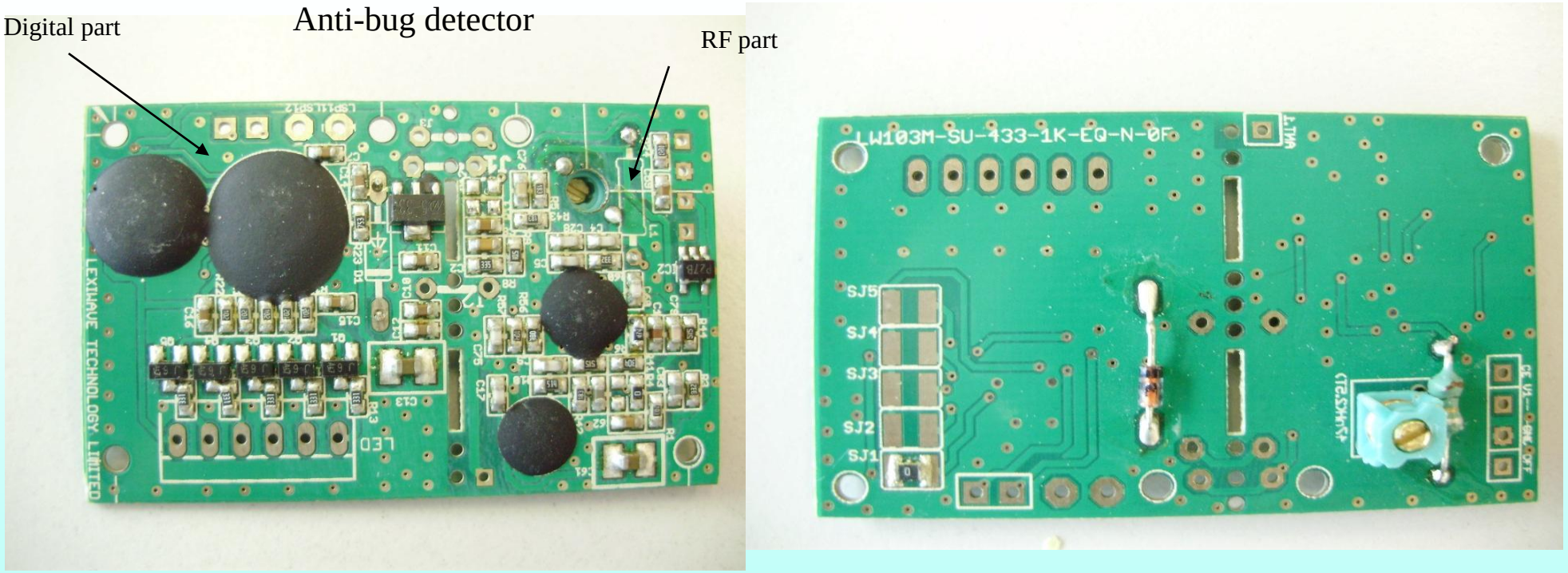
Layer Stackup Assignment

- Double - side
  - Price competitive
  - Prototype turn-around time - 4 days
  - Top layer : component mounting and major signal tracings
  - Bottom layer : primarily with ground plane
  - power trace
  - Put SMD / LT mixed component design on one side to save production cost



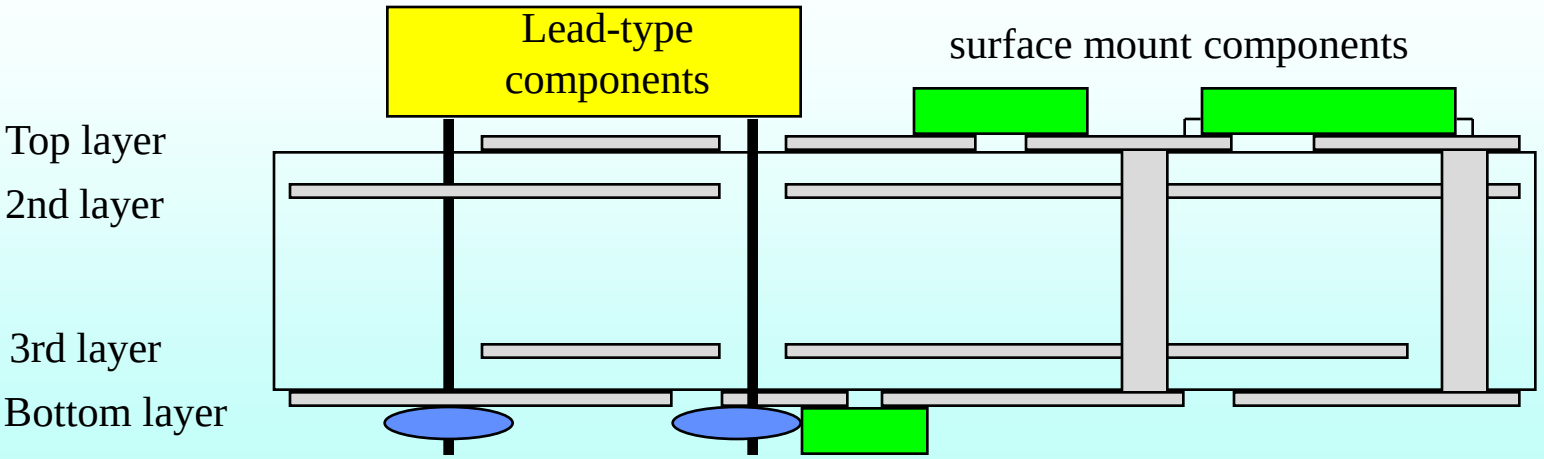
Layer Stackup Assignment

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



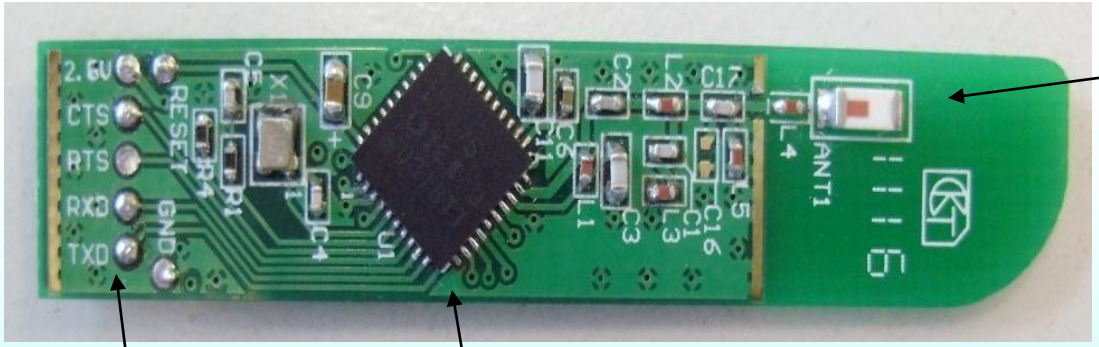
Performance comparison

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

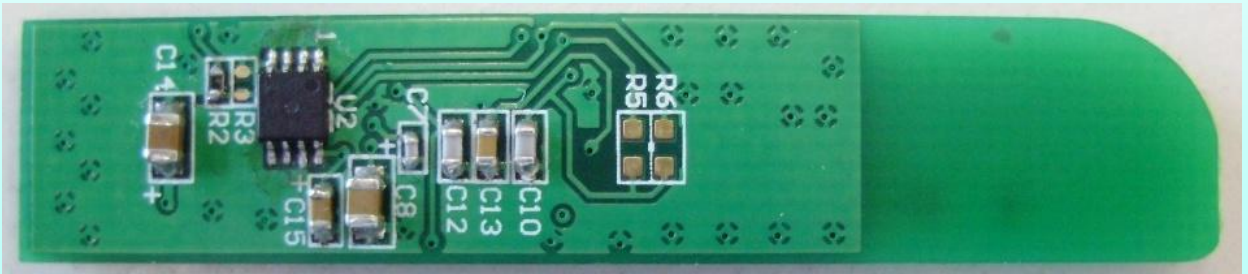


Chip Antenna

2.4GHz Zigbee Wireless Module

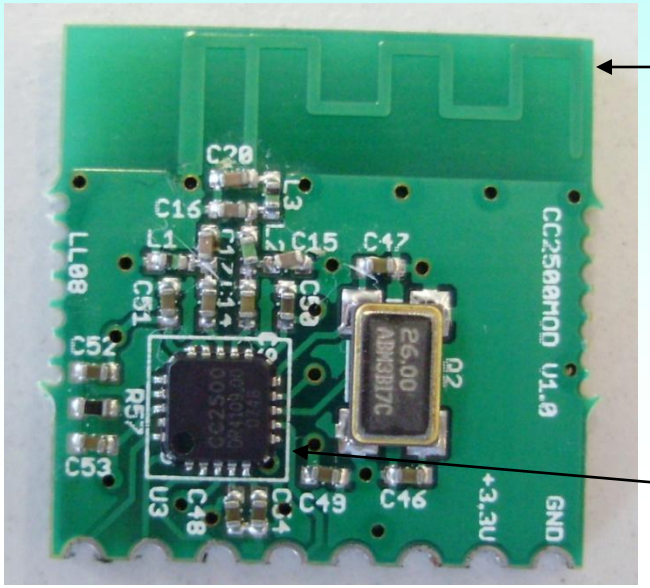
Host MCU interface

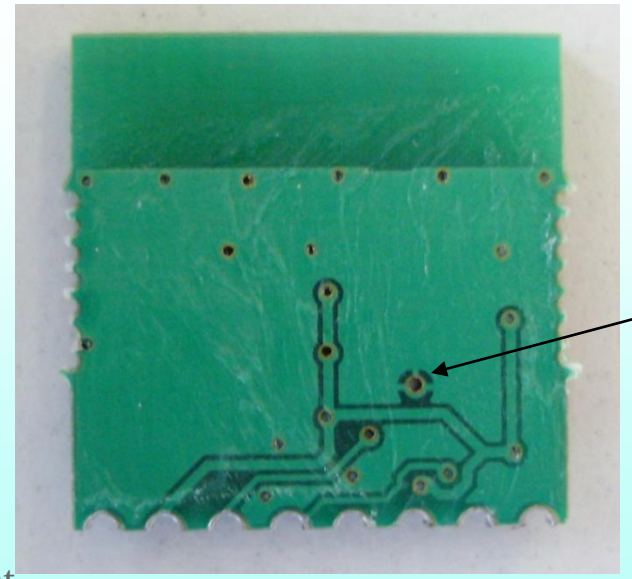
Transceiver



Component Placement

- Identify and segment groups of circuits
  - antenna, analog, digital, switching, audio. . . . . .
- Identify critical components
- Maximize grounding area
- Optimize power traces
- 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

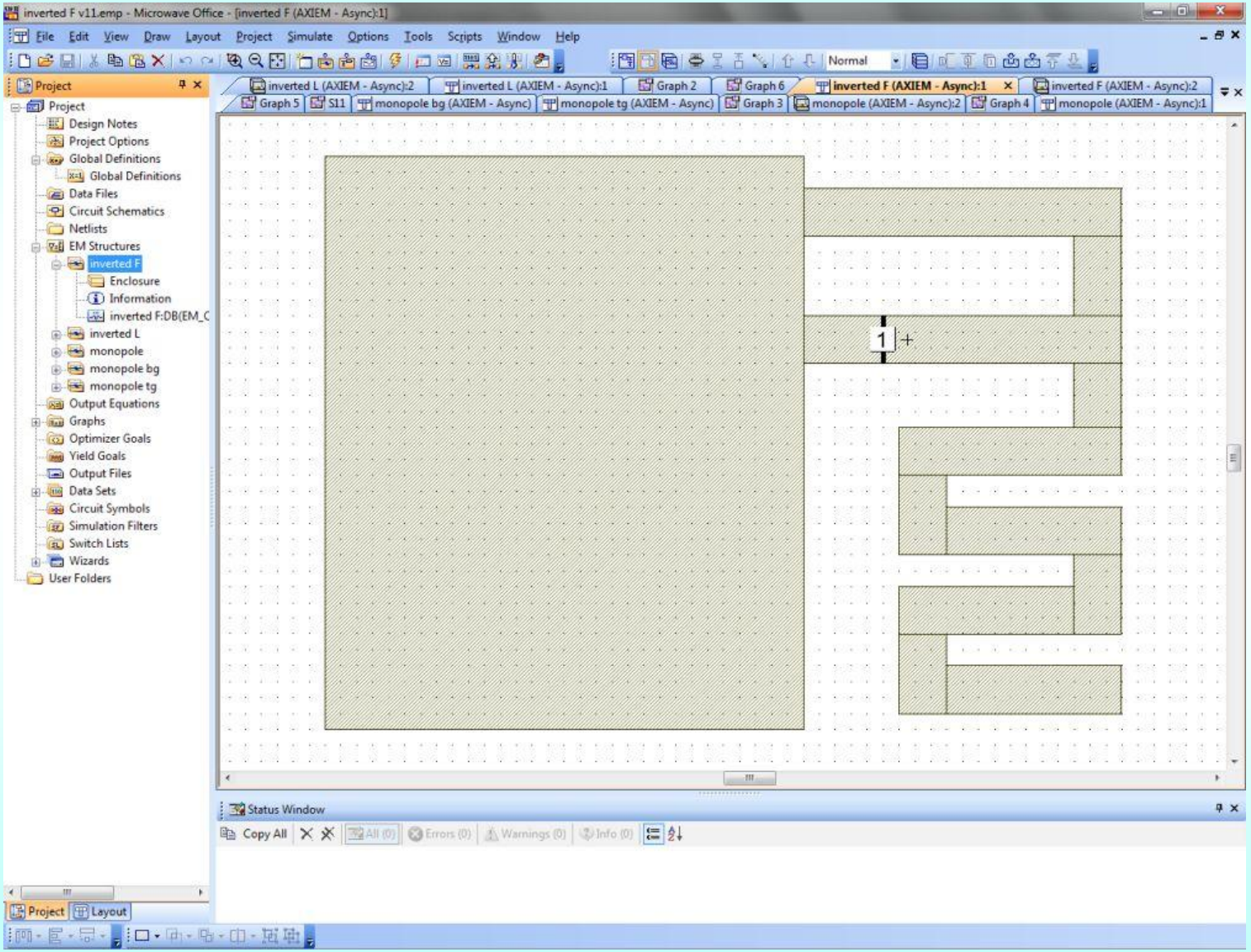
- Place components as close to Integrated Circuits as possible with the priority of RF, IF and audio components
- Put the components with more interconnections close to each other
- Proper bus / ports assignment to shorten trace length and avoid cross-over

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

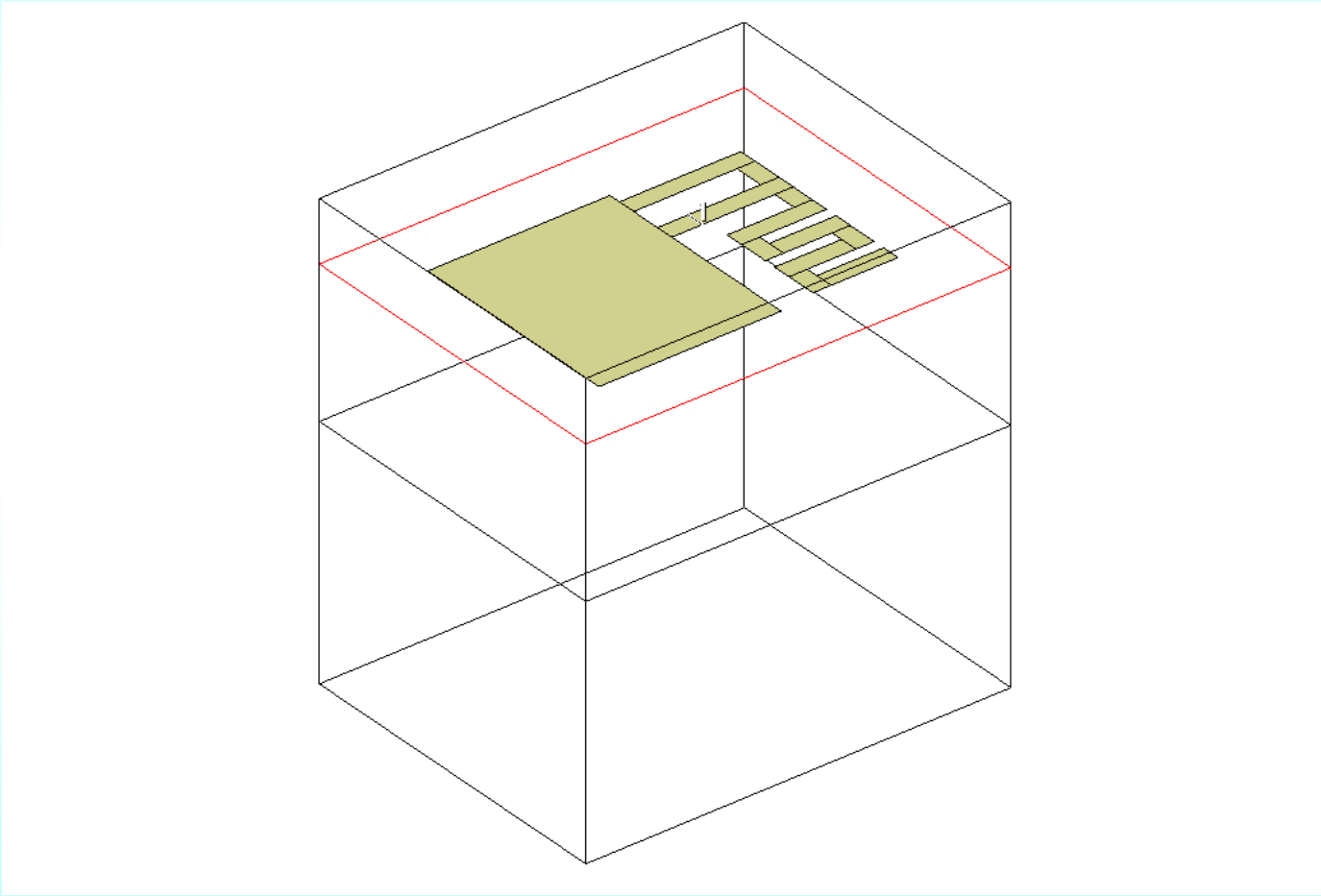
PCB Antenna Design

- AWR
  - EM simulator – Axiem
    - Inverted-F PCB Antenna



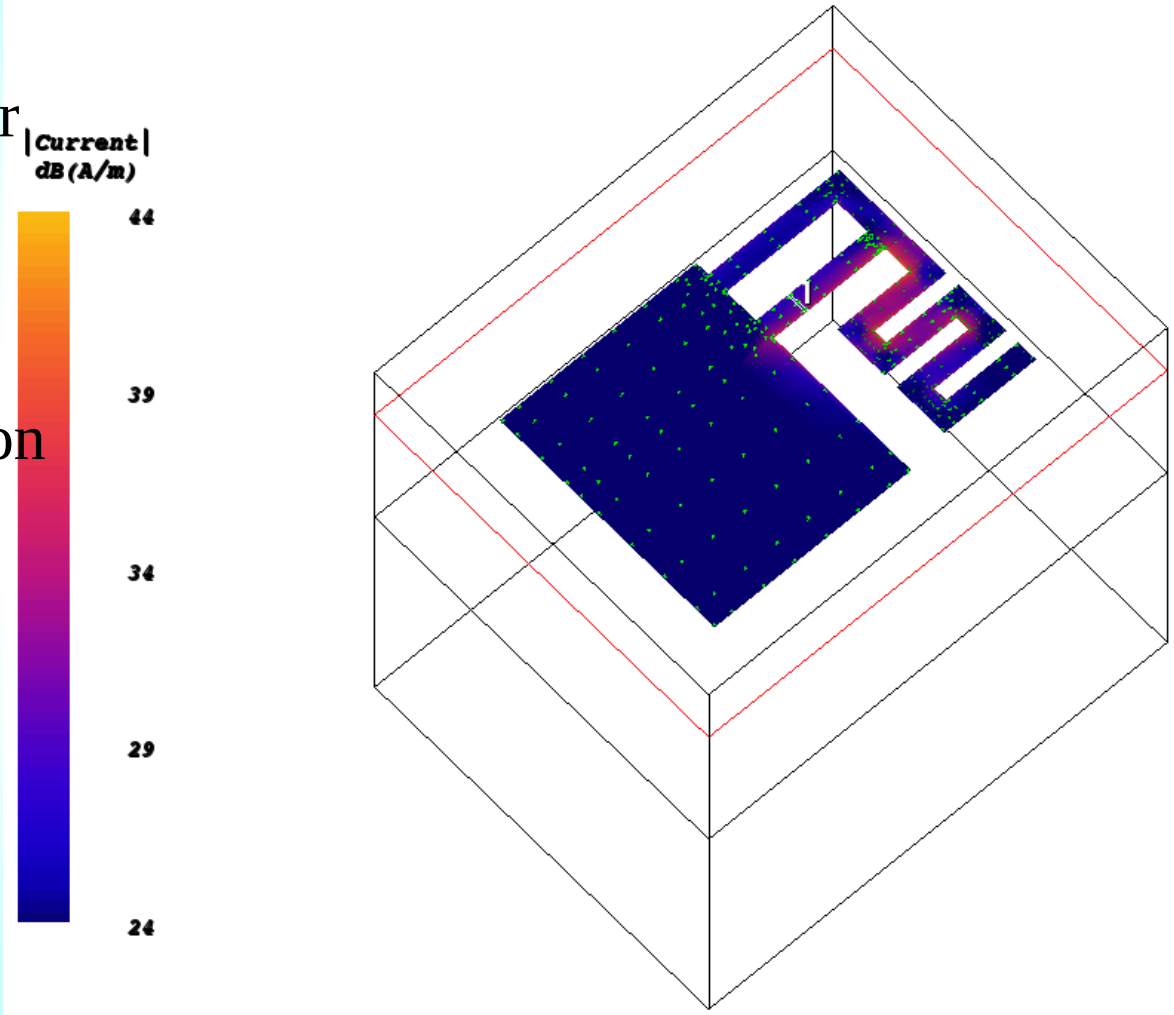
PCB Antenna

- AWR
  - EM simulator – Axiem
  - 3-D Layout View
    - With enclosure



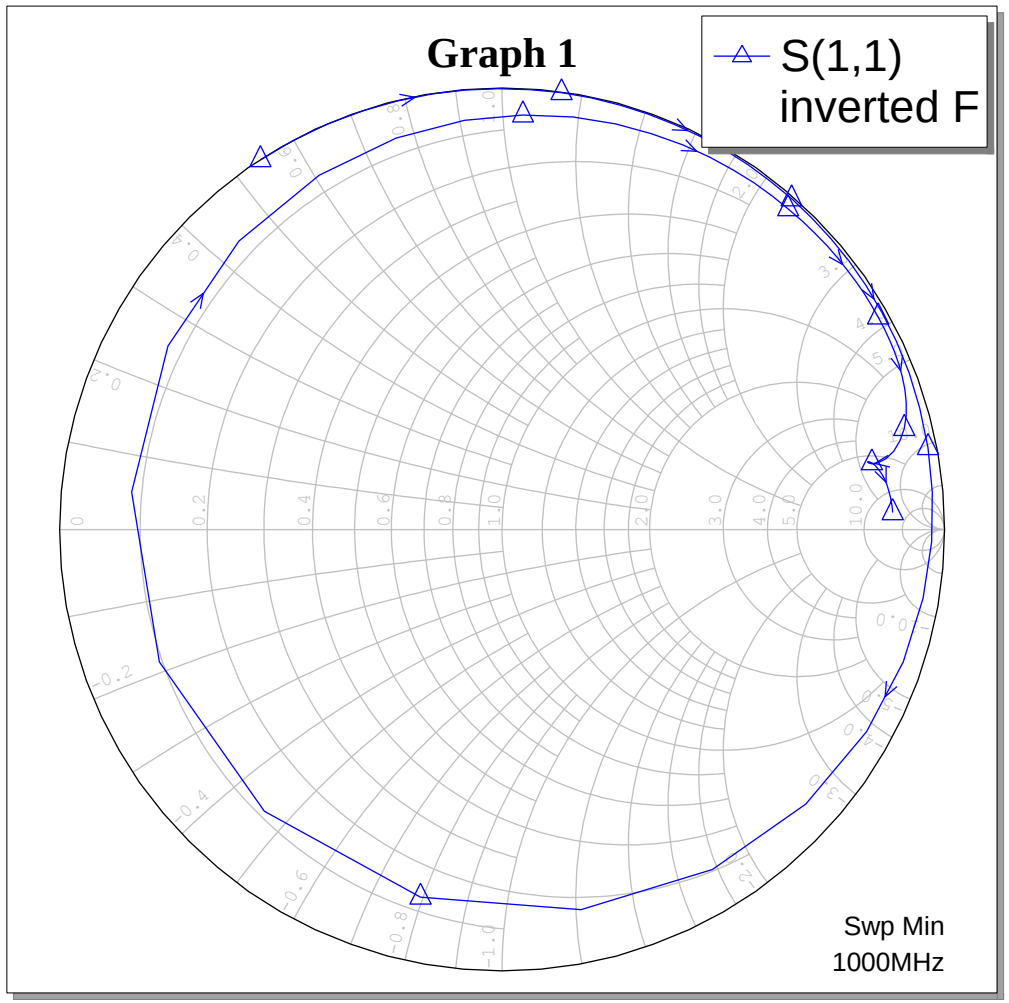
PCB Antenna

- AWR
  - EM simulator – Axiem
    - Current field distribution



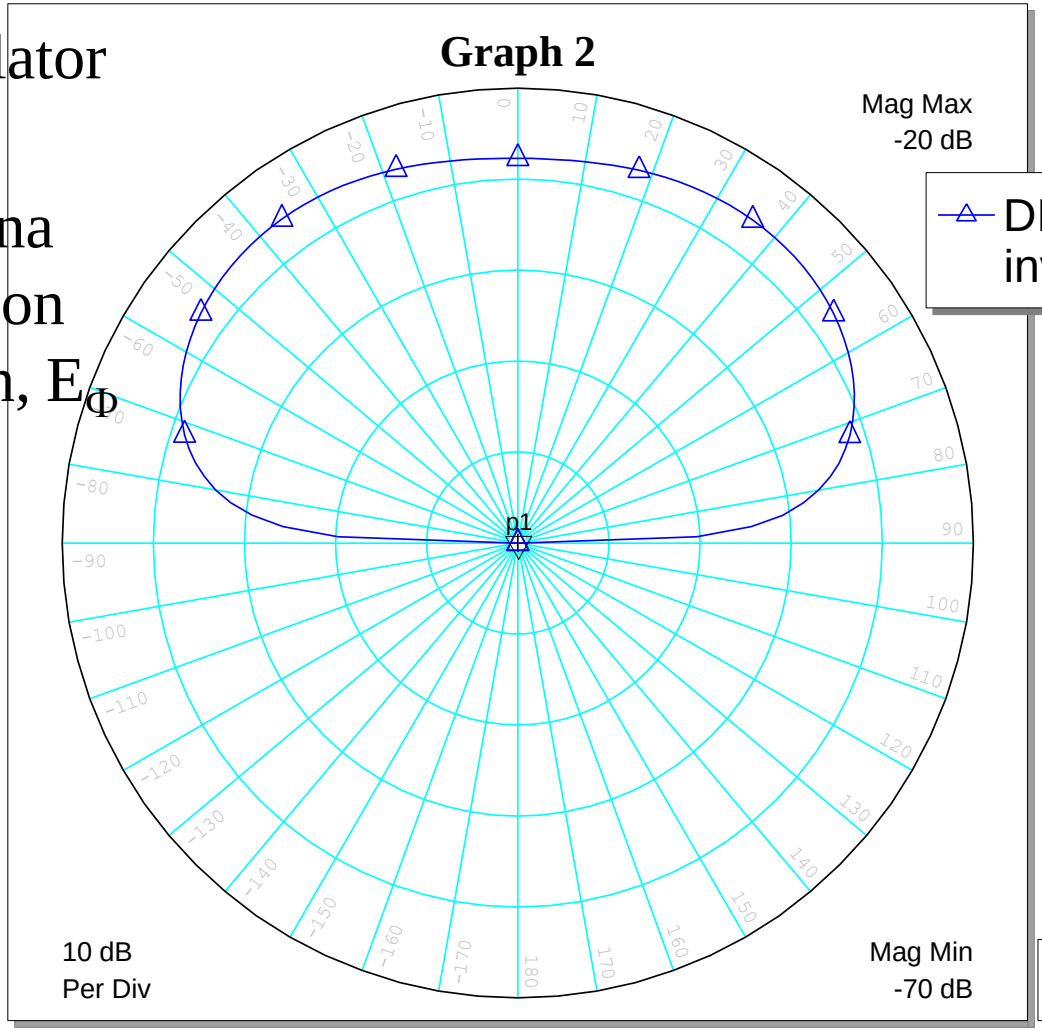
PCB Antenna

- AWR
  - EM simulator – Axiem
    - Simulated input impedance



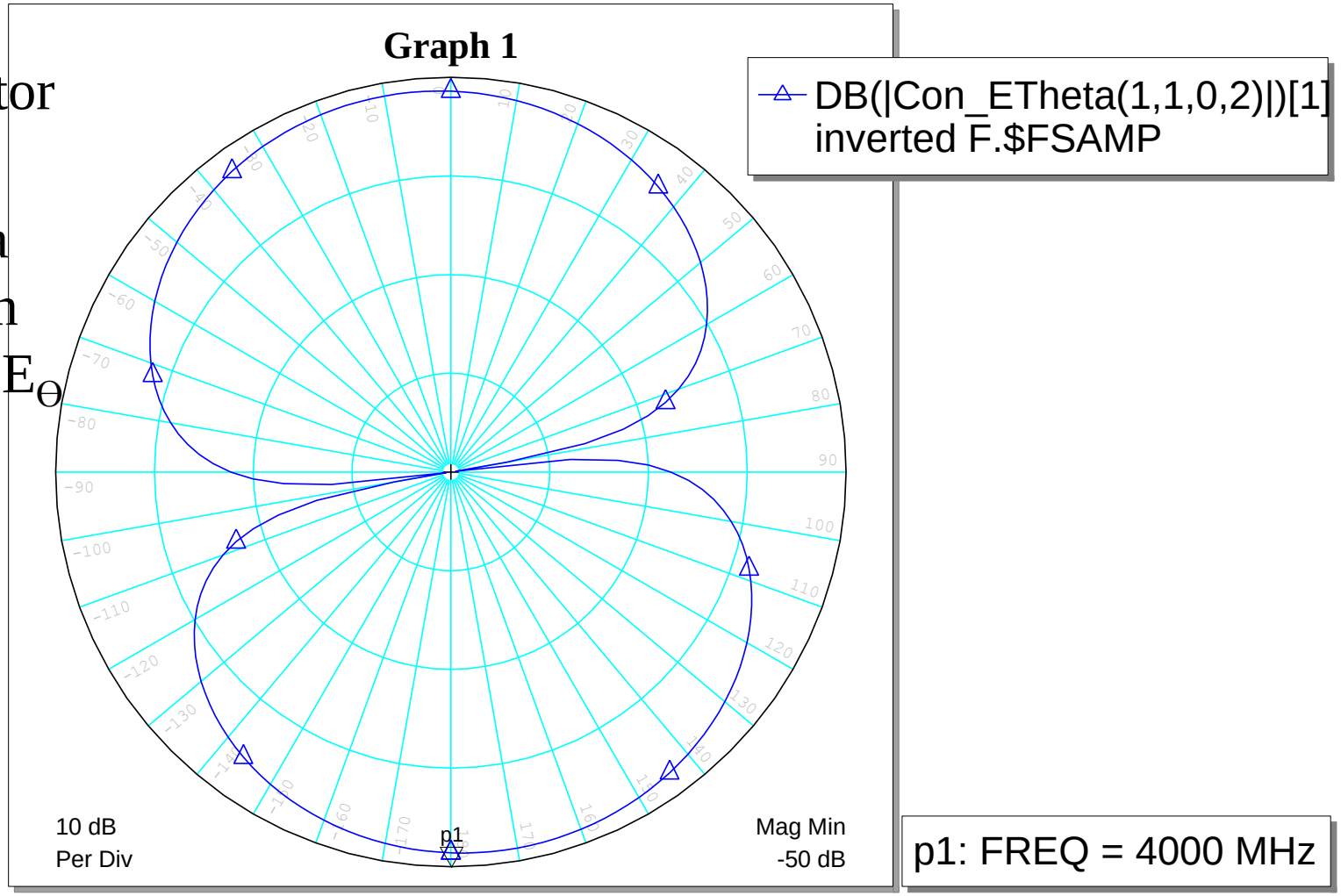
PCB Antenna

- AWR
  - EM simulator – Axiem
    - Antenna radiation pattern, E_Φ



PCB Antenna

- AWR
  - EM simulator – Axiem
    - Antenna radiation pattern, E_Θ



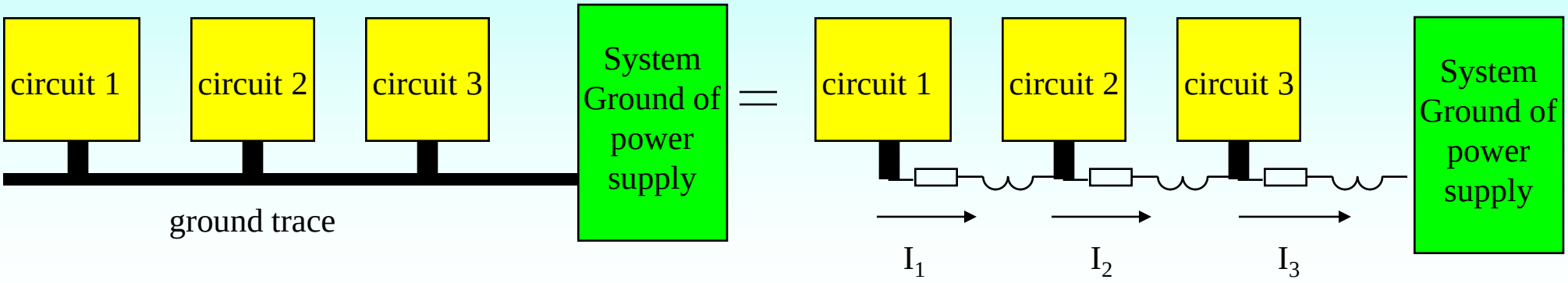
Grounding

- 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

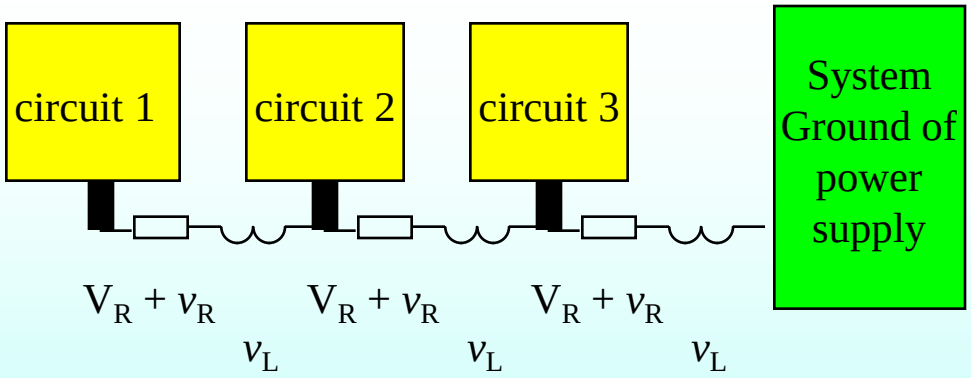
Grounding

- 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
  - Try to be a complete plane
    - avoid interruption from via, signal traces
  - avoid excessive copper pour and unused copper

Grounding Method



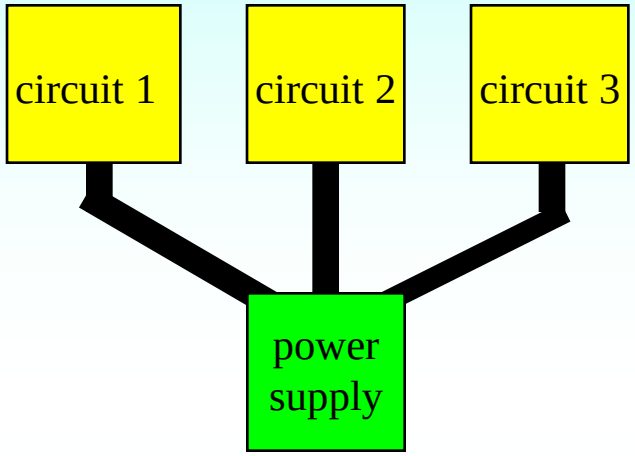
Equivalent circuit of ground trace (series connection)

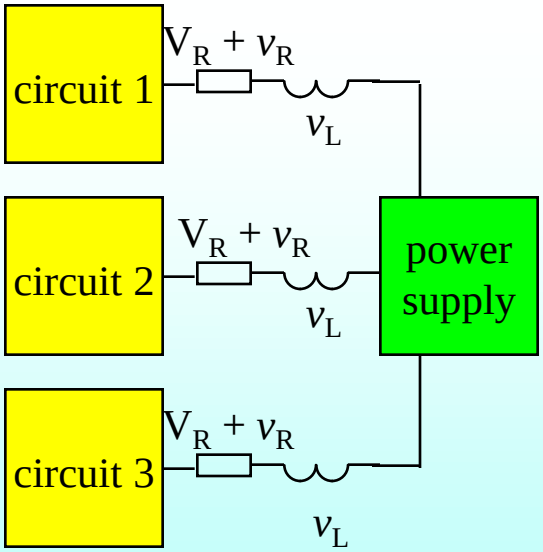


Noise and signal voltage induced by
ground current and imperfect ground connection,
additive noise and signal voltage affects all circuit blocks

Grounding Method

Star Connection

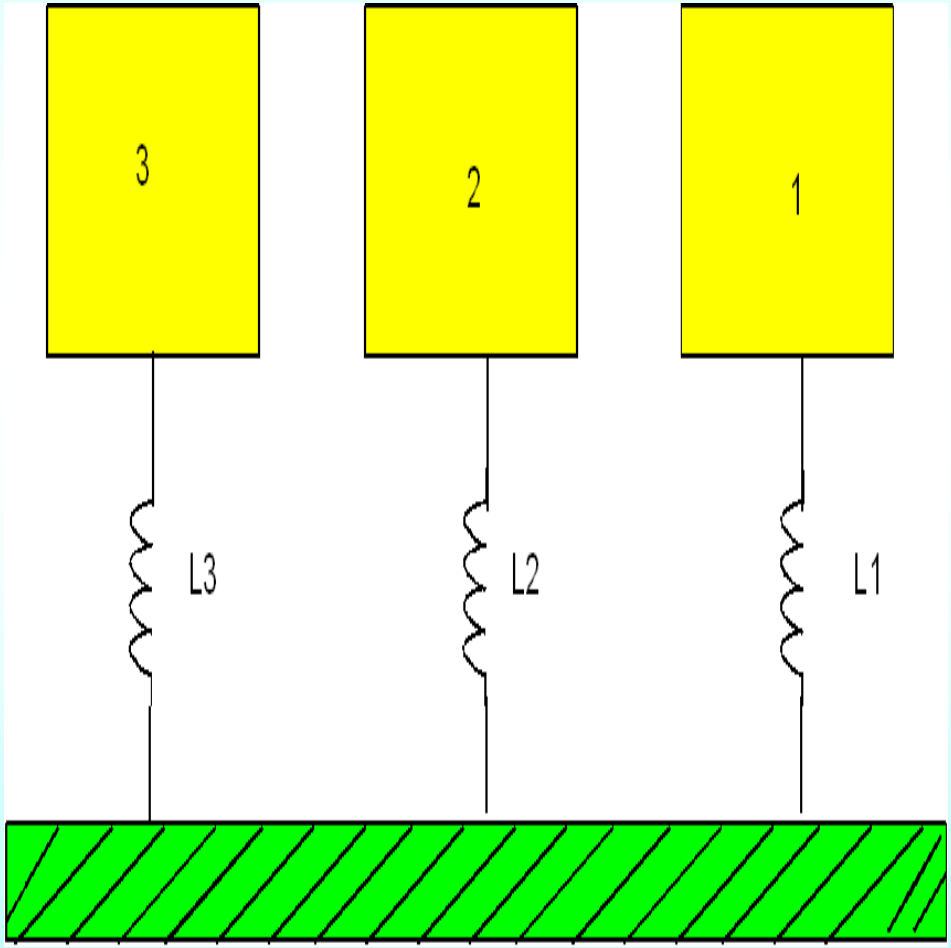




Minimize ground inductance and resistance,
Reduce induced ground noise voltage,
Minimize additive ground noise voltage

Grounding Method

Multipoint Grounding Connection

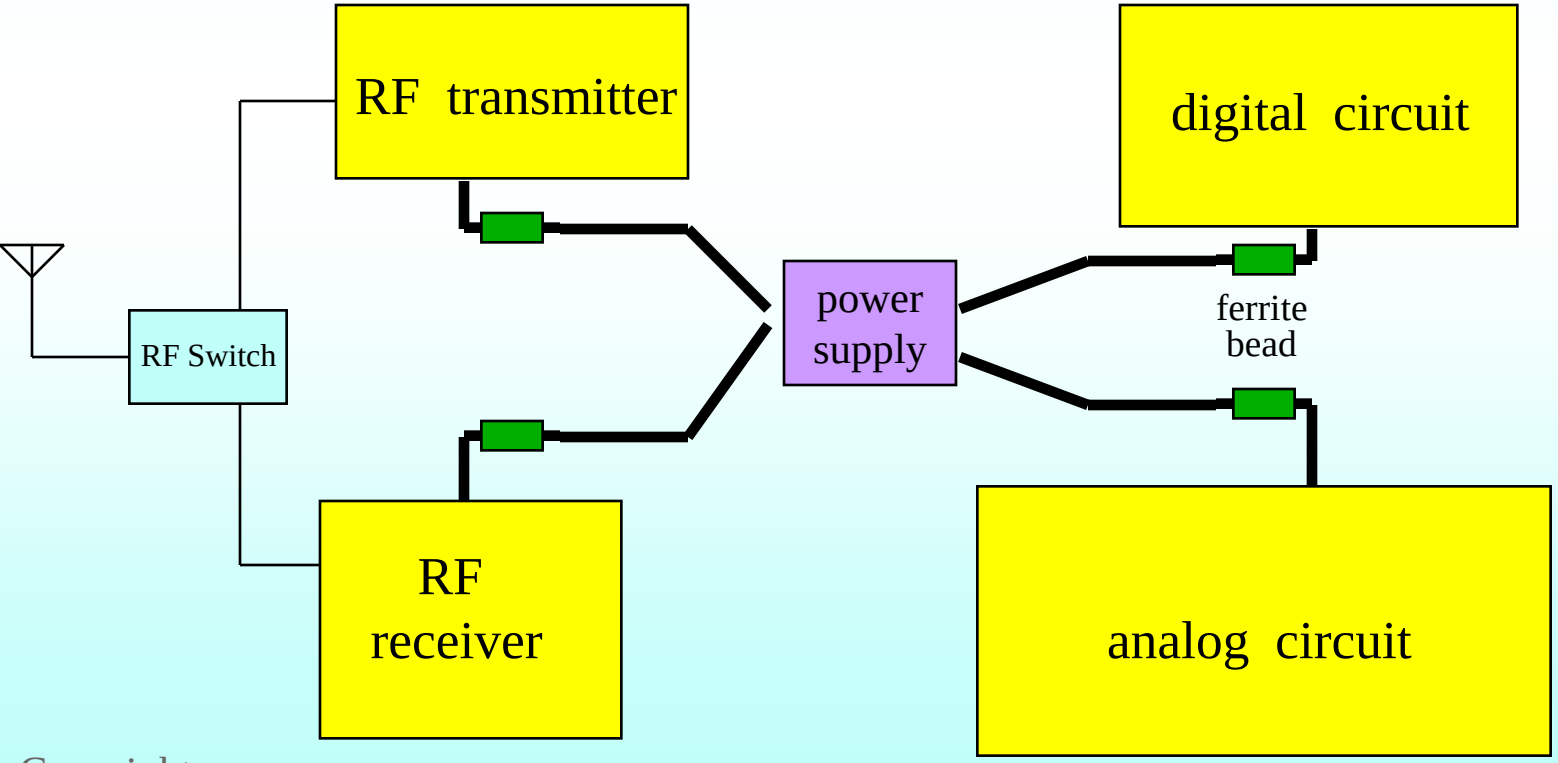


Power Routing and Power Plane

- 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 sub-circuit

Power Routing and Power Plane

- " Star " type connection , work with **GOOD** ground plane
- Ferrite bead presents high impedance at higher frequency, should place near the sub-circuit
- If space provided, printed inductors and printed capacitors can be used above 1 GHz



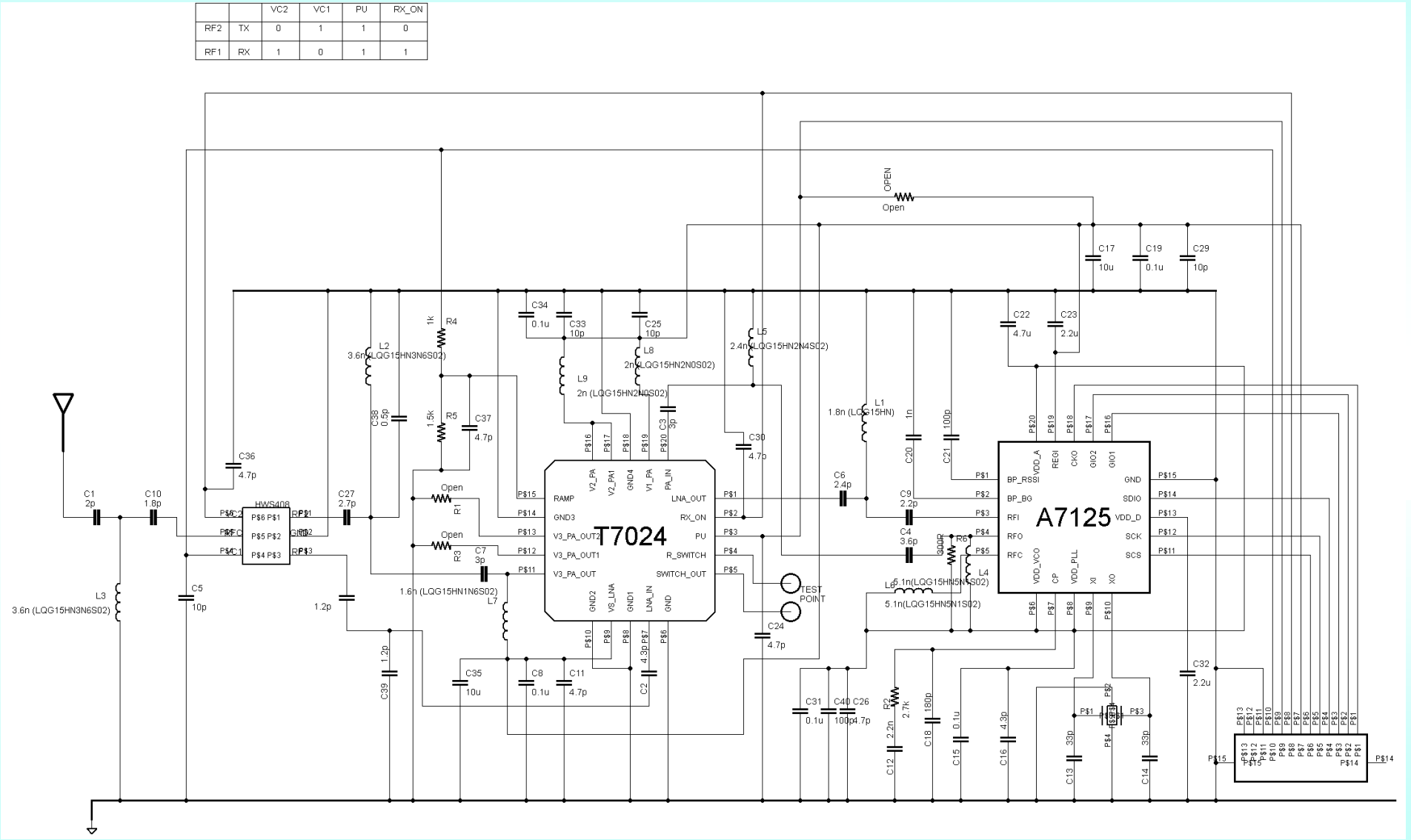
Bypassing & Decoupling

- 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

- Size & Quantity
  - as large and short as possible
    - Inductance and resistance $\alpha \pi \times d / h$
      - *Where d is diameter, h is height*
  - Number of via holes depends on frequency and current
- Location
  - avoid signal via cutting too much on the ground plane
  - Connect ground via immediately to the closest ground from the component
  - Not allowed inside SMD component pads
- multiple via holes for critical signal trace and ground

Routing

- Good component placement automatically can minimize parasitic inductance, capacitance and resistance
  - Parasitic
    * α trace length
    * 1/ α to trace width
    * Avoid sharp corner on high frequency or ESD sensitive traces
- Minimum parasitic allows
  * higher circuit Q with higher performance, ie VCO
  * More controllable
  * wider tuning range, ie. VCO, filter
  * more stable, ie LNA, Mixer

Tips of Routing

- Minimize stitches between layers
- Avoid sharp corner
- 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/EMC compliance
- Identify and understand sources of interference
- Circuit partitioning :

  Receiver : LNA, mixer PLL and IF amplifier

  Transmitter : PLL, oscillator, buffer and power amplifier

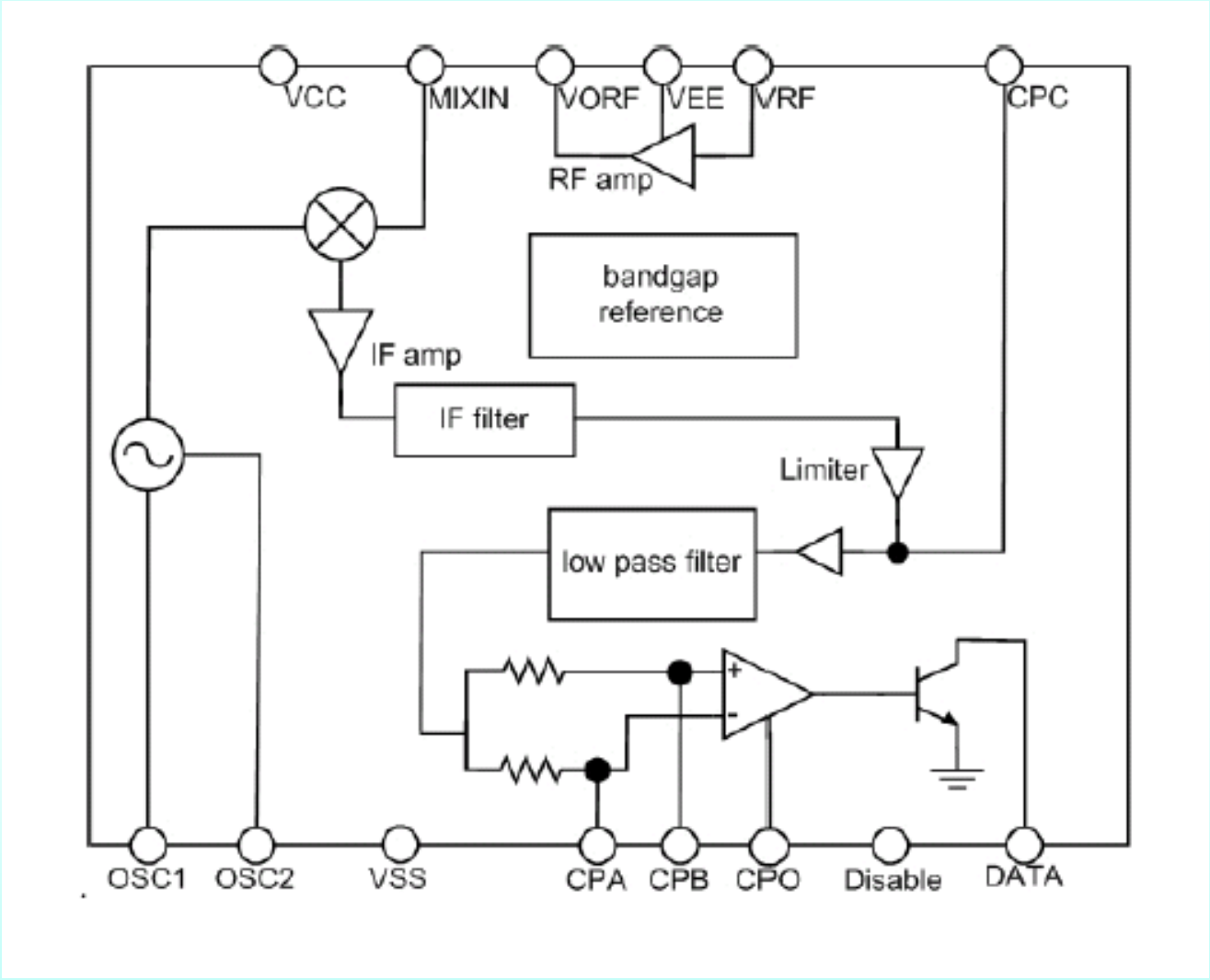
  Digital: high speed clock and signal lines

  Analog: high current/voltage, switching regulator
- Material
  - Metal sheet
  - Conductive Coating
- Openable cover for repair
- Opening for Alignment and test points
- More contact surface for cover

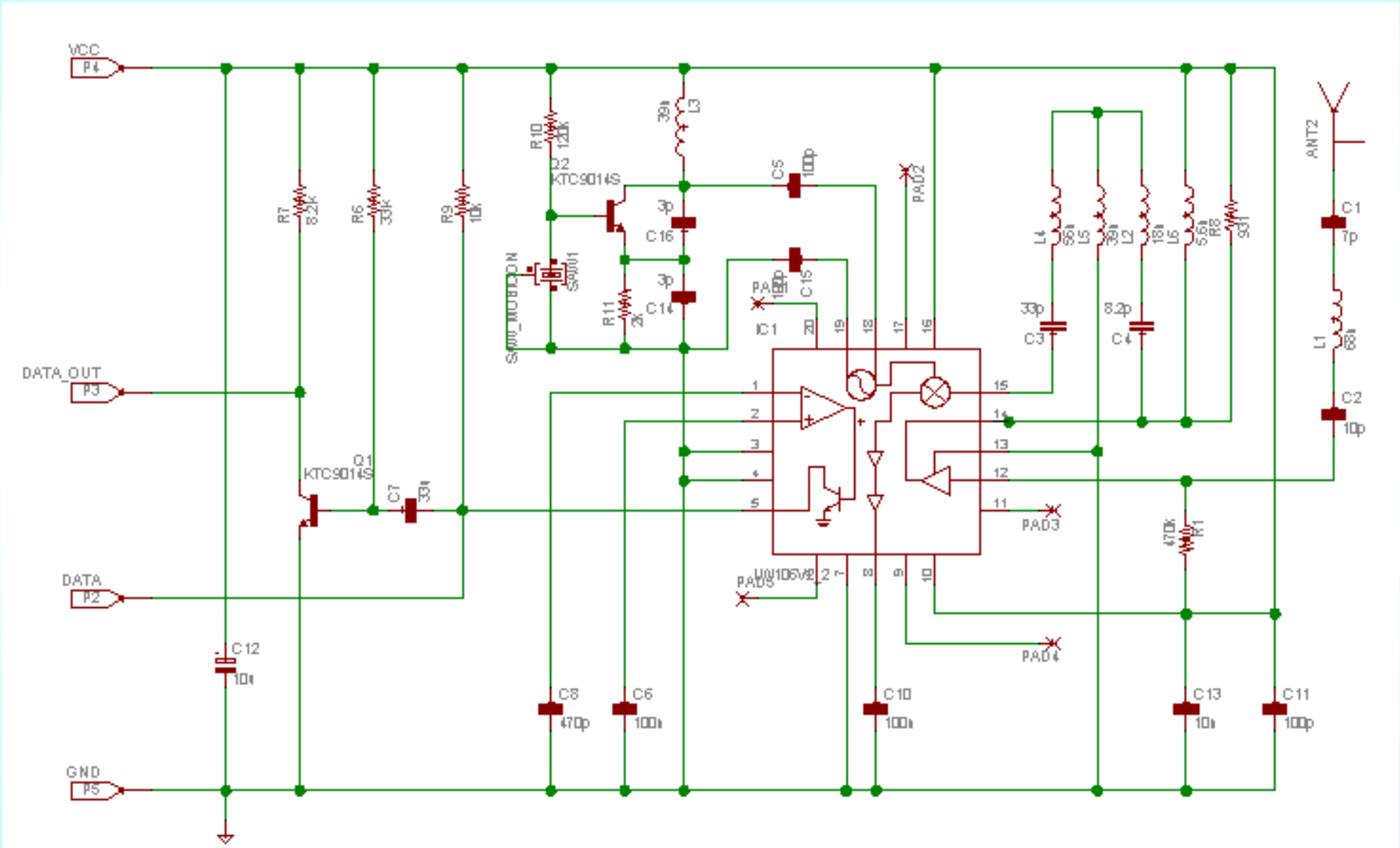
PCB Design for LW106M

- 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

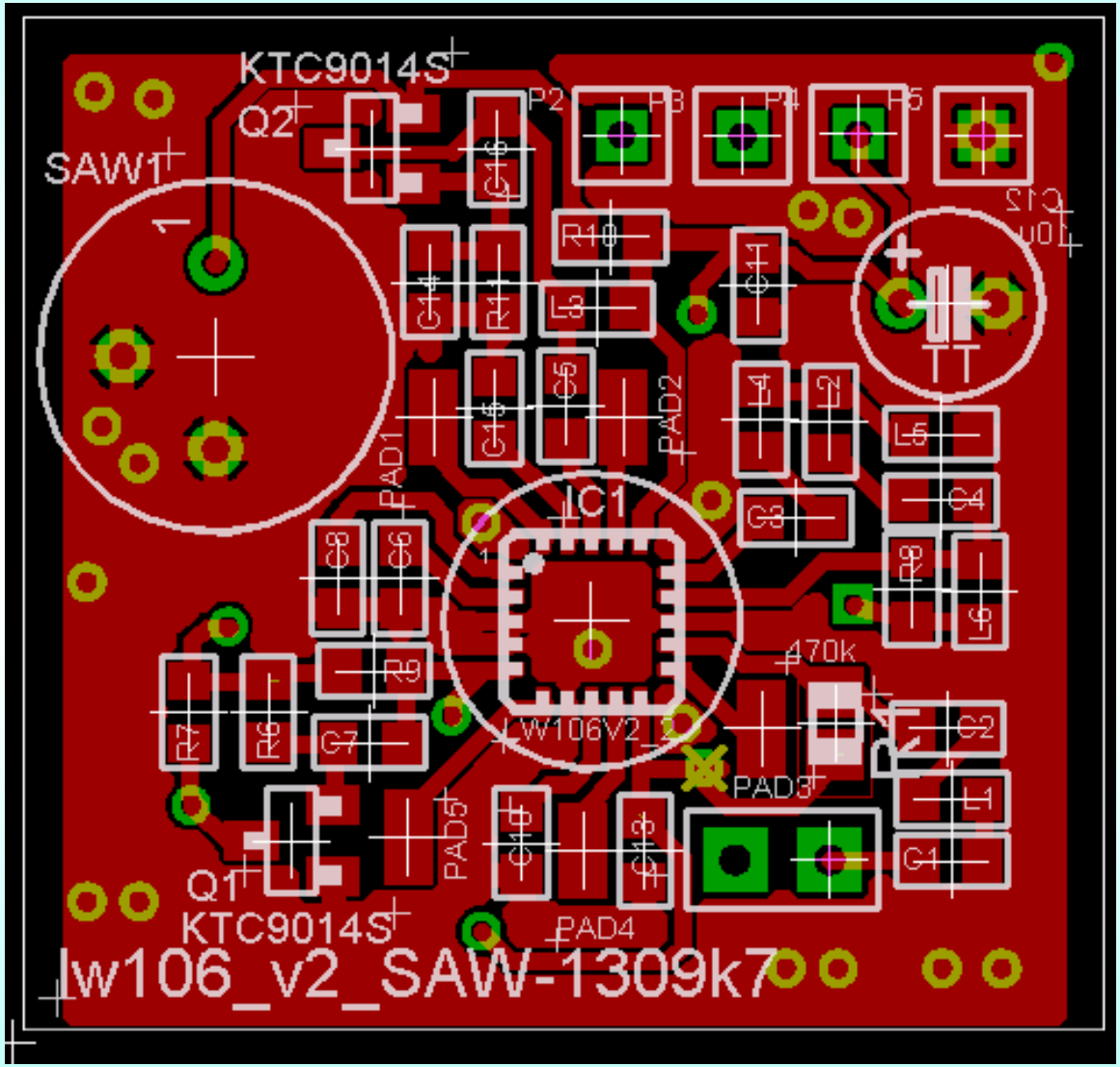
LW106 Block Diagram



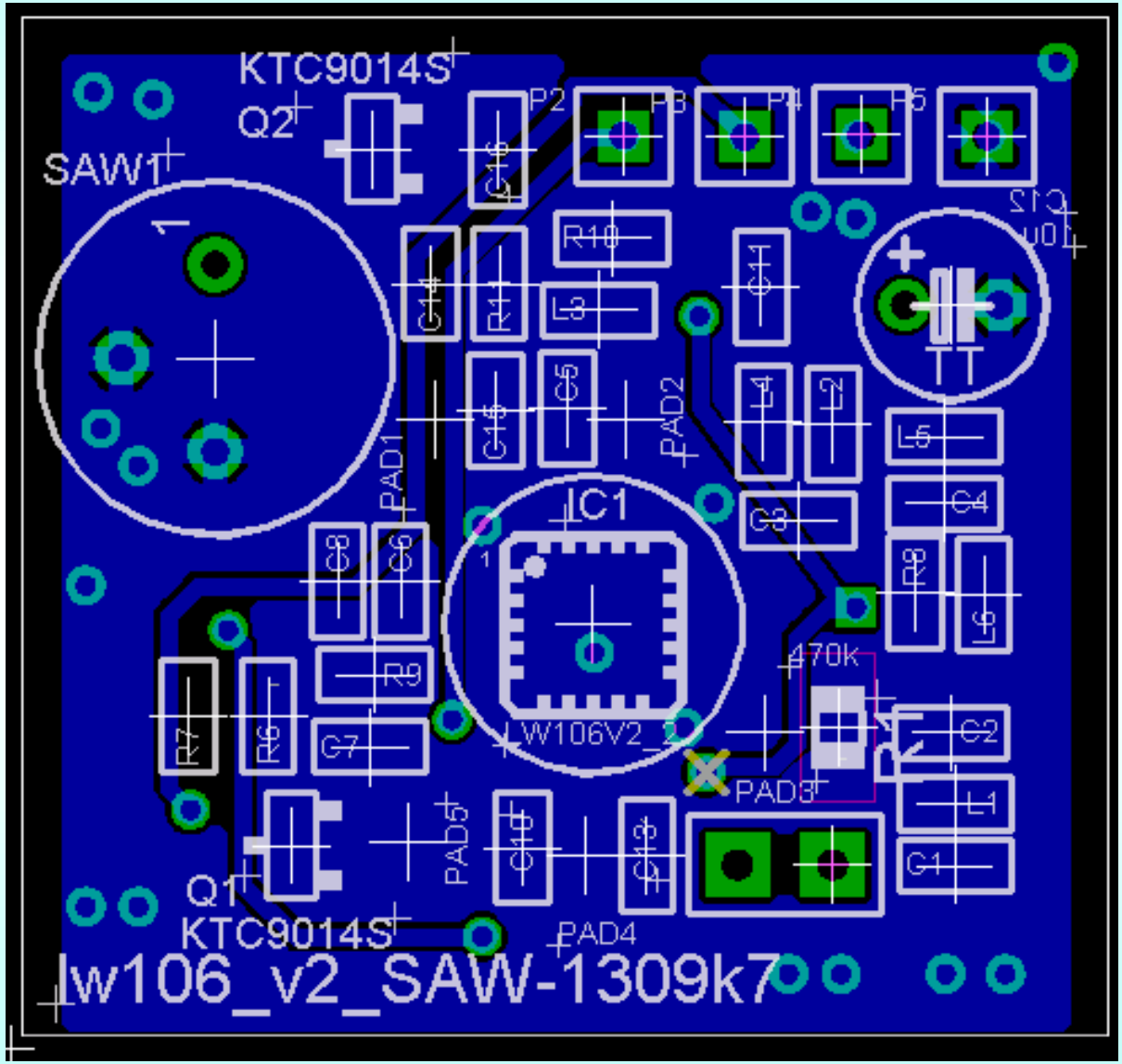
LW106M Schematic Diagram



LW106M PCB Top Layer



LW106M PCB Bottom Layer



Case Study – Interactive Toy

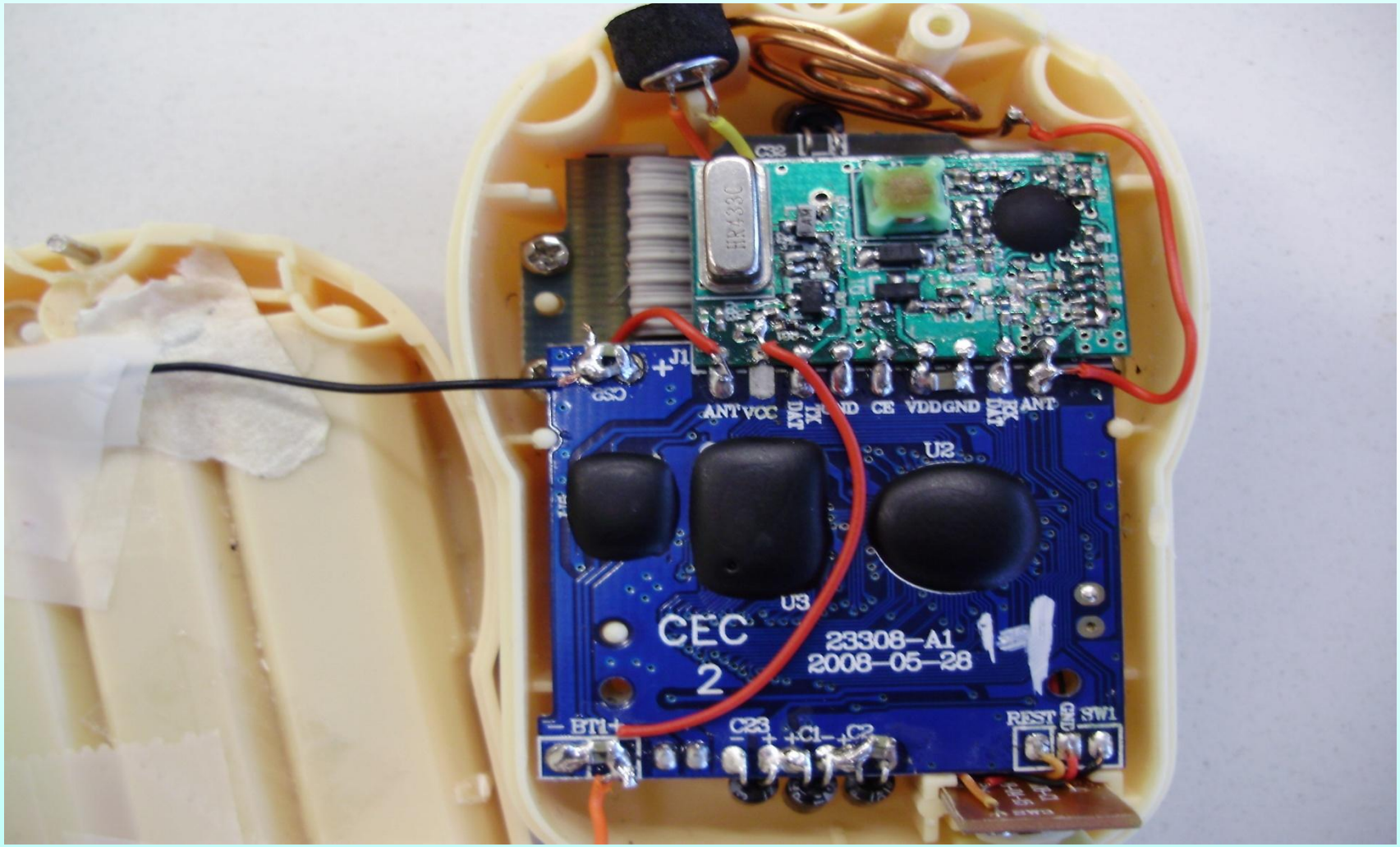
- Interactive Doll – Huru-Humi
  - Bi-directional RF datalink
    - Communicate with each other
    - Voice recognition
    - Link up to 6 units
    - Short distance
  - On sale at
    - Wal-mart
    - Target
    - Toys “R” us

Case Study – Interactive Toy

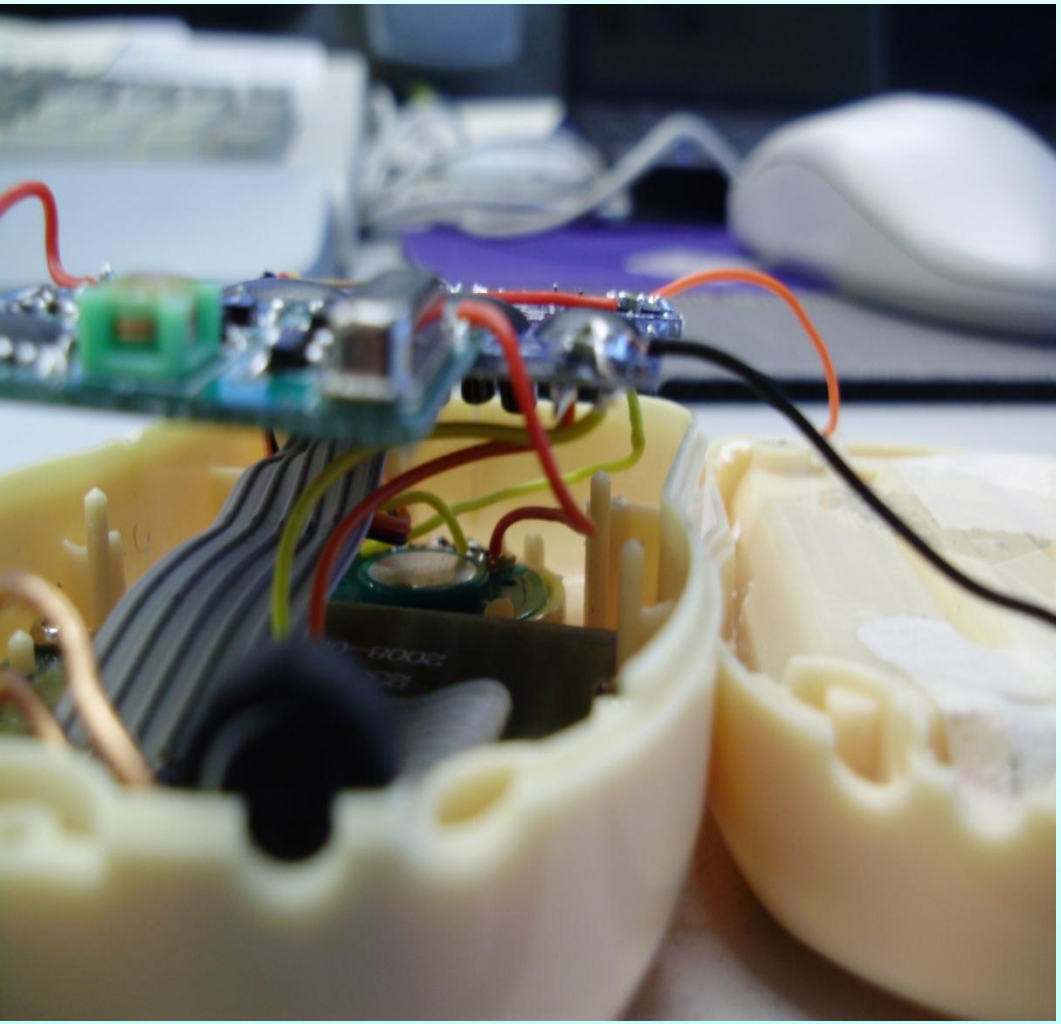
- Key Building Blocks
  - MCU
  - External ROM for speeches
  - MCU address extender
  - LCD driver and display
  - RF Transceiver Module
  - Audio amplifier
  - Microphone amplifier

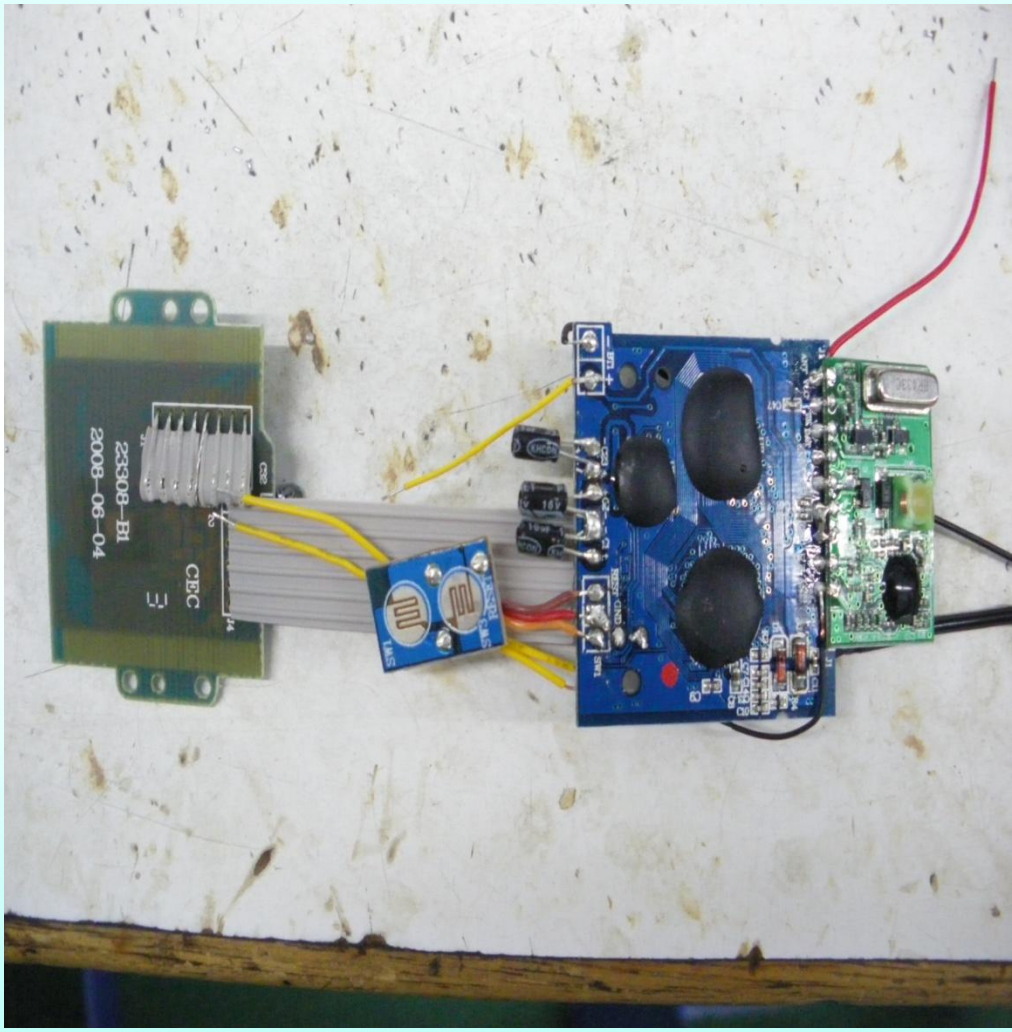
Case Study – Interactive Toy

- Original PCB – poor communication distance



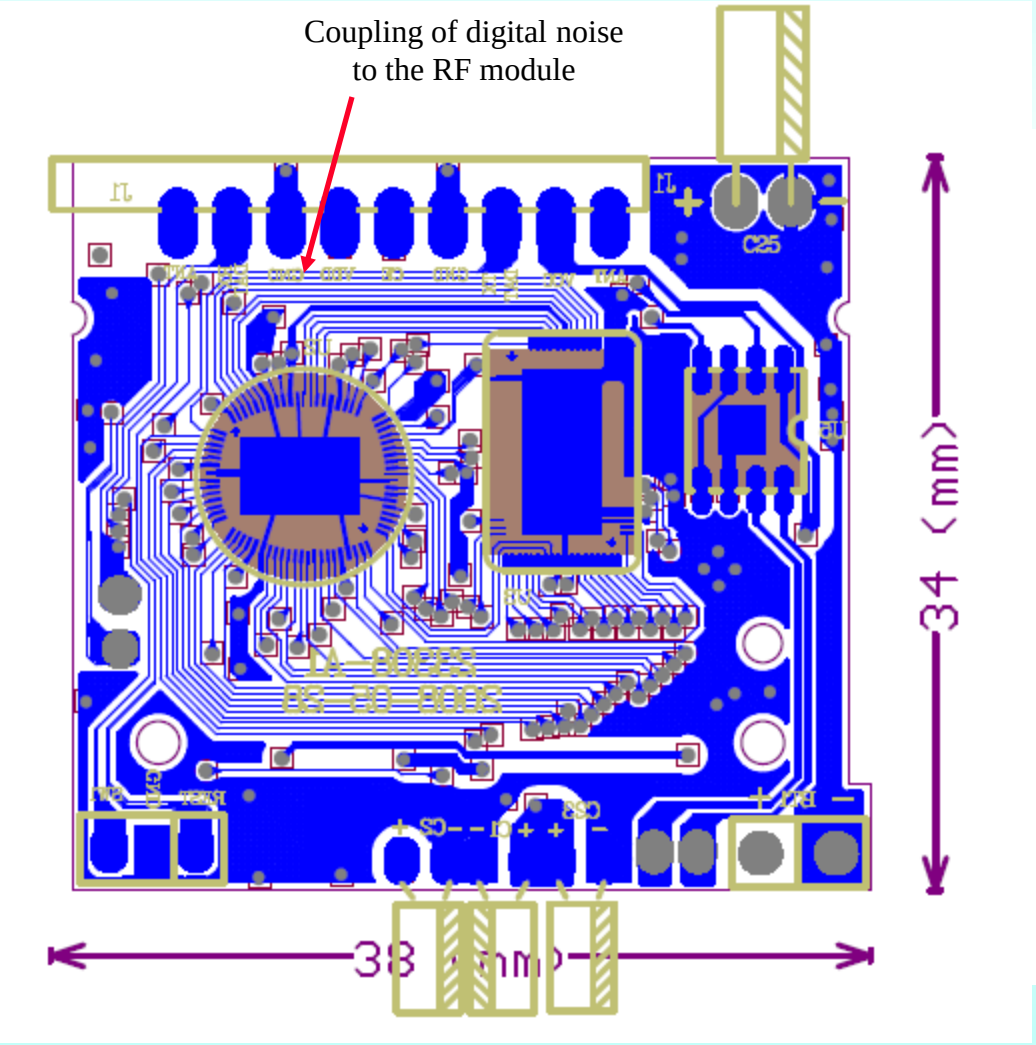
Case Study – Interactive Toy

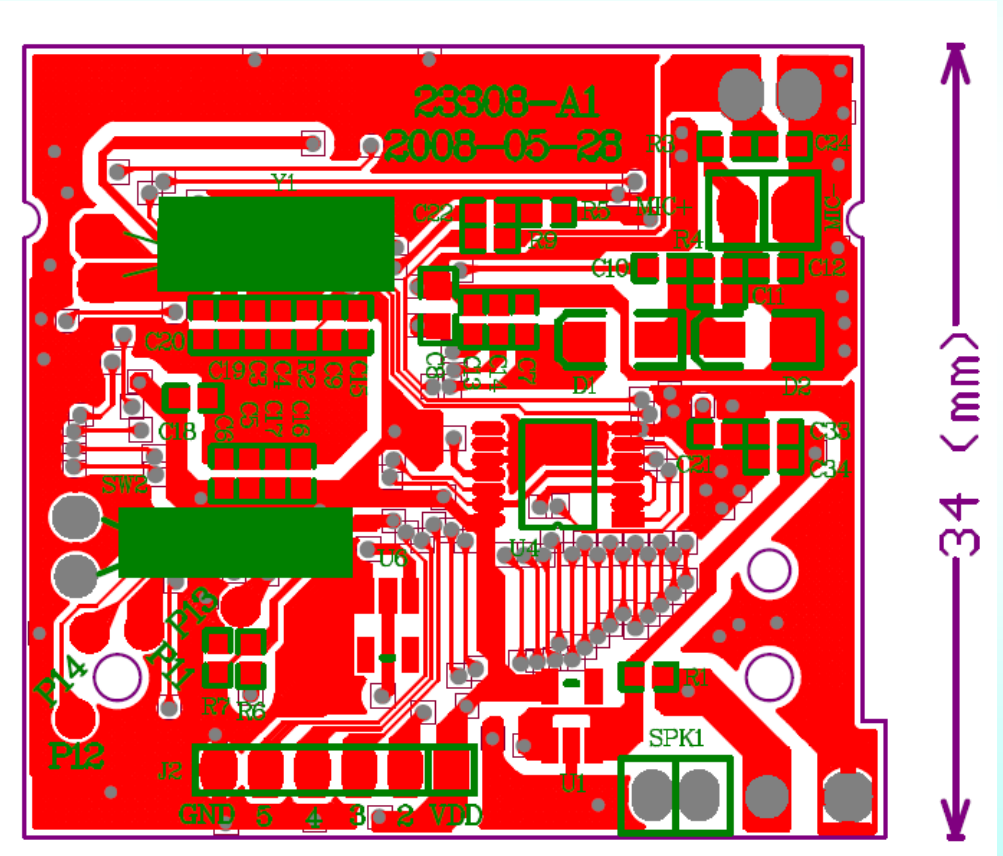




Case Study – Interactive Toy

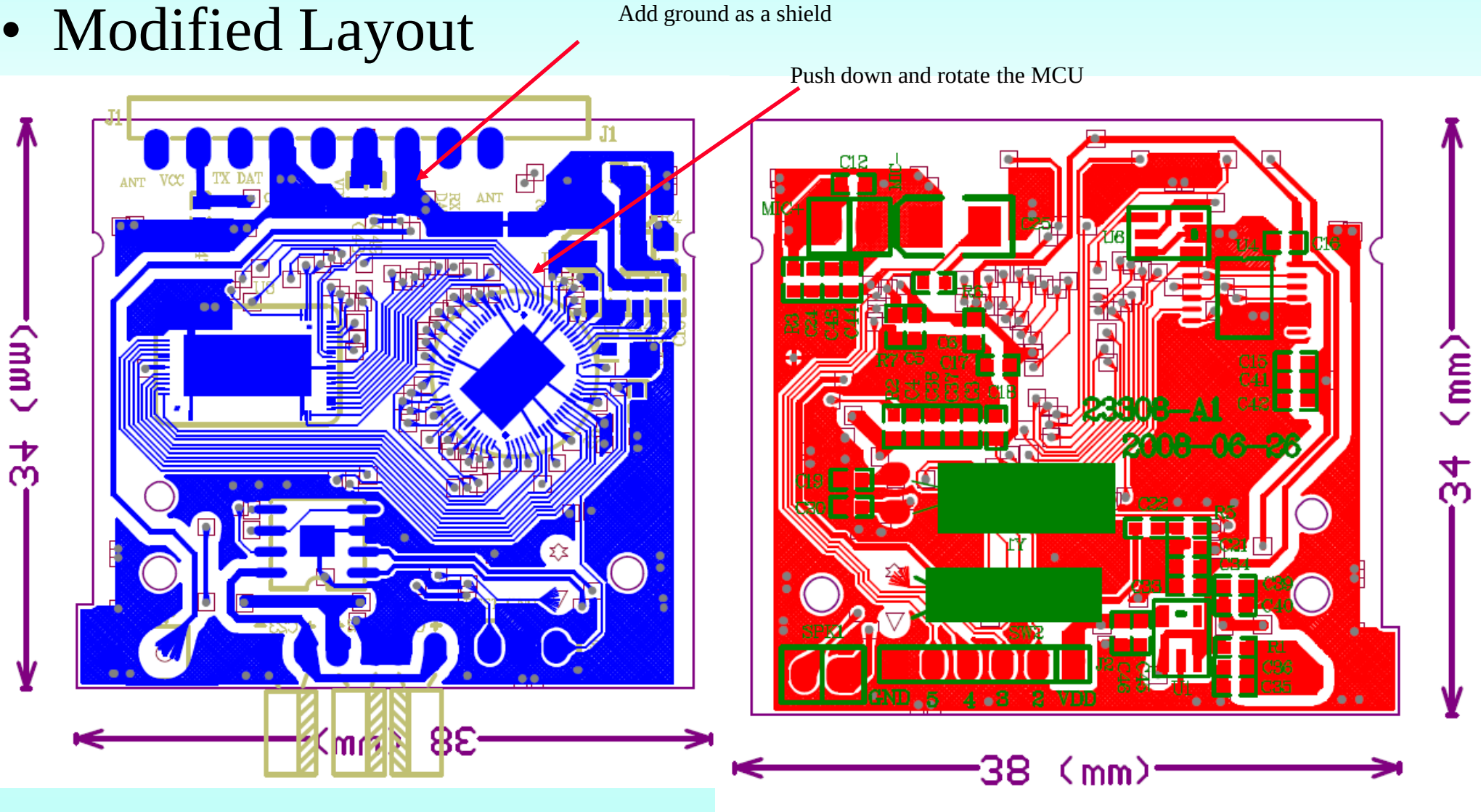
- Original Layout





Case Study – Interactive Toy

- Modified Layout



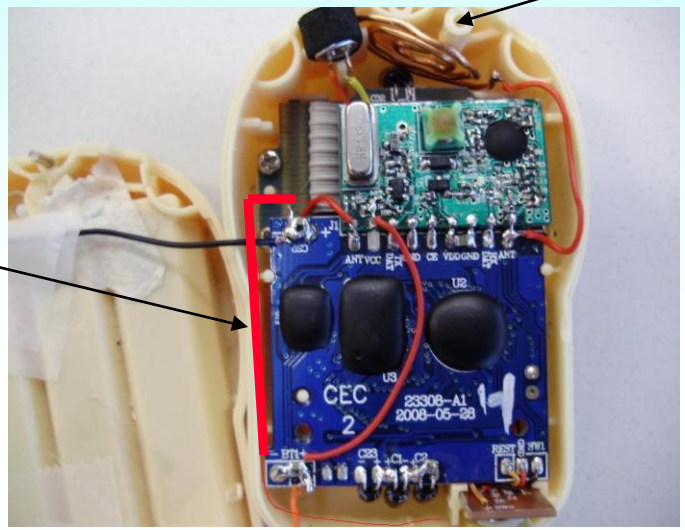
Case Study – Interactive Toy

- Antenna Structure

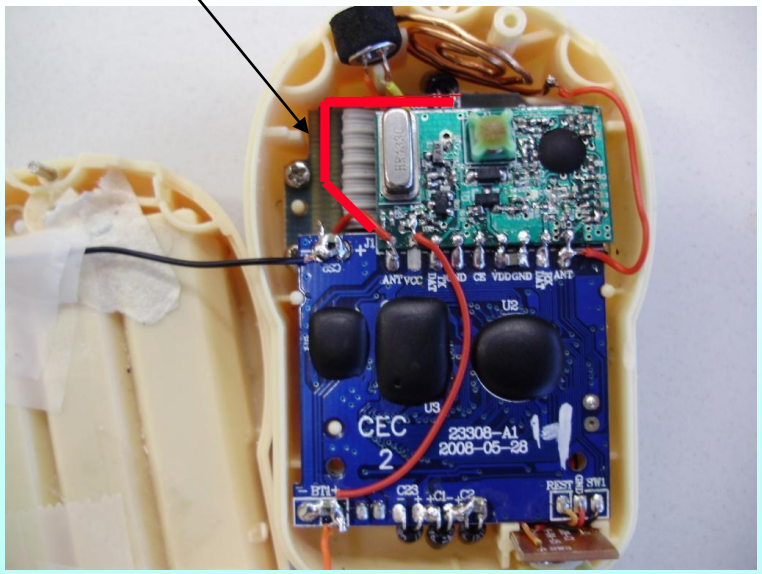
Improved version –

Spiral antenna

Original monopole antenna

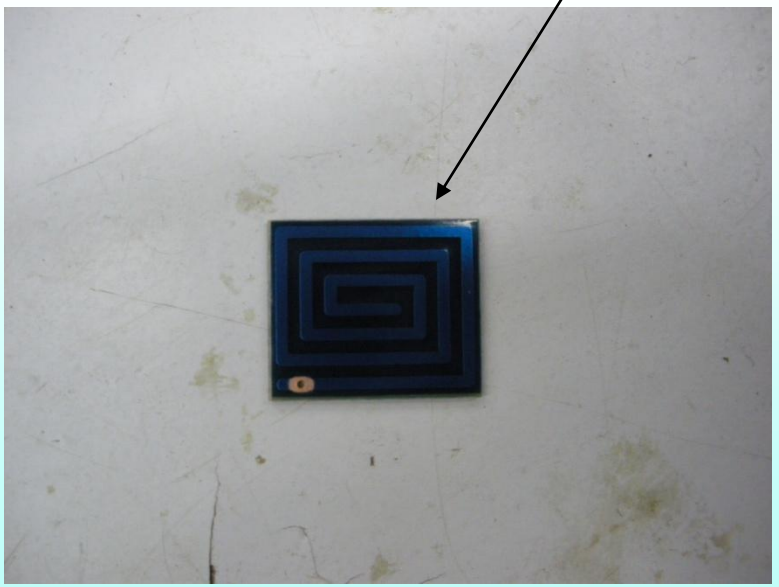


Another suggested antenna



Final production version –

Spiral PCB antenna

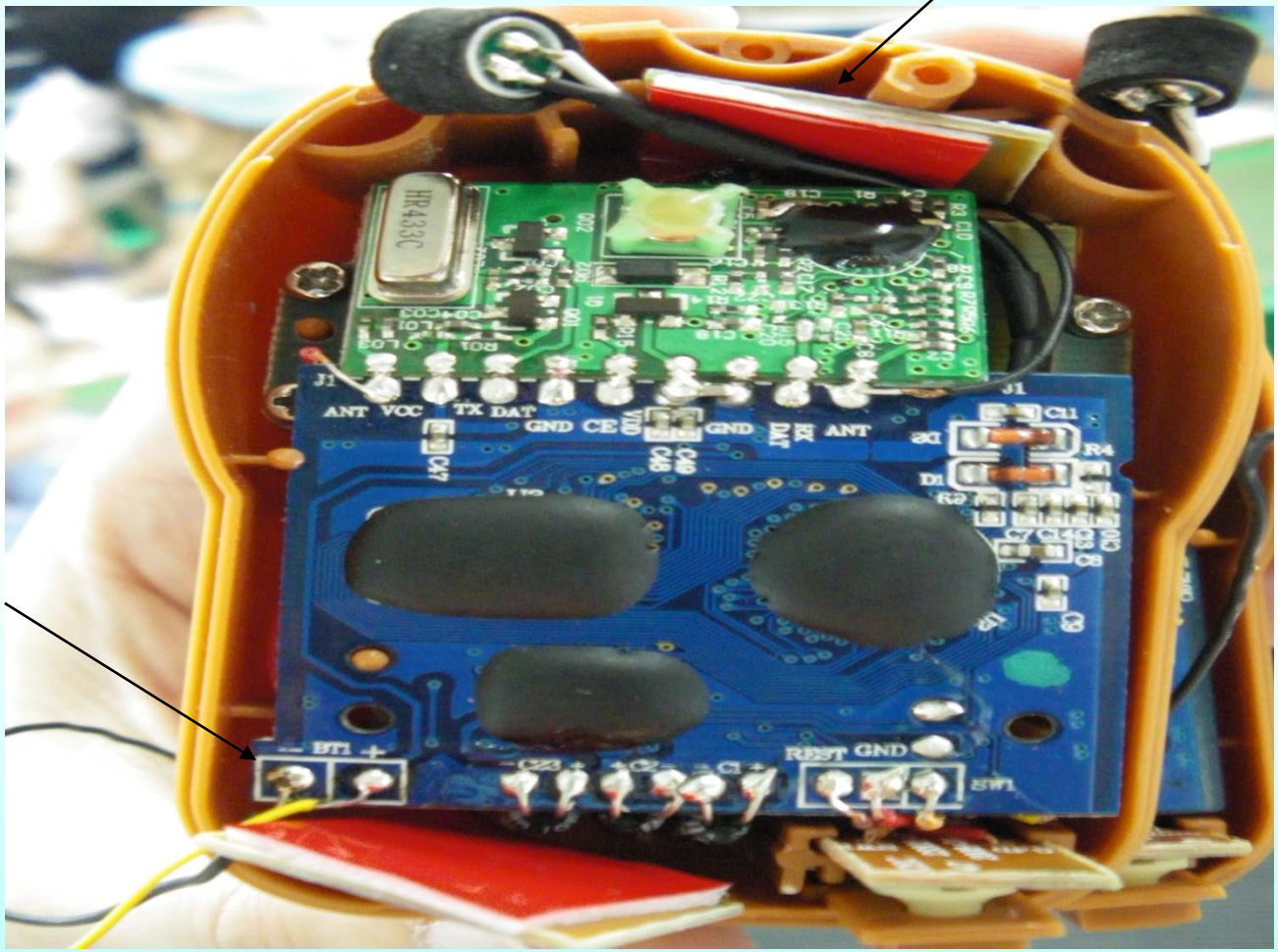


Case Study – Interactive Toy

- Modified PCB

Final production version –

Spiral PCB antenna



Final production version –

Spiral PCB antenna

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 sponsor
National Instruments (formerly AWR Corp.)

www.ni.com/awr