A Practical Guide to High-Speed Printed Circuit Board Layout

John Ardizzoni  
Analog Devices

Dennis Falls  
Avnet Electronics Marketing
Agenda

- Overview
- Schematic
- Location, location, location
- Trust no one
- Power supply bypassing
- Parasitics
- Ground and power planes
- Packaging
- RF Signal routing and shielding
- Checking the layout
- Summary
Overview

- PCB layout is one of the last steps in the design process and often one of the most critical
- High-speed circuit performance is heavily dependant on layout
- A high-performance design can be rendered useless due to a poor or sloppy layout
- Today’s presentation will help:
  - Improve the layout process
  - Ensure expected circuit performance is achieved
  - Reduce design time
  - Lower cost
  - Lower stress for you and the PCB designer
Schematic
The strength of any structure (including PCB’s) is only as good as the foundation on which it built upon!

A good layout starts with a good Schematic!

Schematic flow and content

Include as much information as you can

What should you include?
Items to Include on a Schematic

- Notes
- Component tolerances and case sizes
- Part numbers (internal/external/alternative)
- Board stack up
- Tests or alignment procedures
- Power dissipation
- Controlled impedance and line matching
- Component de-rating
- Thermal requirements
- Keep outs
- Mechanical considerations
- Critical component placement
- Warning flags
- Whatever else you can think of!
**Schematic**

**Freq. Adjust**
1.0 C2=C3, use these 2 capacitors to adjust the -3dB BW.

- **R1**: 1K
- **R2**: 50
- **R3**: 562
- **R4**: 210
- **R5**: 562
- **R6**: 301
- **R7**: 50

**U1**: 40 MHz
**U2**: 40 MHz

**C1**: 0.1uF
**C2**: 0.01uF
**C3**: 0.01uF
**C4**: 2.2uF
**C5**: 0.01uF
**C6**: 0.01uF
**C7**: 2.2uF

**Notes:**
1.0 All resistors and capacitors are 0603 case size unless noted otherwise.
2.0 All Resistors in ohms unless noted otherwise.
3.0 All capacitors in pF unless noted otherwise.
4.0 Run analog traces on Signal 1 layer, run digital traces on Signal 2 layer
5.0 Remove ground plane on all layers under the mounting pins of U2
6.0 U1 SOIC-14, U2 SOT-23-6, U3, SOIC-8, U4 SOIC-8

**Board Stack Up**

**ADP567**
**AD590**
**U4 Temperature Sensor**

**Signal 1**
**Analog Ground 1**
**Power plane**
**Digital Ground**
**Analog Ground 2**
**Signal 2**

**0.062”**
Location, location, location!
Location, Location, Location

- Just as in real estate location is everything!
- Input/output and power connections are typically defined...Everything else is undefined
  - Critical component placement
  - Signal routing
  - Circuit and component proximity
Trust No One
Trust No One

- If you’re doing your own layout, that’s one thing.
- If you’re not ....
Trust No One

- If you’re doing your own layout, that’s one thing.
- If you’re not ....
  - Don’t assume the CAD group is going to read your mind and get it right!
Trust No One

- If you’re doing your own layout, that’s one thing.
- If you’re not ....
  - Don’t assume the CAD group is going to read your mind and get it right!
  - In the end you’re responsible for making it work!
Trust No One

- If you’re doing your own layout, that’s one thing.
- If you’re not ....
  - Don’t assume the CAD group is going to read your mind and get it right!
  - You’re responsible for making it work!
- When working with the CAD Group
Trust No One

- If you’re doing your own layout, that’s one thing.
- If you’re not ....
  - Don’t assume the CAD group is going to read your mind and get it right!
  - You’re responsible for making it work!
- When working with the CAD Group
  - Make sure you and the designer are on the same page
Trust No One

- If you’re doing your own layout, that’s one thing.
- If you’re not ....
  - Don’t assume the CAD group is going to read your mind and get it right!
  - You’re responsible for making it work!
- When working with the CAD Group
  - Make sure you and the designer are on the same page
  - Brief circuit explanation
Trust No One

◆ If you’re doing your own layout, that’s one thing.
◆ If you’re not ....
  ● Don’t assume the CAD group is going to read your mind and get it right!
  ● You’re responsible for making it work!
◆ When working with the CAD Group
  ● Make sure you and the designer are on the same page
  ● Brief circuit explanation
  ● Critical component placement
Trust No One

- If you’re doing your own layout, that’s one thing.
- If you’re not ....
  - Don’t assume the CAD group is going to read your mind and get it right!
  - You’re responsible for making it work!
- When working with the CAD Group
  - Make sure you and the designer are on the same page
  - Brief circuit explanation
  - Critical component placement
  - Input/Output connections
Trust No One

◆ If you’re doing your own layout, that’s one thing.
◆ If you’re not ....
  ● Don’t assume the CAD group is going to read your mind and get it right!
  ● You’re responsible for making it work!
◆ When working with the CAD Group
  ● Make sure you and the designer are on the same page
  ● Brief circuit explanation
  ● Critical component placement
  ● Input/Output connections
  ● Board outline drawing and stack up
Trust No One

◆ If you’re doing your own layout, that’s one thing.
◆ If you’re not ....
  ● Don’t assume the CAD group is going to read your mind and get it right!
  ● You’re responsible for making it work!
◆ When working with the CAD Group
  ● Make sure you and the designer are on the same page
  ● Brief circuit explanation
  ● Critical component placement
  ● Input/Output connections
  ● Board outline, stack up
  ● Tell them to call you if they have a question!
Power Supply Bypassing
Power Supply Bypassing

- Bypassing is essential to high speed circuit performance
Power Supply Bypassing

- Bypassing is essential to high speed circuit performance
- Capacitors right at power supply pins
Power Supply Bypassing

- Bypassing is essential to high speed circuit performance
- Capacitors right at power supply pins
  - Capacitors provide low AC impedance to ground
  - Provide local charge storage for fast rising/falling edges
Power Supply Bypassing

- Bypassing is essential to high speed circuit performance
- Capacitors right at power supply pins
  - Capacitors provide low AC impedance to ground
  - Provide local charge storage for fast rising/falling edges
- Keep trace lengths short

Equivalent decoupled power line circuit resonates at:

\[ f = \frac{1}{2\pi \sqrt{LC}} \]

\[ f = 500\text{kHz} \]
Power Supply Bypassing

- Bypasing is essential to high speed circuit performance
- Capacitors right at power supply pins
  - Capacitors provide low AC impedance to ground
  - Provide local charge storage for fast rising/falling edges
- Keep trace lengths short

![Diagram showing IC with R1, L1, C1, and +Vs connections]
Power Supply Bypassing

- Bypassing is essential to high speed circuit performance
- Capacitors right at power supply pins
  - Capacitors provide low AC impedance to ground
  - Provide local charge storage for fast rising/falling edges
- Keep trace lengths short
- Close to load return
  - Helps minimize transient currents in the ground plane
Power Supply Bypassing

- Bypassing is essential to high speed circuit performance
- Capacitors right at power supply pins
  - Capacitors provide low AC impedance to ground
  - Provide local charge storage for fast rising/falling edges
- Keep trace lengths short
- Close to load return
  - Helps minimize transient currents in the ground plane
- Values
  - Individual circuit performance
Power Supply Bypassing

- Bypassing is essential to high speed circuit performance
- Capacitors right at power supply pins
  - Capacitors provide low AC impedance to ground
  - Provide local charge storage for fast rising/falling edges
- Keep trace lengths short
- Close to load return
  - Helps minimize transient currents in the ground plane
- Values
  - Individual circuit performance
  - Maintains low AC impedance
Power Supply Bypassing

- Bypassing is essential to high speed circuit performance
  - Capacitors right at power supply pins
    - Capacitors provide low AC impedance to ground
    - Provide local charge storage for fast rising/falling edges
  - Keep trace lengths short
  - Close to load return
    - Helps minimize transient currents in the ground plane
- Values
  - Individual circuit performance
  - Maintains low AC impedance
  - Multiple resonances
Power Supply Bypassing

- Bypassing is essential to high speed circuit performance
- Capacitors right at power supply pins
  - Capacitors provide low impedance AC return
  - Provide local charge storage for fast rising/falling edges
- Keep trace lengths short
- Close to load return
  - Helps minimize transient currents in the ground plane
- Values
  - Individual circuit performance
  - Maintains low AC impedance
  - Multiple resonances
- Ferrite beads
Parasitics
Parasitics

- Parasite – *An organism that grows, feeds, and is sheltered on or in a different organism while contributing nothing to the survival of its host.*

- *Parasitics in high-speed PCB’s, can destroy circuit performance!*
Parasitics

- PCB parasites take the form of undesired capacitors, inductors and resistors embedded within the PCB
- Parasitics are extremely difficult to remove from a PCB
- Prevention is the best method to minimize parasitics
Trace/Pad Capacitance

\[ C = \frac{kA}{11.3d} \]

- **K** = relative dielectric constant
- **A** = area in cm\(^2\)
- **d** = spacing between plates in cm
Trace/Pad Capacitance

\[ C = \frac{kA}{11.3d} \]

- \( K = \) relative dielectric constant
- \( A = \) area in cm\(^2\)
- \( d = \) spacing between plates in cm

Example: Pad of SOIC

- \( L = 0.2\text{cm} \quad W = 0.063\text{cm} \)
- \( K = 4.7 \)
- \( A = 0.0126\text{cm}^2 \)
- \( d = 0.073\text{cm} \)
- \( C = 0.072\text{pF} \)
**Trace/Pad Capacitance**

\[ C = \frac{kA}{11.3d} \]

K = relative dielectric constant
A = area in cm²
d = spacing between plates in cm

**Example: Pad of SOIC**

L = 0.2cm  W = 0.063cm
K = 4.7
A = 0.0126cm²
d = 0.073cm
C = 0.072pF

**Reduce Capacitance**
1) Increase board thickness or layers
2) Reduce trace/pad area
3) Remove ground plane
Approximate Trace Inductance

\[
\text{STRIP INDUCTANCE} = 0.0002L \left\{ \ln \left( \frac{2L}{W+H} \right) + 0.2235 \left( \frac{W+H}{L} \right) + 0.5 \right\} \mu\text{H}
\]

All dimensions are in mm
Approximate Trace Inductance

\[ \text{STRIP INDUCTANCE} = 0.0002L \left( \ln \left( \frac{2L}{W+H} \right) + 0.2235 \left( \frac{W+H}{L} \right) + 0.5 \right) \ \mu\text{H} \]

Example

- \( L = 25.4\text{mm} \)
- \( W = 0.25\text{mm} \)
- \( H = 0.035\text{mm} \) (1oz copper)

Strip Inductance = 28.8nH

At 10MHz \( Z_L = 1.86 \Omega \) a 3.6% error in a 50\( \Omega \) system

All dimensions are in mm
Approximate Trace Inductance

![Diagram of a trace with dimensions L, W, and H]

Strip Inductance = \(0.0002L \left[ \ln \left( \frac{2L}{W+H} \right) + 0.2235 \left( \frac{W+H}{L} \right) + 0.5 \right] \mu H\)

All dimensions are in mm

Example

L = 2.54cm = 25.4mm
W = .25mm
H = .035mm (1oz copper)

Strip Inductance = 28.8nH

At 10MHz \(Z_L = 1.86 \Omega\) a 3.6% error in a 50\(\Omega\) system

Minimize Inductance

1) Use Ground plane
2) Keep length short (halving the length reduces inductance by 44%)
3) Doubling width only reduces inductance by 11%
Via Parasitics

Via Inductance

\[ L \approx 2h \left[ \ln\left( \frac{4h}{d} \right) + 1 \right] nH \]

\( L \) = inductance of the via, \( nH \)
\( H \) = length of via, \( cm \)
\( D \) = diameter of via, \( cm \)

Given:
\( H = 0.157 \text{ cm thick board, } \)
\( D = 0.041 \text{ cm} \)

\[ L \approx 1.2nh \]

Via Capacitance

\[ C \approx \frac{0.55\varepsilon_r TD_1}{D_2 - D_1} \text{ pF} \]

\( D_2 \) = diameter of clearance hole in the ground plane, \( cm \)
\( D_1 \) = diameter of pad surrounding via, \( cm \)
\( T \) = thickness of printed circuit board, \( cm \)
\( \varepsilon_r \) = relative electric permeability of circuit board material
\( C \) = parasitic via capacitance, \( pF \)

Given:
\( T = 0.157cm, \)
\( D_1 = 0.071cm \)
\( D_2 = 0.127 \)

\[ C \approx 0.51pf \]
Via Cross Section
Capacitor Parasitic Model

C = Capacitor
RP = insulation resistance
RS = equivalent series resistance (ESR)
L = series inductance of the leads and plates
RDA = dielectric absorption
CDA = dielectric absorption
Resistor Parasitic Model

\[ R = \text{Resistor} \]
\[ C_p = \text{Parallel capacitance} \]
\[ L = \text{equivalent series inductance (ESL)} \]
Low Frequency Op Amp Schematic
High Speed Op Amp Schematic
High Speed Op Amp Schematic

Parasitic Capacitance
NI Multisim Analog Devices Edition

Free downloadable SPICE simulator for Analog Devices component evaluation.

With NI Multisim™ Analog Devices Edition, ADI and the NI Electronics Workbench Group equip circuit board designers with a free, downloadable version of NI Multisim tailored for evaluating ADI components. Avoid costly and time consuming prototyping work by using this easy-to-use interactive SPICE simulator.

With the NI Multisim Analog Devices Edition, engineers can:
- Build simulated component evaluation circuits to quickly assess behavior of over 800 Analog Devices operational amplifiers, switches and voltage references
- Examine the unit under test in the intended circuit topology with up to 25 components
- Use built-in instruments and analyses including oscilloscopes and worst-case analysis
- Swap components easily to pinpoint best design options
- Link to the Analog Devices Design Center for more online evaluation tools
- Instantly access product pages and datasheets of each Analog Devices component
- Upgrade to a full edition of NI Multisim to complete designs and transfer to board layout with NI Ultiboard


Other Resources:
- Upgrade to the NI Multisim Professional Edition
Stray Capacitance Simulation Schematic
Frequency Response with 2pF Stray Capacitance

1.8dB peaking
Stray Inductance

Parasitic Inductance
Parasitic Inductance Simulation Schematic

24.5mm x .25mm” = 29nH
Pulse Response With and Without Ground Plane
Transient Response AD8009
1GHz Current Feedback Amplifier

![Diagram of the AD8009 amplifier circuit with labels for components such as $R_F$, $R_G$, $V_I$, $+5V$, $-5V$, $10\mu F$, $0.1\mu F$, and $150\Omega$.](image-url)
Small Changes Can Make a Big Difference!

Circuit A

Circuit B
Improper Use of Scope Probe Ground Clip
Effect of Clip Lead Inductance
Proper Grounding for Scope Probe in High-Speed Measurements
Small Changes Make Big Differences

- **Circuit A**: 21ns
- **Circuit B**: 17ns

25% reduction in ringing duration and amplitude
Ground and Power Planes
Ground and Power Planes Provide

- A common reference point
- Shielding
- Lower noise
- Lower resistance
- Lower impedance
- Reduces parasitics
- Heat sink
- Power distribution
Ground Plane
Ground Plane and Trace Routing

Wrong Way
Ground Plane and Trace Routing

Wrong Way

Sensitive Analog Circuitry Disrupted by Digital Supply Noise

 Incorrect
Ground Plane and Trace Routing

Right Way

Sensitive Analog Circuitry Safe from Digital Supply Noise

CORRECT

\( V_D \) \( V_A \) \( V_{IN} \) \( I_D \) \( I_A \)
Ground Plane and Trace Routing

Right Way

Sensitive Analog Circuitry Safe from Digital Supply Noise
Ground Plane and Trace Routing

Grounding Example:

- Top layer is solid ground.
- Bottom has a trace/transmission line connecting the RF connector to the load.
- Return current flows in the top layer ground plane directly above the trace on the opposite side.
Ground Plane and Trace Routing

Grounding Example: DC Current vs. AC Current:

- In a split or broken ground, the return currents follow the path of least impedance
- At DC, the current follows the path of least resistance
- As the frequency increases, the current follows the path of least inductance
- Since there is now a ‘loop’ the inductance can be quite high and the circuit can now propagate EMI/RFI
Grounding Mixed Signal ICs: Single PC Board
Ground Plane Recommendations

- There is no single grounding method which is guaranteed to work 100% of the time!
- Remove ground plane under op amps to reduce parasitic capacitance
- At least one layer on each PC board MUST be dedicated to ground plane!
- Provide as much ground plane as possible especially under traces that operate at high frequency
- Use thickest metal as feasible (reduces resistance and provides improved thermal transfer)
- Use multiple vias to connect same ground planes together
- Do initial layout with split analog and digital ground planes
- Follow recommendations on device data sheet (read datasheet)
- Keep bypass capacitors and load returns close to reduce distortion
- Connect analog, digital and RF grounds at one point
Packaging and Pinout
Op Amp Packaging and Pinout

- Packaging plays a large role in high-speed applications

**Smaller packages**
- Better at higher speeds
- Less parasitics
- Compact layout

**Analog Devices Low Distortion Pinout**
- Intuitively makes more sense
- Compact layout
- Streamline signal flow
- Lower distortion
Op Amp SOIC Packaging

- Traditional SOIC-8 layout
- Feedback routed around or underneath amplifier
Op Amp SOIC Packaging

- Traditional SOIC-8 layout
- Feedback routed around or underneath amplifier
Analog Devices Low Distortion Pinout

- Pinout enables compact layout

Original Pin-Out

NC 1 8 Disable
–IN 2 –
+IN 3 +
–VS 4 –

SOIC

LFCSP

NC 1 8 +VS
FEEDBACK 2 7 OUTPUT
–IN 3 –
+IN 4 +
–VS 5 –

NC
Analog Devices Low Distortion Pinout

- Pinout enables compact layout
- Lower distortion
Analog Devices Low Distortion Pinout

- Pinout enables compact layout
- Lower distortion
- Improved thermal performance
Analog Devices Low Distortion Pinout

- Pinout enables compact layout
- Lower distortion
- Improved thermal performance
- LFCSP
  - AD8099, AD8045, AD8000, ADA4899, ADA4857, ADA4817

Original Pin-Out

LFCSP
Low distortion pinout enables compact and streamline layout
Low distortion pinout enable compact and streamline layout
RF Signal Routing and Shielding
In This Section

- RF Components from Analog Devices
- PC Board Circuit Material Types and Minimizing Losses
- Microstrip and Stripline Transmission Lines
- Ground Plane Layout Considerations
- Developing a RF Printed Circuit Board
- Using Discrete Components with RF Devices
- Shielding of RF Circuit Boards
RF/IF Components

The ADF9010 is a fully integrated RF Tx modulator and Rx analog baseband front end that operates in the frequency range from 840 MHz to 3600 MHz. The receiver path consists of a fully differential I/O baseband PFA, low pass filter, and general signal conditioning before connecting to an RxADC for baseband conversion. Learn more about the ADF9010.

Product Categories

- DDS Modulators
- Digital Up / Down Converters
- Direct Digital Synthesis (DDS)
- Log Amps / Detectors
- Mixers / Multipliers
- Modulators / Demodulators
- PLL Synthesizers/VCOs
- RF/IF Amplifiers
- RF/IF Transceivers
- RF Switches
- RMS Detectors
- RX/TX Subsystems
- Short Range Transceivers

Analog Devices Components Requiring Matched RF Interfaces

- AD60x, AD8xxx and ADL533x series of RF/IF and Variable Gain Amplifiers
- ADF70xx and 702x series Radio Transmitters and Transceivers
- AD4xxx and ADF7xxx series of PLL Synthesizers and VCO’s
- AD84xx and ADL53xx series of Modulators and Demodulators
- AD83xx and ADL539x series of Mixers and Multipliers
- AD83xx and ADL5519 series of Log Amps and Detectors
- AD836x and ADL550x series of RMS Detectors
- ADG9xx series of RF Switches
### PC Board Circuit Material Type and Minimizing Losses

- PC board material selection is usually based on price verses performance
- Select PC board dielectric material to have the lowest loss tangent
- Some types of “FR4” dielectric materials are low loss below 8-10 GHz
- PTFE (Teflon) dielectric material is usually used for the lowest loss at the higher RF and microwave frequency ranges, but at a much higher price
- Be sure that the correct impedance transmission line is used for the interconnection of the RF devices
- Use as wide of a transmission line as possible for the correct impedance, and try to keep it short to reduce “Skin Effect” losses
- Use high “Q”, or low loss passive components for all RF matching, coupling, and bypassing requirements
Microstrip and Stripline Transmission Lines

- 50 ohm interfaces are most often used between most “integrated” RF devices
- Interconnects less than 1/20 of a wavelength long can usually be made without a matched transmission line
- Avoid long microstrip lines as they could become “antennas” (microstrip) and radiate RF
- To minimize coupling to the transmission line, DO NOT place other traces or ground plane closer than three times of the dielectric height
- Use proper technique for making bends in microstrip lines
- Locate the microstrip lines on the component side of the board if possible
Microstrip Transmission Lines

**Microstrip**
Controlled Impedance Line Cross Section

- Advantages:
  - Transmission line on outside layer of board
  - Easy to attach components to trace
  - Components can be placed at different locations along the line to aid in tuning
  - Aid in RF testing as you are able to measure levels along the line

- Disadvantages:
  - Slightly higher loss
  - Not shielded and could radiate RF signal

\[
Z_o = \frac{87}{\sqrt{\varepsilon_r + 1.41}} \ln \left[ \frac{5.98H}{(0.8W + T)} \right]
\]
Stripline Transmission Lines

Stripline
Controlled Impedance Line Cross Section

- Advantages:
  - Lower loss at higher microwave frequencies
  - Shielded transmission line, no RF radiation from board

- Disadvantages:
  - Requires vias to connect to line
  - No ability to connect tuning or termination components to line
  - No access to line to make adjustments or connections to line for RF testing
  - Higher PCB cost

\[
Z_o(\Omega) = \frac{60}{\sqrt{\varepsilon_r}} \ln \left[ \frac{1.9B}{(0.8W+T)} \right]
\]
Microstrip and Stripline Transmission Lines

- **Bends in Microstrip and Stripline**

  - In order to preserve a constant impedance around a bend, some general layout rules MUST be followed.
  - **DO NOT** make a right angle bend as shown.
  - A right angle bend can be made with a “swept” bend, or a “Mitered” bend.
  - Bends in a transmission line that are less than 90 degrees can also be mitered as shown.
Ground Plane Layout Considerations

- Do NOT have breaks or voids in the RF ground plane under, or over RF transmission lines
- Ground plane Vias around the RF circuits should be spaced closer than 1/20 of a wavelength as a minimum, or closer if possible at the higher frequencies
- Use as large size vias as practical to minimize inductance
- “Stitch” the top and bottom ground planes together with as many vias as possible
- Signal and bias lines can be placed below RF ground plane layer followed by another “power” ground plane layer for the DC and digital returns
- Under components that require heat sinking, have solid ground plane with many closely spaced vias to transfer heat to all ground plane layers
Ground Plane Layout Considerations

Closely spaced vias under components requiring heat sinking

50 ohm microstrip transmission line

Proper spacing of ground plane away from transmission line (gap= 3 times dielectric height)

Closely spaced Ground plane vias along transmission line

“Stitch” ground planes together with multiple vias

Vias placed in ground plane at each end of the DC bias bypass components

Proper layout pattern for connecting edge mounted RF connector to PC board

Maximum amount of ground plane on top and bottom sides of PC board
Developing a RF Printed Circuit Board

- Draw Schematic of circuit to be placed on the PC board
- Have data sheets on components to indicate pkg size, pinouts, etc
- Determine location and orientation of active devices to optimize RF interfaces
- Place RF matching/terminating components around the device to provide the shortest possible connections
- Use as small of mounting pad as possible with discrete RF components to keep stray capacitance to a minimum
- Observe proper orientation of discrete components if placed next to each other to avoid coupling effects
- Separate inductors from each other in the layout, or place perpendicular to each other to prevent coupling of their magnetic fields
- Make sure that components that are connected to the ground plane have a via(s) as close to the end of the component as possible
- Use wide power traces if possible to lower DC losses and provide higher stray capacitance to ground (will also act as a RF bypass cap)
Designing a RF Printed Circuit Board

- Draw Schematic of circuit
- Draw layout of components to optimize parts placement and interconnections
Designing a RF Printed Circuit Board

- Ground plane is not close to transmission line to reduce coupling
- Ground vias at end of RF and bypass components
- Place bias line bypassing components close to RF device
- Matching components placed close to RF device
- Large number of vias under devices to provide good RF grounds and thermal conductivity
- RF devices placed in a straight line to aid in overall circuit stability
- Inductors placed at right angles to reduce coupling of their magnetic fields
- Short interstage RF transmission lines between stages
- Very short ground leads from device to ground plane
- RF attenuator placed close to board connector
- ADL5523 LNA
- AD5350 Mixer
- AD8353 IF AMP
- LO Input
- RF Input
- IF Output
Shielding on RF Circuit Boards

- On multilayer circuit boards, use Stripline transmission lines if possible
- Route DC bias and signal traces on inner layers between the ground planes
- If required, place shielded enclosures around the RF stages on the board
- Be careful as to the physical size of the shielded enclosures, as it could become a resonate “cavity” at the higher frequencies
- Traces going to or from shielded sections should be routed on inner layers if possible

All board mounting holes should be plated through to provide good RF grounds to the external housing

On multilayer board use stripline transmission lines to provide shielding
Checking the Layout
Checking the Layout

- Design review
- Colleague review
Checking the Layout

- Design review
- Colleague
- Colored pencils
  - Old School
  - Helps trace signal path on schematic and PCB
Checking the Layout

- Design review
- Colleague
- Colored pencils
  - Old School
  - Helps trace signal path on schematic and PCB
- Sit with the designer when board corrections are made
  - Trust no one
  - A change in one area of the board could inadvertently change another part of the board
Next Steps

- Order Boards
- Build and test
- Evaluate performance
- Iterate and try again if required
- Successful High Speed/RF PCB design is a combination of education and experience
Summary
Summary

- High speed PCB design requires deliberate thought and attention to detail!
- Load the schematic with as much information as possible
- Where you put individual components on the board is just as important as to where you put entire circuits
- Take the lead when laying out your board, don’t leave anything to chance
- Use multiple capacitors for power supply bypassing
- Parasitics must be considered and dealt with
- Ground and Power planes play a key role in reducing noise and parasitics
- New packaging and pinout options allow for improved performance and more compact layouts
- There are many options for signal distribution, make sure you choose the right one for your application
- Check the layout and check it again
- Successful High Speed PCB design is a combination of education and experience and sometimes a little luck!
Summary

- Work directly with PC board designer as they most likely will not understand proper RF layout techniques
- Provide designer with a drawing of the location of the critical high frequency components and transmission lines
- Instruct the board designer that transmission line widths and lengths are very critical and must be exactly as calculated
- Place the components to minimize the length of RF interconnections
- Generally try to place components in a “straight line” to avoid feedback loops and instabilities
- Place circuit blocks such as oscillators, mixers, amplifiers in separate sections on the board if possible
- Do NOT mix digital, low level analog, or bias traces with RF interconnects to avoid unwanted coupling
- Locate the components operating at the highest frequencies close to board interconnects
- With the PC board designer, check, and recheck the layout before sending out for fabrication
References

- Ardizzoni, John “A Practical Guide to High-Speed Printed-Circuit-Board Layout”
- Brokaw, Paul and Jeff Barrow, “Grounding for Low- and High-Frequency Circuits,” Analog Devices Application Note AN-345.
- Kester, Walt, The Data Conversion Handbook, Elsevier-Newnes, 2005 available on Amazon.com
References

- Hartley, Rick, “RF / Microwave PC Board Design and Layout”
- Reed, Dale, RF and Microwave Basics Impact PCB Design
- Howe, Harlan, “Stripline Circuit Design”
- Rogers Corporation, “Microwave Impedance Calculator (MWIJ 1.0)”
## Contact Information

<table>
<thead>
<tr>
<th>John Ardizzoni</th>
<th>Dennis Falls</th>
</tr>
</thead>
<tbody>
<tr>
<td>Analog Devices, Inc.</td>
<td>Avnet Electronics Marketing</td>
</tr>
<tr>
<td>804 Woburn Street</td>
<td>9200 Indian Creek Parkway</td>
</tr>
<tr>
<td>Wilmington, MA 01887-1017</td>
<td>Suite 600</td>
</tr>
<tr>
<td></td>
<td>Overland Park, Kansas 66210</td>
</tr>
<tr>
<td>Email: <a href="mailto:john.ardizzoni@analog.com">john.ardizzoni@analog.com</a></td>
<td>E-mail: <a href="mailto:Dennis.falls@avnet.com">Dennis.falls@avnet.com</a></td>
</tr>
</tbody>
</table>

Thank You