Impedance Basics

A Beginner's Guide to Impedance Control in PCB Design

!Impedance Control

What is Impedance?

Impedance in PCB design is the combined effect of **resistance**, **inductance**, and **capacitance** that electrical signals encounter as they travel through circuit board traces. Think of it as the "resistance" to AC signals.

Why Impedance Matters

In modern high-speed digital circuits, controlling impedance is crucial because:

- **Signal Integrity**: Prevents signal degradation
- **Power Transfer**: Maximizes energy efficiency
- **EMI Reduction**: Minimizes electromagnetic interference
- **Reliability**: Ensures consistent circuit performance

When Do You Need Impedance Control?

You need controlled impedance when:

- **High-speed signals** (>50 MHz clock rates)
- **Differential pairs** (USB, HDMI, Ethernet, PCIe)
- **RF circuits** (WiFi, Bluetooth, cellular)
- **Long traces** (>1/10 of signal wavelength)
- **Clock signals** and fast edge rates

Key Concepts

1. Characteristic Impedance

The characteristic impedance (Z_0) of a trace depends on:

- **Trace width** (W)
- **Trace thickness** (T)
- **Dielectric constant** (εr) of the material
- **Height above reference plane** (H)
- **Solder mask** thickness

2. Common Impedance Values

Application	Impedance	Tolerance
Single-ended (general)	50Ω	±10%
Single-ended (memory)	40Ω	±10%
USB	90Ω (differential)	±10%
HDMI	100Ω (differential)	±10%
PCIe	85Ω (differential)	±10%
Ethernet	100Ω (differential)	±10%
RF (50 ohm systems)	50Ω	±5%

3. Microstrip vs. Stripline

Microstrip (external layer):
- Trace on outer layer

- Reference plane below
- Easier to fabricate
- Higher radiation
- Simpler calculation

Stripline (internal layer):
- Trace between reference planes
- Better shielding
- Lower radiation
- More complex fabrication
- Tighter control

Calculating Impedance

Microstrip Formula (simplified):

$Z_0 \approx (87 / \sqrt{(\varepsilon r + 1.41)}) \times \ln(5.98H / (0.8W + T))$

Where:
- Z_0 = Characteristic impedance (ohms)
- εr = Dielectric constant
- H = Height to reference plane (mils)
- W = Trace width (mils)
- T = Trace thickness (mils)

Online Calculators

For accurate calculations, use:
- Saturn PCB Design Toolkit (free)
- EEWeb Impedance Calculator
- Manufacturer's calculators (recommended)

Design Guidelines

Step 1: Determine Requirements

- **Identify signals** needing impedance control

- **Determine impedance value** for each signal

- **Define tolerance** requirements

- **Check manufacturer capabilities**

Step 2: Choose Stack-up

Recommendations:
- Use **symmetrical stack-ups**
- Place controlled impedance on **consistent layers**
- Maintain **constant dielectric thickness**
- Use **reference planes** properly

Step 3: Calculate Trace Geometry

Process:

- Select dielectric material (FR-4 typical: εr ≈ 4.0-4.8)

- Determine layer height

- Calculate required trace width

- Adjust for manufacturing tolerances

Example (50Ω Microstrip on FR-4):
- Dielectric: FR-4 (εr = 4.2)
- Height to plane: 8 mils
- Trace thickness: 1.4 mil (1 oz copper)
- Calculated width: ~15 mils

Step 4: Route the Traces

Best Practices:
- Avoid 90° bends: Use 45° or curved
- Minimize vias: Each via changes impedance
- Keep traces short: Shorter = better signal integrity
- Maintain spacing: Avoid coupling between traces
- Use reference planes: Continuous ground/power

Step 5: Document for Manufacturer

Required Information:

- Impedance value and tolerance
- Layer number and reference plane
- Target trace width
- Material specifications
- Testing requirements

Common Mistakes to Avoid

❌ Mistake 1: Ignoring Via Effects

Problem: Vias create impedance discontinuities
Solution: Use via stitching or minimize via count

❌ Mistake 2: Inconsistent Reference Planes

Problem: Gaps in planes cause impedance changes
Solution: Ensure solid, continuous planes

❌ Mistake 3: Wrong Dielectric Constant

Problem: εr varies with frequency
Solution: Use manufacturer's data for specific frequency

❌ Mistake 4: Neglecting Solder Mask

Problem: Solder mask affects impedance
Solution: Account for it in calculations (≈3-5% change)

❌ Mistake 5: Not Communicating with Manufacturer

Problem: Design doesn't match capabilities
Solution: Consult early and provide detailed specs

Testing and Verification

1. TDR Testing

Time Domain Reflectometry:
- Sends fast pulse down trace
- Measures reflections
- Calculates actual impedance
- Provides impedance profile

2. Impedance Test Coupons

Design:
- Include test structures on panel
- Same layer stack-up
- Same trace geometry
- Measured during fabrication

3. Polar CITS System

Controlled Impedance Test System:
- Standard industry test method
- Non-destructive
- High accuracy
- Provides test report

Practical Example: USB Differential Pair

Requirements:

- 90Ω differential impedance
- ±10% tolerance
- FR-4 material
- Microstrip routing

Design Process:

- Calculate single-ended impedance:

  - For 90Ω differential, target ~50Ω single-ended

- Determine trace parameters:

  - Using calculator with FR-4 (εr=4.2, H=8mils, T=1.4mils)

- Result: W ≈ 12 mils for 50Ω

- **Calculate differential spacing**:

  - For 90Ω differential: S ≈ 8 mils (edge-to-edge)

- **Route the pair**:

  - Maintain constant spacing
  - Match trace lengths
  - Avoid 90° bends
  - Keep both traces on same layer

- **Verify**:

  - Use TDR to measure
  - Check 90Ω ±9Ω (81-99Ω range)
  - Document results

Cost Implications

Controlled impedance affects cost:

Factor	Impact
Material	Standard FR-4 = minimal impact
Layer Stack-up	May require specific thickness
Testing	Adds test coupon cost
Tolerance	Tighter tolerance = higher cost
Volume	Higher volume spreads tooling cost

Cost-saving tips:
- Use standard materials when possible
- Group impedance values (use 50Ω for most)
- Use manufacturer's preferred stack-ups
- Plan ahead to avoid changes

Future Considerations

Higher Frequencies

- **5G and above**: More critical impedance control
- **Signal loss**: Becomes major concern
- **Material selection**: Low-loss materials needed

Advanced Techniques

- **Dietorted routing**: Non-parallel traces for isolation
- **Length matching**: For timing-critical signals
- **Via optimization**: Back-drilling, via-in-pad

Simulation Tools

- **Field solvers**: For complex geometries
- **SI simulation**: Signal integrity analysis
- **EM simulation**: Electromagnetic modeling

Getting Started Checklist

For beginners starting with impedance control:

- [ ] **Learn basics**: Understand transmission lines
- [ ] **Identify needs**: Which signals need control?
- [ ] **Choose tools**: Impedance calculator software
- [ ] **Consult manufacturer**: Get design guidelines
- [ ] **Design stack-up**: Plan layer structure
- [ ] **Calculate geometry**: Determine trace widths
- [ ] **Route carefully**: Follow best practices
- [ ] **Include test coupons**: For verification

- [ ] **Document clearly**: Communicate with fabricator
- [ ] **Test results**: Verify impedance values

Conclusion

Impedance control is essential for modern PCB design, especially as speeds increase and devices become more compact. While it adds complexity, understanding the fundamentals and following systematic design practices will ensure reliable, high-performance circuits.

Key Takeaways:

- **Plan ahead**: Design impedance control from the start
- **Use calculators**: Don't guess at trace geometry
- **Work with manufacturers**: They're your partners
- **Test and verify**: Ensure your design works as intended
- **Keep learning**: Technology continues to evolve

Sources:
- Beginner's Guide to Impedance Control
- Impedance Control Technical Guide
- Controlled Impedance PCB Guide