Striplines and Microstrips
(PCB Transmission Lines)
Disclaimer:

This presentation is merely a compilation of information from public sources. It does not constitute engineering “advice” or “consulting”. Eatman Associates assumes no responsibility related to the use of the information presented here.

Models shown in this presentation assume certain conditions which may not be present.
What is a Transmission Line?

- A transmission line is *a specialized cable or other structure designed to transfer electrical power* - preferably with minimal losses
- Maximum power transfer occurs when the *characteristic impedances* of the source, line, and load are all matched
Characteristic Impedance $Z_0$:

- A transmission line that is terminated at one end with a resistor equal to the characteristic impedance appears to the source like an infinitely long transmission line. That is to say that, **properly terminated, the end of a uniform transmission line produces no reflections**

- The *characteristic impedance* (or surge impedance) of a uniform transmission line, usually written $Z_0$, is the ratio of the amplitudes of voltage and current of a single wave propagating along the line assuming no reflections (scalar, not vector)
History of Transmission Lines:

• Initial research was for the telegraph industry
  - Samuel Morse invented the telegraph in 1837
  - transmission line theory was needed
• In 1858, the first transatlantic telegraph cable was laid – it lasted three weeks
• Wildman Whitehouse, chief electrician for the Atlantic Telegraph Company, blew the cable up by applying too much signal voltage (2000 volts!) in an attempt to overcome signal losses
• 1880 – Oliver Heaviside patented coaxial cable
One Type of PCB Transmission Line – the Microstrip:

Looks pretty familiar!

\[ t_{pd} \text{ (ps/in)} \sim 85(\sqrt{0.475 \varepsilon_r + 0.67}) \neq c ! \]
Microstrip $Dk = 4$ - Special Rule of Thumb:

Use transmission line theory and matched load impedance when the microstrip trace length exceeds 2 in./ns of rise/fall time

- Applies only for microstrip lines when $\varepsilon_r = 4$
- Use rise or fall time, whichever is faster

- At 1 GHz with 0.35 ns rise time, the critical line length would be 0.70 inches
Transmission Line Model: Telegrapher’s Equations

- Developed by Heaviside circa 1880
- Can be derived from Maxwell’s equations or loop and node equations of the lumped model

\[
\frac{\partial}{\partial x} V(x, t) = -L \frac{\partial}{\partial t} I(x, t) - RI(x, t)
\]
\[
\frac{\partial}{\partial x} I(x, t) = -C \frac{\partial}{\partial t} V(x, t) - GV(x, t)
\]
Effects of Capacitance, Inductance, and Resistance:

- Capacitors store energy in an electric field
- Inductors store energy in a magnetic field
- Resistors convert electrical energy into heat
  - results in loss of signal amplitude
Capacitance, Inductance, and Resistance:

• Capacitance and inductance affect varying signals (AC and RF) - includes on-off transitions in DC applications

• Discontinuities in distributed capacitance and inductance along a transmission line cause signal reflections - like an optical mirror does - *PC board vias are discontinuities*
How Discontinuities Cause Reflections:

Boundary conditions at the load: no current
PC Board Traces as Transmission Lines:

- Board traces are conductors
- The board material itself (FR-4, etc.) is a dielectric
- Impedance discontinuities ➤ reflected waves
- At DC frequency (zero Hz), the return signal follows the path of least resistance
- At RF frequencies, the return signal follows the path of least impedance
Characteristic Impedance $Z_0$:

- In the ideal case of a “lossless” transmission line (line resistance and conductance = 0)

$$Z_0 = \sqrt{\frac{L}{C}}$$

lossless line

1. **Capacitance goes up, $Z_0$ goes down**
   - higher $\varepsilon_r$ and thinner dielectric = higher capacitance

2. **Inductance goes up, $Z_0$ goes up**
   - wider signal traces have lower inductance
Typical PCB Trace Impedance Discontinuities:

- Bends
- T
- Cross
- Width variation
- Proximity coupling
Propagation Delay:

• Speed of the signal through a PCB transmission line depends upon the board material

• Higher dielectric constant (\(\varepsilon_r\)) = lower speed

• *Propagation delay is important when estimating clock skew on PC boards*

• Formula for signal velocity on a stripline:

\[
V = \frac{3 \times 10^8 \text{ m/s}}{\sqrt{\varepsilon_r}}
\]
Types of PC Board Transmission Lines
Surface Microstrip:

Estimated $Z_0$:

$$Z_0(\Omega) = \frac{87}{\sqrt{\varepsilon_r + 1.41}} \ln\left[\frac{5.98H}{(0.8W + T)}\right]$$

Signal velocity $\sim$

$$c/\sqrt{0.475\varepsilon_r + 0.67}$$
Surface Microstrip:

Advantages:
• Air dielectric on top
  - lower combined dielectric constant
  - higher speed
• Top conductor can be probed during prototyping
• Cheaper to implement – fewer PC board layers

Disadvantages:
• Radiates EMI off the top of the board
• Is susceptible to EMI
Surface Microstrip Variants:

Embedded Microstrip
Symmetric Stripline:

Estimated $Z_0$:

$$Z_0(\Omega) = \frac{60}{\sqrt{\varepsilon_r}} \ln \left[ \frac{1.9(B)}{(0.8W + T)} \right].$$

Signal velocity ~ $c/\sqrt{\varepsilon_r}$
**Stripline:**

**Advantages:**
- Signal traces shielded – less EMI, less crosstalk

**Disadvantages:**
- Difficult to troubleshoot – no top board access
- More costly (more layers)
- For flex circuits, less flexible (thicker)
- Slower speed
Stripline variants:

Edge-coupled Stripline

Asymmetric Stripline
Other Surface Topologies:

- Coplanar Waveguide (CPW)
- Edge-coupled CPW with Ground (CPWG)
Which One is Best?

- Lower board cost: Surface topologies (microstrip and CPW use fewer layers)
- Ease of prototyping: Surface topologies easier to troubleshoot
- Low loss: CPW
- Best EMI performance: Stripline
- Lowest impedance: Stripline
- Metal shields often used to reduce EMI from surface traces and components
Which One is Best?

- Adhesiveless flex circuit cores have better thickness control
- Teflon® composite flex material can have $\varepsilon_r$ as low as 2.3 and loss tangent of .0015
- Low Dk rigid board laminates are also available
  - suitable for very high speeds
- Soldermasks have dielectric properties and loss tangents similar to board materials
  - affects performance of microstrip and CPW
Reducing Unwanted Coupling (Crosstalk)
Reducing Unwanted Coupling (Crosstalk):

- Crosstalk is caused by undesired capacitive, inductive, or conductive coupling from one circuit, part of a circuit, or channel, to another. Coupling affects impedance.

- A transmission line can be a EMI source and/or victim

- Reducing unwanted coupling requires *reorientation, distance, or shielding*

- Effect of coupling decreases with the *square of the distance* between the conductors
Minimizing Crosstalk:

• Use striplines or embedded microstrips vs microstrips

• Minimize congestion of traces through component placement

• Move signal lines as close to the ground plane as possible - while maintaining the desired impedance

• Minimize parallel run lengths between signals

• Route signals on different layers orthogonal to each other
Minimizing Crosstalk:

• For single-ended line:
  * Keep adjacent traces as far apart as possible
  * at least 2 trace widths

• For a differential pair: crosstalk cancels out
  * Route differential pair traces (for one signal) as close together as possible

• Use differential pairs with equal length lines
  * Nested bends = unequal length traces
Transmission Line Losses
Components of Attenuation / Signal Loss:

- Metal (resistive) losses
  - “skin effect” loss proportional to $\sqrt{\text{frequency}}$
  - also affected by metal surface roughness (“tooth”)

- Dielectric loss
  - proportional to frequency – see “loss tangent”

- Dielectric conductivity loss
  - stable over frequency
  - may be ignored if resistivity $> 10,000$ Ohm-cm

- Radiation loss
Bad Design ➤ Bad Performance

• Choose the right PC board materials:
  - choose material properties @ frequency of use
  - choose metal for resistance @ frequency

• Use good general PC board layout practices:
  - don’t route separate signals in parallel
  - eliminate sharp corners on traces
  - avoid ground plane slots
  - use clean return paths directly under signal lines
  - pick the right stackup
  - etc…
PC Board Manufacturing Issues Affecting Impedance:

• Interlayer voids = Δ capacitance
• Pre-preg thickness variation = Δ capacitance
• Poor via-to-pad registration - drill deflection
• Inconsistent etching ➔ varying trace widths
• RA copper has less “tooth” than ED copper

• Vendor and processes affect your outcome
Transmission line design goals:

Perfect transfer of power (signal) from one end to the other (source to load)

- No signal reflections down the line (no discontinuities)
- No radiated or induced EMI (no coupling or crosstalk)
- No amplitude reduction (no resistive or dielectric losses)

All this in a double sided board with a tight layout, dirt cheap materials, and a sketchy board fabricator

NOT EXACTLY
The End