CEG4480_07
PCB

Reference: Chapter 5 Ground Planes and Layer Stacking of *High speed digital design* , by Johnson and Graham

Introduction

- What is a PCB (Printed Circuit Board) (PWB: Printed Wire Board)?
- Why we need one?
  - For large scale production/repeatable fabrication
  - Reliable: much better than ad hoc bread board
  - Controlled Electrical characteristics
- Many videos showing you how to make one on You Tube
  - E.g. http://www.youtube.com/watch?v=e-gMsABCRTI
- Our lecture:
  - Not on how to make one (you will try one later)
  - More concern about issues on reliability; electrical characteristics

Functions of Ground and Power Planes

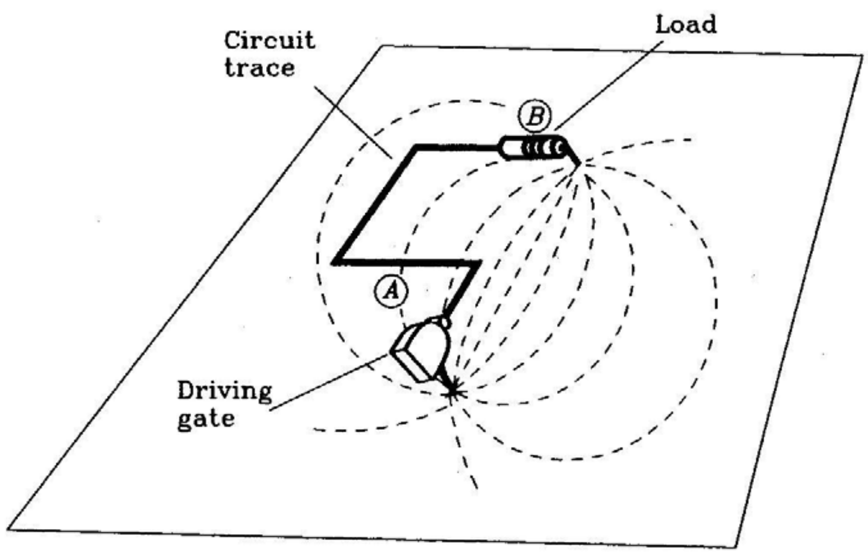
- Provide stable reference voltage for digital signal exchange
- Distribute power
- Control crosstalk between signals

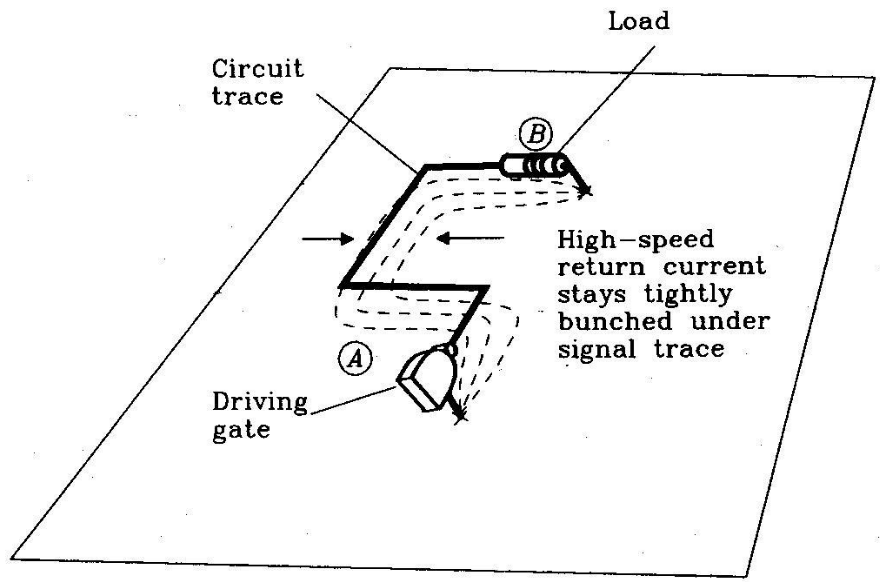
- Note: all formula are approximations

  In this book, signal trace = tracks on PCB

High-speed I follows paths of least inductance

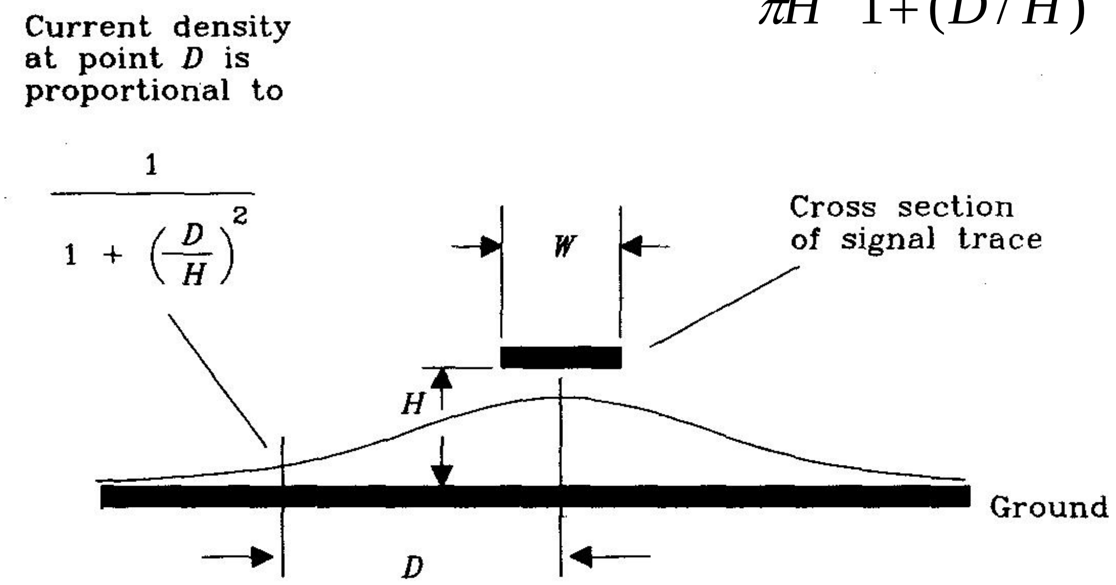
- At low speed, current follows the least "resistance" path
  - Current density corresponds to the conductance of each path
- At high speed, current follows the least "inductance" path
  - Lowest inductance path is directly underneath signal track due to minimum loop area





- Distribution of return-current density is a function of
  - H = height of trace above PCB
  - D = perpendicular distance
- Current density at D is (Io is the total signal current)

$$i(D) = \frac{I_o}{\pi H} \cdot \frac{1}{1+(D/H)^2}$$

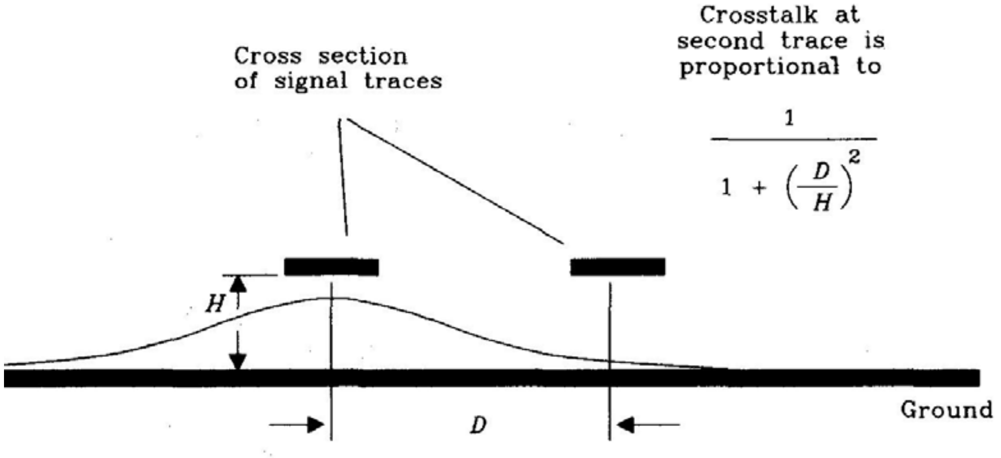


Distribution of high-frequency current density underneath a signal trace.

Crosstalk in Solid Ground Planes

- Returning signal currents generate magnetic fields, which in turn induce voltages in other circuit traces
- Crosstalk depends on mutual inductance & mutual capacitance
  - Usually, inductance effect dominates
- Induced voltages are proportional to the derivates of the driving voltage
  - Faster rise time will make it worse.
  - Can be expressed as a ratio of measured noise;
  - Constant K depends on the rise time and length of trace.
- The induced noise coupled into adjacent traces falls off with the square of increasing distance.

$$Crosstalk \approx \frac{K}{1+(D/H)^2}$$



Cross section of two traces showing crosstalk.

Simple Crosstalk Experiment

- 26” Cu track separated by 0.08” centre to centre
- Ground plane is a solid Cu sheet
- Sandwich between dielectric spacers of known thickness
  - Height above ground plane is varied.

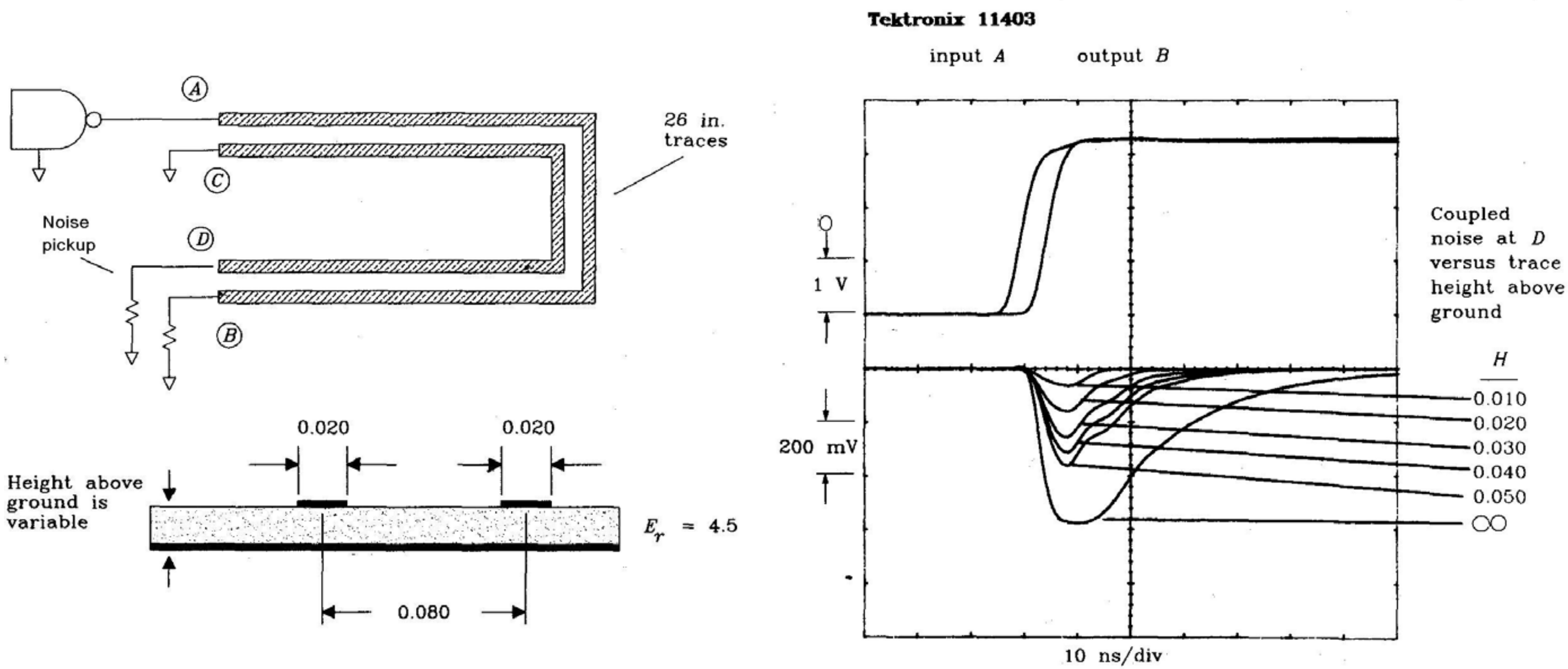
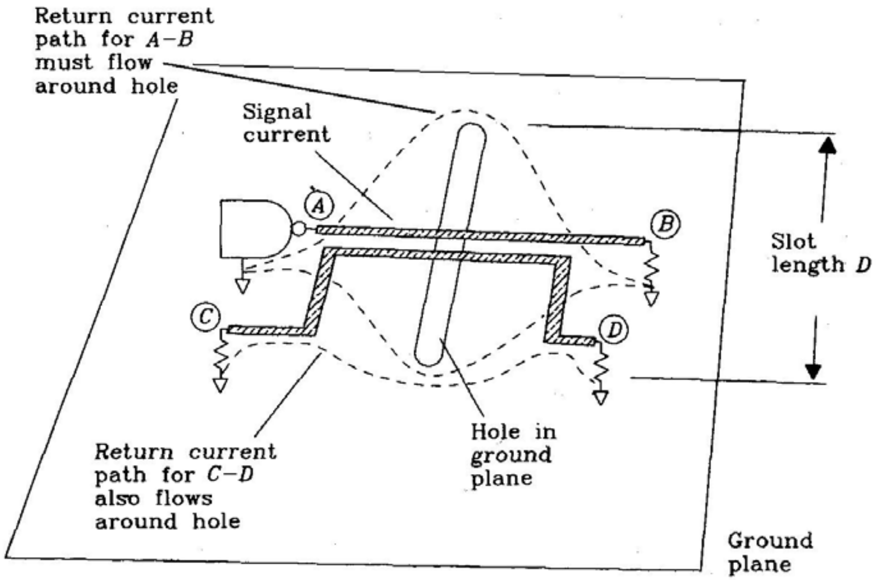


Figure 5.6 Step response of a mutual coupling experiment.

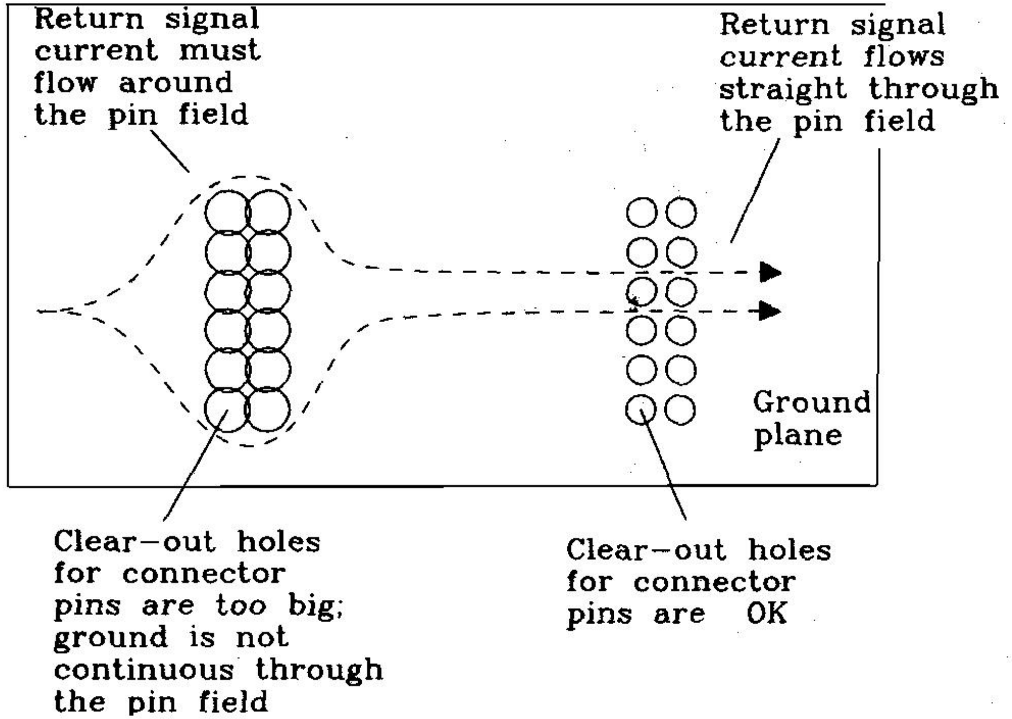
Crosstalk in Slotted ground planes

- Sometimes signal tracks are placed on the ground plane due to space. This is done by cutting a long slot in the ground plane.
- Ground slots add inductance to traces passing perpendicular over the slots, and increase crosstalk.
  - Diverted current makes a large loop which increases the inductance and slows the rise time of the signal path.
  - Diverted current also overlaps with other signal paths and increases mutual inductance.

- All slot widths, no matter how thin, will have the same effect.
- Must not tolerate this practice.



Crosstalk in a slotted ground plane.

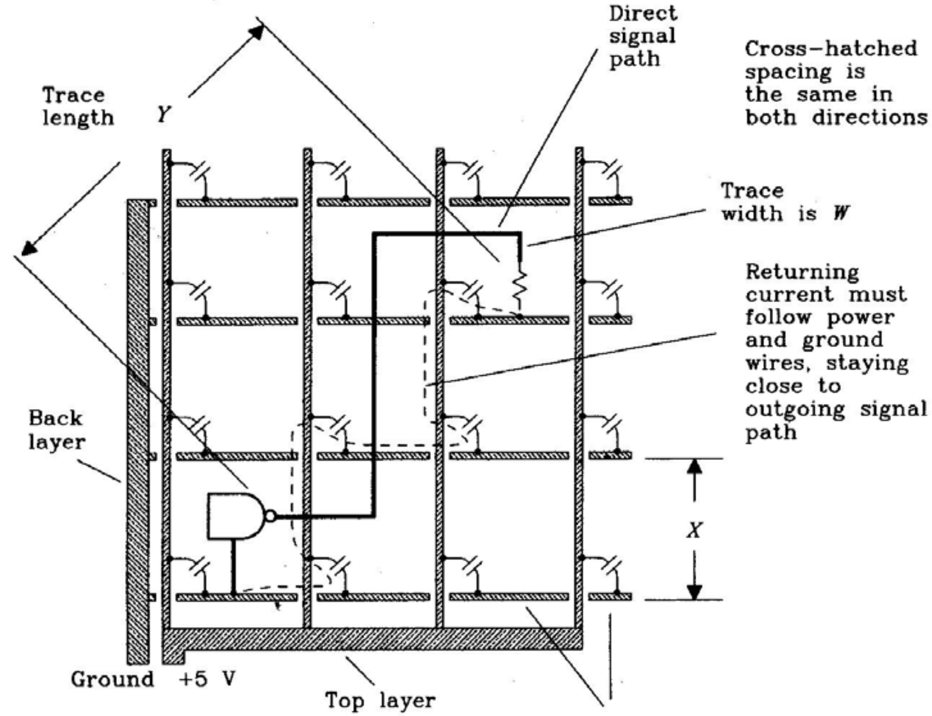


Ground slot caused by improper connector layout.

- Ground slots also happen on dense backplanes with many connection holes (vias).
- Remember
  - Slots in ground plane creates unwanted inductance
  - Slots inductances slows down rising edges
  - Slot inductance creates mutual inductive crosstalk

Crosstalk in Cross-Hatched Ground Plane

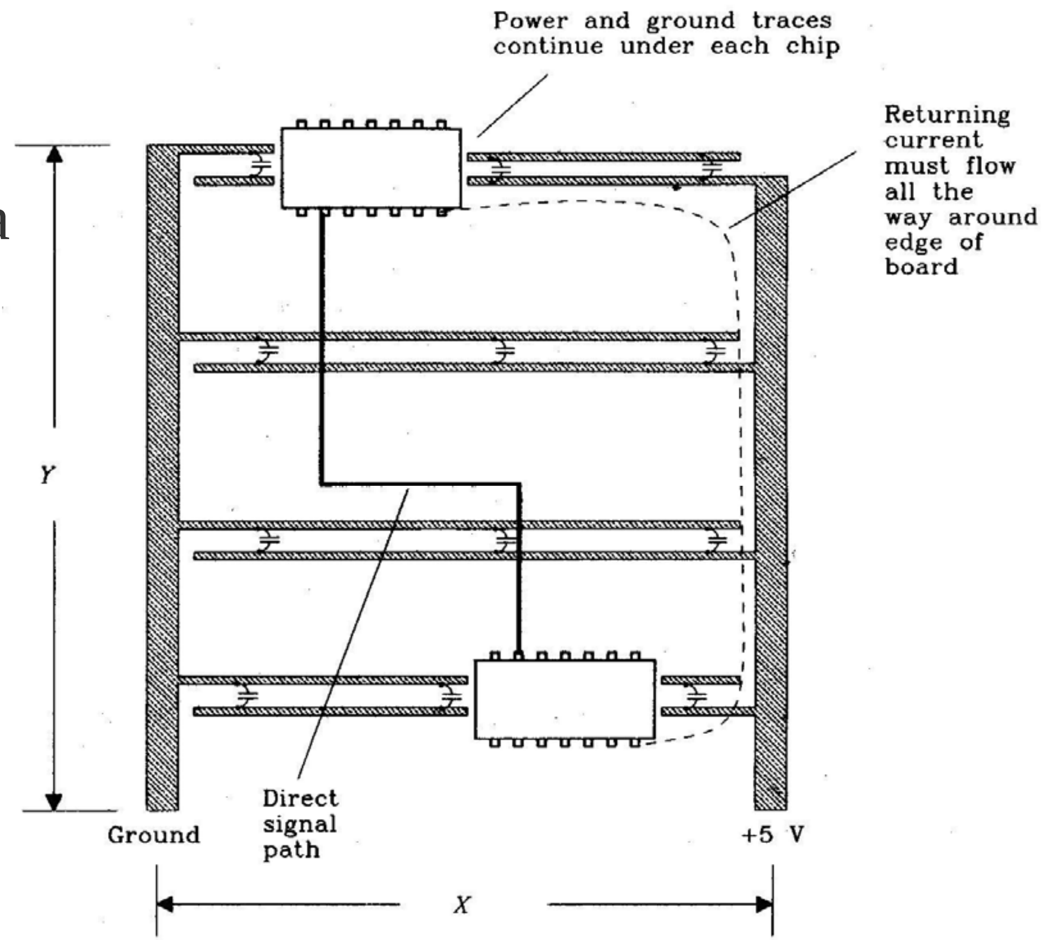
- 2-layer board design: Cross-hatched power & ground grid saves separate power & ground planes; but at the expense of increased mutual inductance.
  - Not good enough for high speed system
- Need good bypass capacitors because signals traverse several capacitors to return to the driving gate



Power and ground grid on two layers.

Crosstalk with Power and Ground Fingers

- One power + ground layer; 1 signal layer
- Saves even more board area
- Worse mutual inductance because most signal must go round the edge of the board
  - For very slow circuit only
  - Fatten the ground fingers will not help



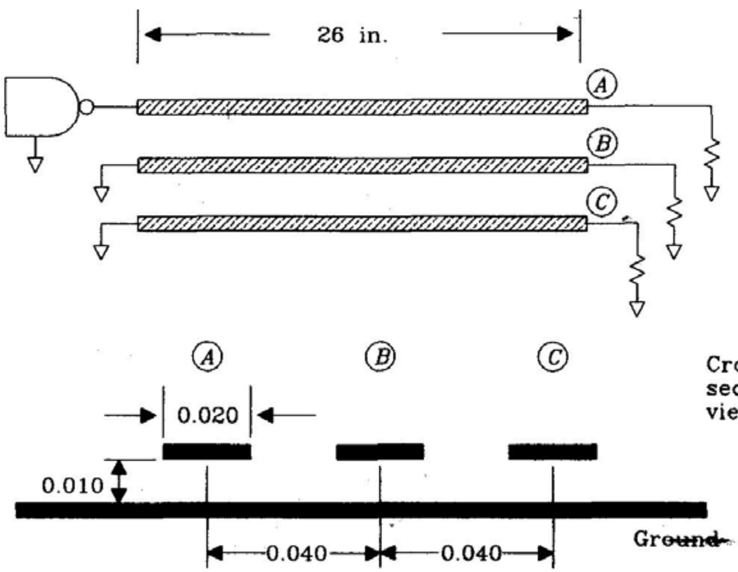
Ground fingers layout.

Guard Traces

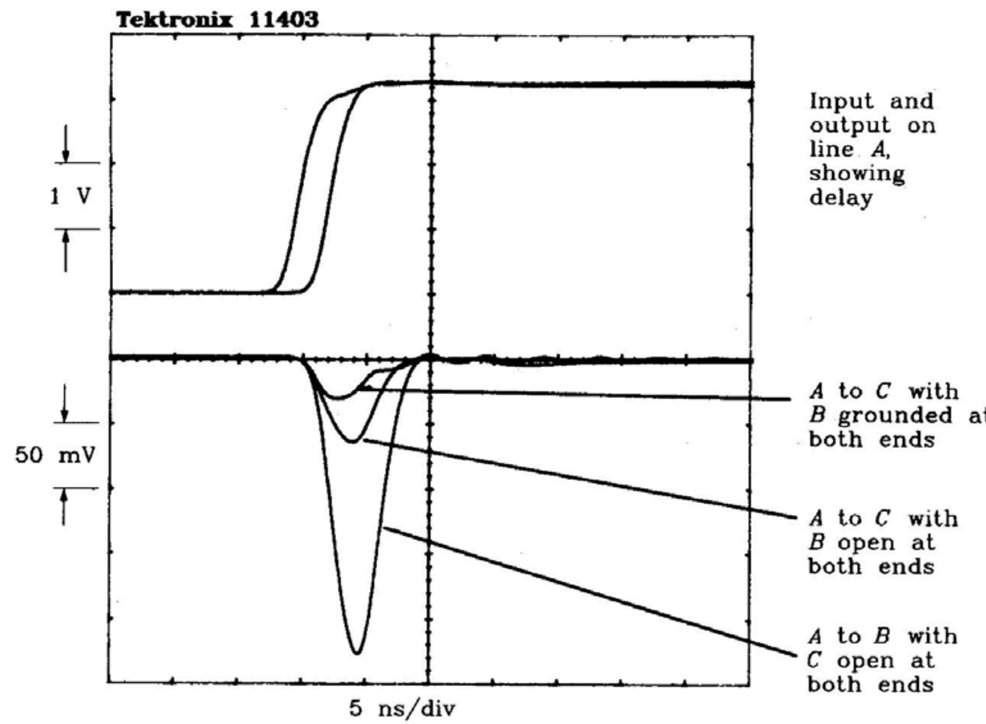
- Appear extensively in Analogue Systems
  - On a two layer board for audio frequencies without a solid ground plane
  - A pair of grounded traces running parallel to a sensitive input can reduce crosstalk by an order of magnitude.
- In digital system
  - Solid ground plane is preferred
  - No extra benefit using guard trace
- In general, a trace grounded at both ends separating two signal traces will half the coupling
- In homogeneous digital system, solid ground plane can reduce the crosstalk to 1-3% which is good enough. But need to be more careful with analogue system which mixes high power (output) signals with low power (input) signals.

Experiment measuring Guard Trace Efficiencies

- Traces are 26" long with a characteristic impedance of 50 ohms



Guard trace demonstration.

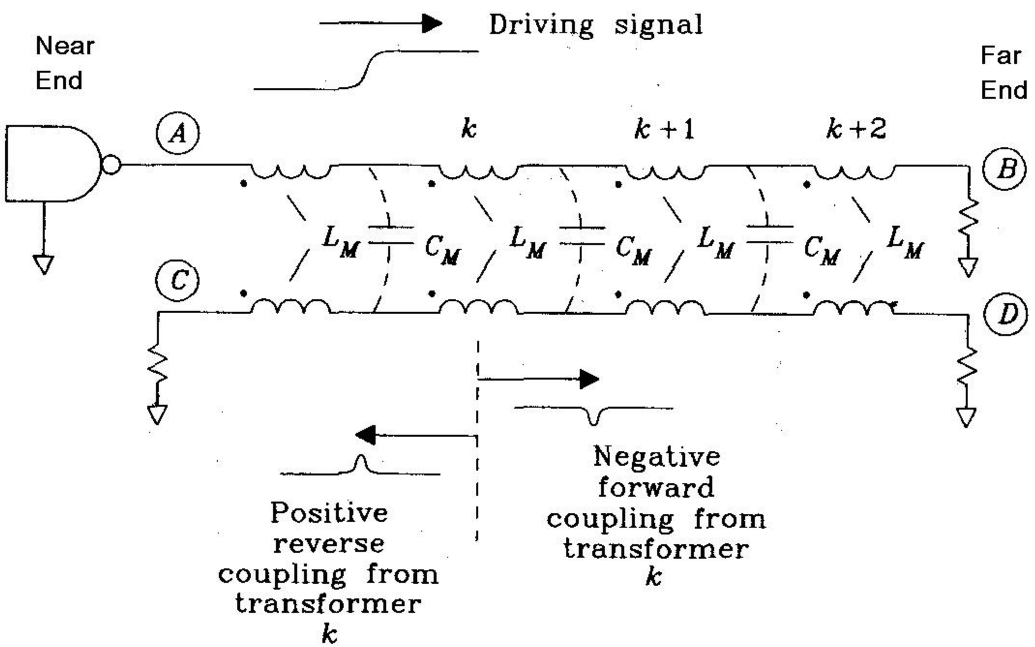


Example showing guard trace effect on coupling.

Near-end and Far-end crosstalk

- Descriptions so far based on lumped-circuit analysis
  - No good for long transmission lines
- **Inductive Coupling Mechanism (refer to diagram)**
  - Wire A-B carries a signal whose magnetic field induce voltages in wire C-D
  - Magnetic coupling (mutual inductance) works like a transformer
  - As signal steps propagate down the signal line, a series of blips appear on the adjoining line
  - Blips propagates both forward and backward along line C-D

$$L_M = L\frac{dI}{dT}$$



Mutual coupling between two long transmission lines.

- Polarity at each end of transformer K is different
  - Positive blips goes to the left (← +ve)
  - Negative blips goes to the right (-ve →)
- Reflection diagram shows the blips from all transformers adding in a curious pattern.
  - All forward (negative) blips arrive at the far end at the same time
    - Total forward blips is proportional to the total mutual inductance
    - Longer line will have bigger mutual inductance.
  - Reverse (positive) blips will arrive at the source (near end) at different times.
    - Total mutual inductance is the same as the forward case but it spreads out over a period of 2Tp
    - In practice, all the reverse blips smooth into one continuous blob of reverse coupling
    - Longer line will only increase the duration but not the height.

- **Capacitance Coupling Mechanism**
  - Similar to distributed mutual inductance except the polarity of couplings

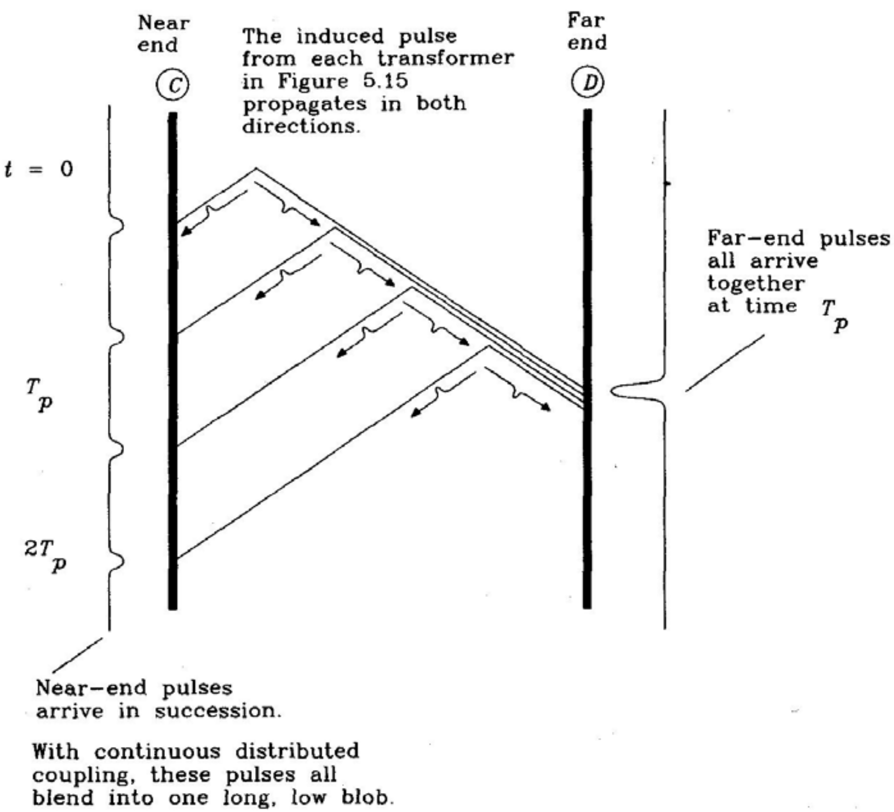
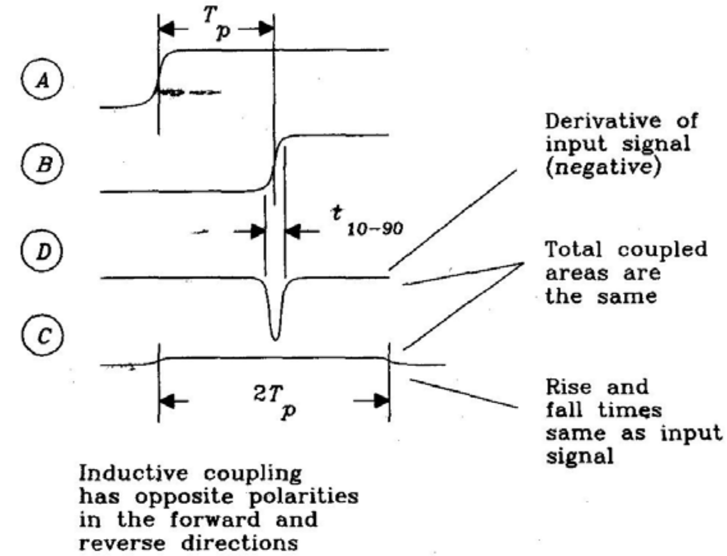
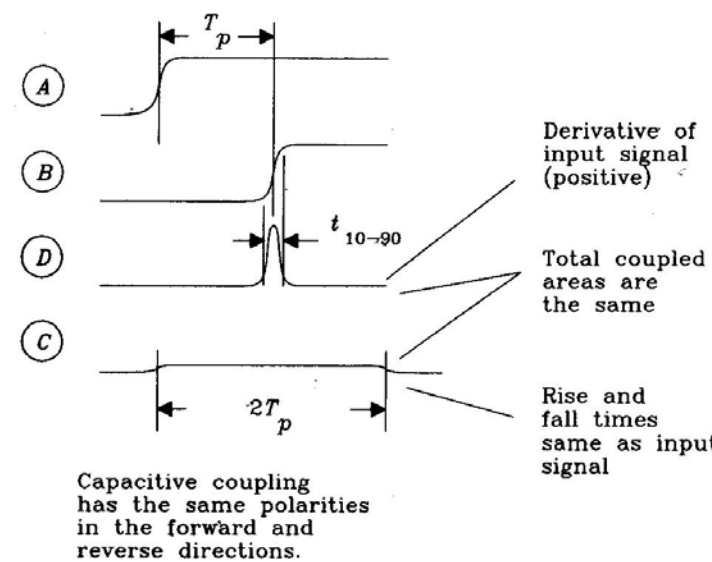


Figure 5.16 Reflection diagram showing mutual inductive coupling from the four transformers shown in Figure 5.15.



Forward and reverse mutual inductance coupling (distributed)



Forward and reverse mutual capacitance coupling (distributed)

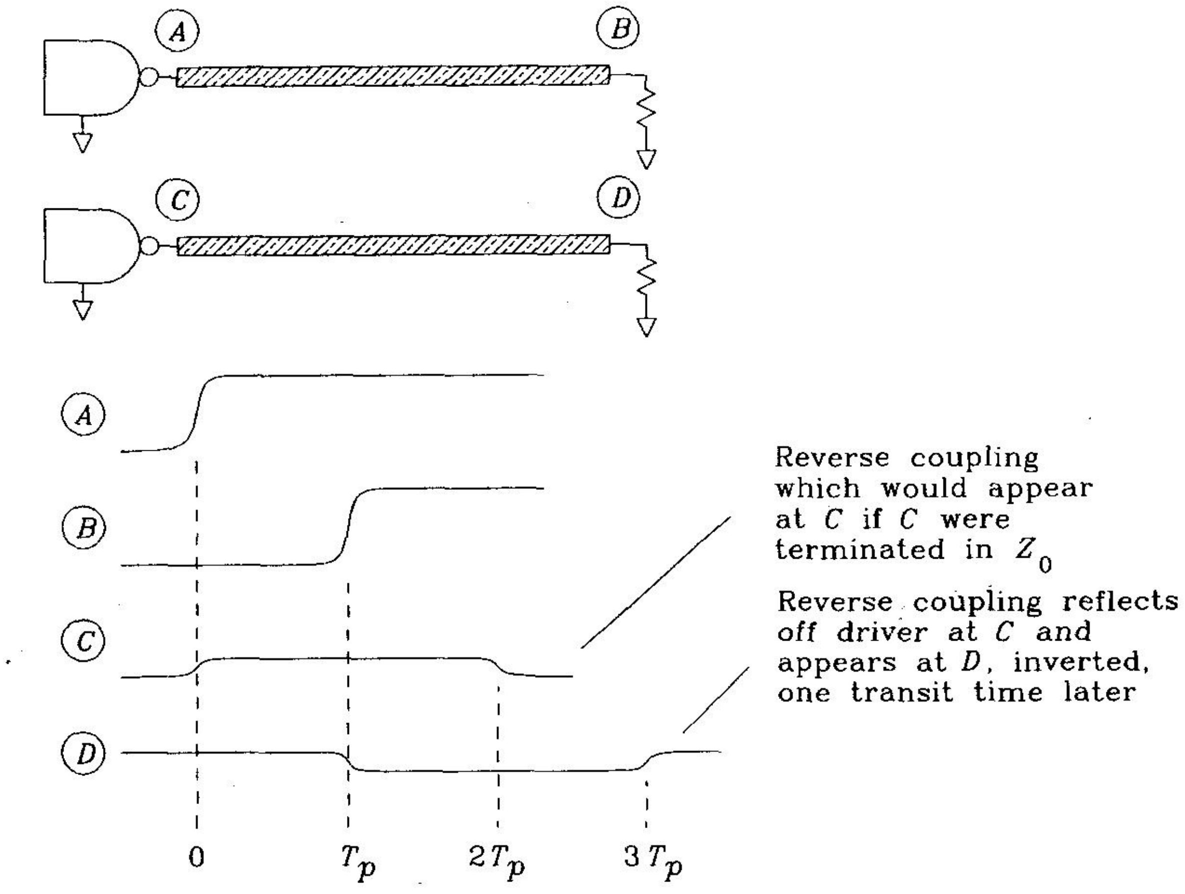
Combining Mutual Inductive & Mutual Capacitive Coupling

- Under normal conditions, over a solid ground plane, the inductive and capacitive crosstalk voltages are roughly equal
  - The forward crosstalk component cancel while the backward crosstalk components reinforce
- Over slotted, hatched or imperfect ground plane
  - Inductive component is much larger
  - Forward cross talk is large & negative

Near-end crosstalk can become a far-end problem

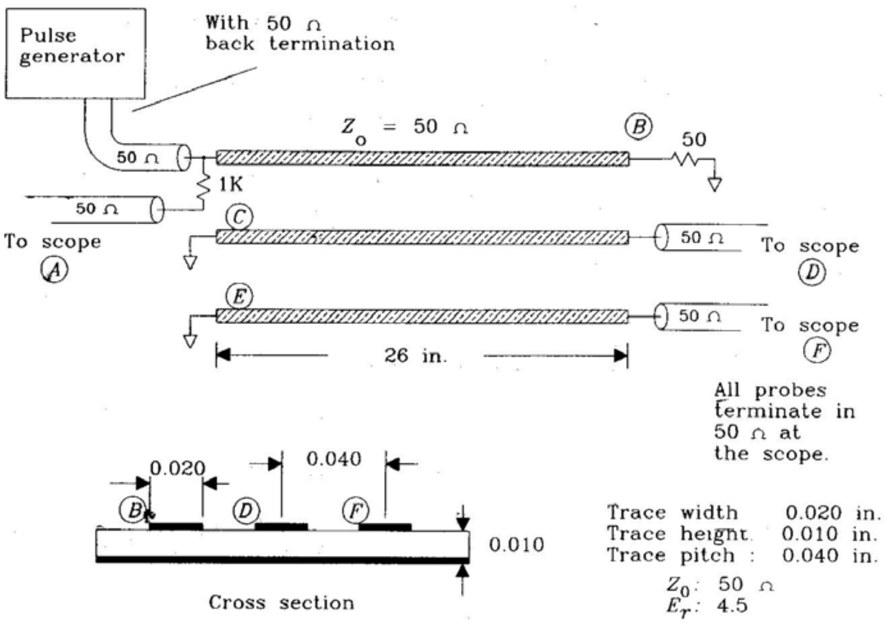
- In practical applications without source terminators
  - Source is a low impedance driver,
  - Reverse crosstalk reflects when it hits the near-end.
  - Reflection coefficient = -1
  - Signal seen at the far-end is a copy of the reverse coupling signal at C, delayed by one propagation delay and inverted

- Even though forward couplings due to mutual inductive and mutual capacitive effects cancel, the reflected signal from the backward component at the near-end will still causing problems.

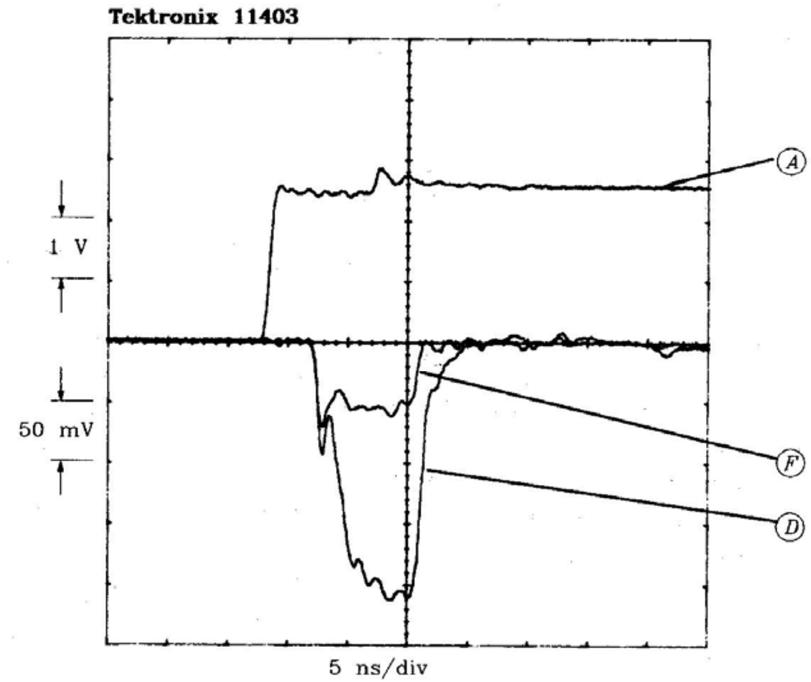


Reverse coupling reflecting from a low-impedance driver.

Reverse Crosstalk Experiment



Setup for reflected reverse crosstalk measurement.



Reflected reverse crosstalk measurement.

Summary (for long transmission line)

- Over solid ground, inductive and capacitive crosstalk are equal
  - Forward crosstalk cancel
  - Reverse crosstalk reinforce
- Over a slotted or imperfect ground plane, the inductive coupling exceeds capacitive coupling
  - Forward crosstalk large and negative
- Forward crosstalk is proportional to the derivative of the input signal and to line length
- Reverse coupling is like a square pulse, with a constant height and duration 2Tp
- Reverse crosstalk when it hits a low-impedance driver, reflects towards the far end.

Stacking Circuit Board Layers

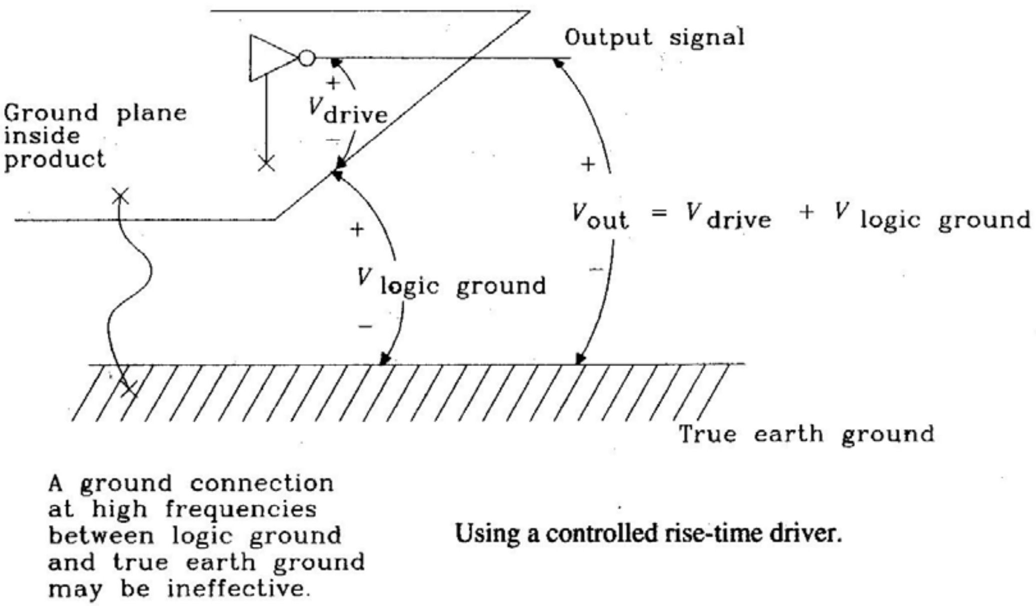
- Need to specify
  - Which are the power, ground and signal layers
  - Dielectric constant of the substrate
  - Spacing between layers
  - Desire trace dimensions and minimum trace spacing

Power & Ground Planning

- Choose solid, hatched or finger ground plane model
- Use ground & power planes in pair
  - Symmetric pairing in a layer stack helps prevent wrapping
- Both ground and power planes may be used as low-inductance signal return paths
- Adequate bypass capacitors between ground and power planes

Chassis Layer

- May want to run a signal outside your digital system
- Use a low-rise time or controlled rise-time driver
- Digital logic grounds are notorious for high-frequency noise especially when there are returning signals
- Without precaution, driver picks up ground noise and broadcasts it outside the chassis
- Add a chassis plane to the layer stack (next to a ground plane); the high capacitive coupling between the ground and chassis layer will act as effective short at high frequency
  - Screw (solder) one end of the chassis layer to the external chassis
  - Much more effective than a simple bypass capacitor
  - Digital logic and external chassis remains electrically isolated



Using a controlled rise-time driver.

Selecting Trace Dimensions

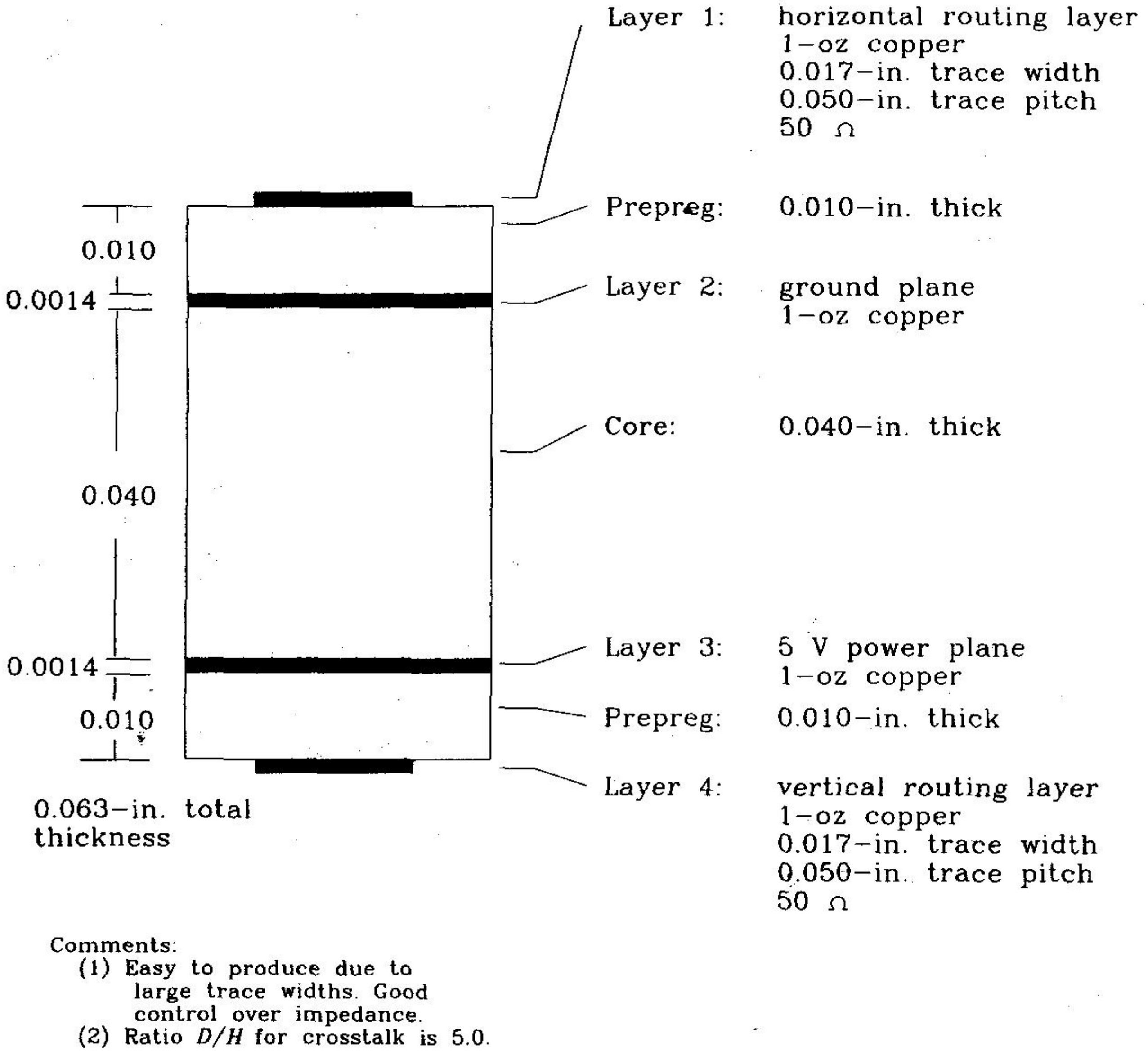
- Dense design requires fewer layers but
- Smaller, more closely spaced traces also yield more crosstalk and power-handling capacity problem
- Power-handling capacity depends on
  - Cross sectional area
  - Allowable temperature rise (amount of power dissipated)
- Power is not a problem for a large distribution bus; is a big problem for extremely small trace
- High density will lower yield, thus increase cost; avoid using minimum attainable line width
- Other factors:
  - Control etching process to avoid wide line width variations to control the impedance

Routing density versus number of layers

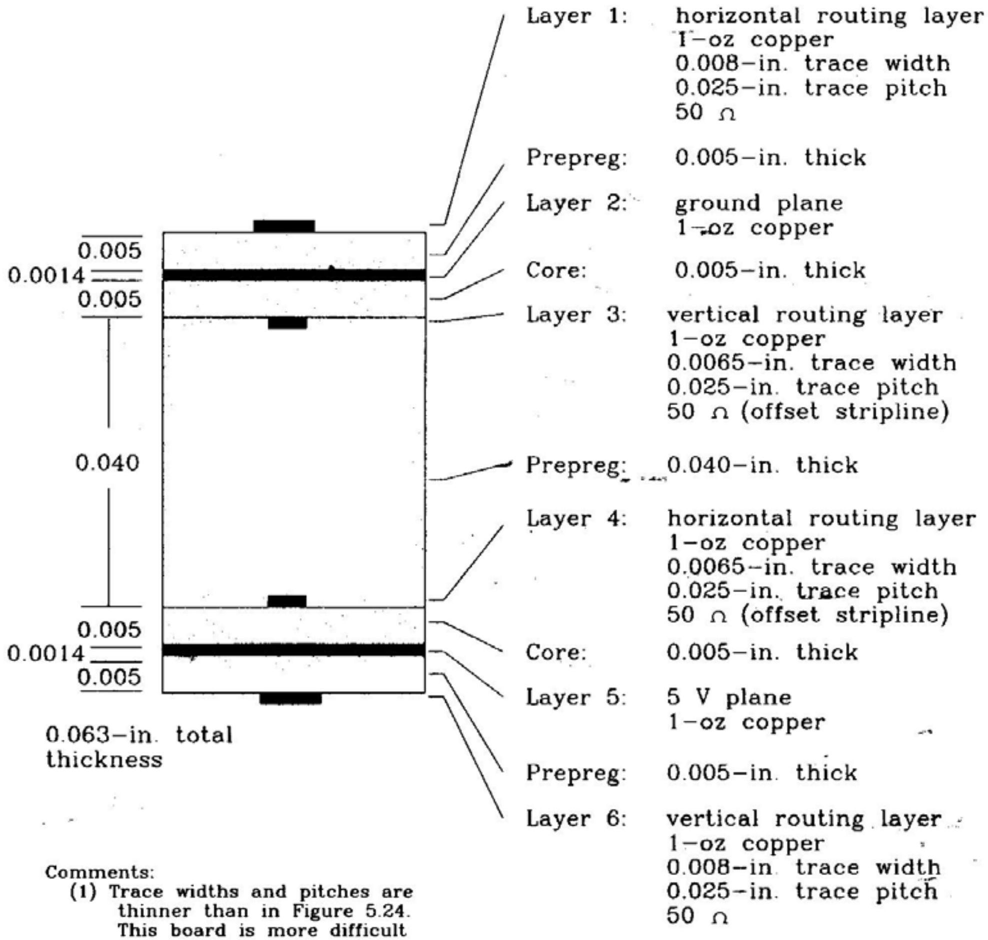
- More layer will cost more but easier to lay
- From experience
  - Divide the circuit into quadrants, half of the wires will stay with a quadrant
  - Same statistics when this quadrant is further subdivided into quadrants
  - Average wire length = spacing between quadrants

Classic Layer Stacks

- Core & prepreg refer to materials used in the substrate lamination process.
- Multilayer buildup process
  - Start with a set of raw two-sided laminated layers coated with copper on each side
  - Etch the inner layers with the wiring patterns (strip lines) (leave outer layer intact) ; Etched laminates are called **cores**.
  - Stack cores together, with a sheet of **prepreg epoxy** materials placed between each pair of cores. This sheet melts into an epoxy glue when heated and pressed.
    - Thickness of prepreg determines the spacing between core layers
    - Harden epoxy layer have same dielectric constant as the core layer
  - Holes are drilled through the assembly
  - Plating step covers the inside surfaces of the hole and the outer surface so that they are connected.
  - Etch away the unwanted copper on the outer layer
  - Board is then tinned, coated with solder mask, and silk screen on both sides

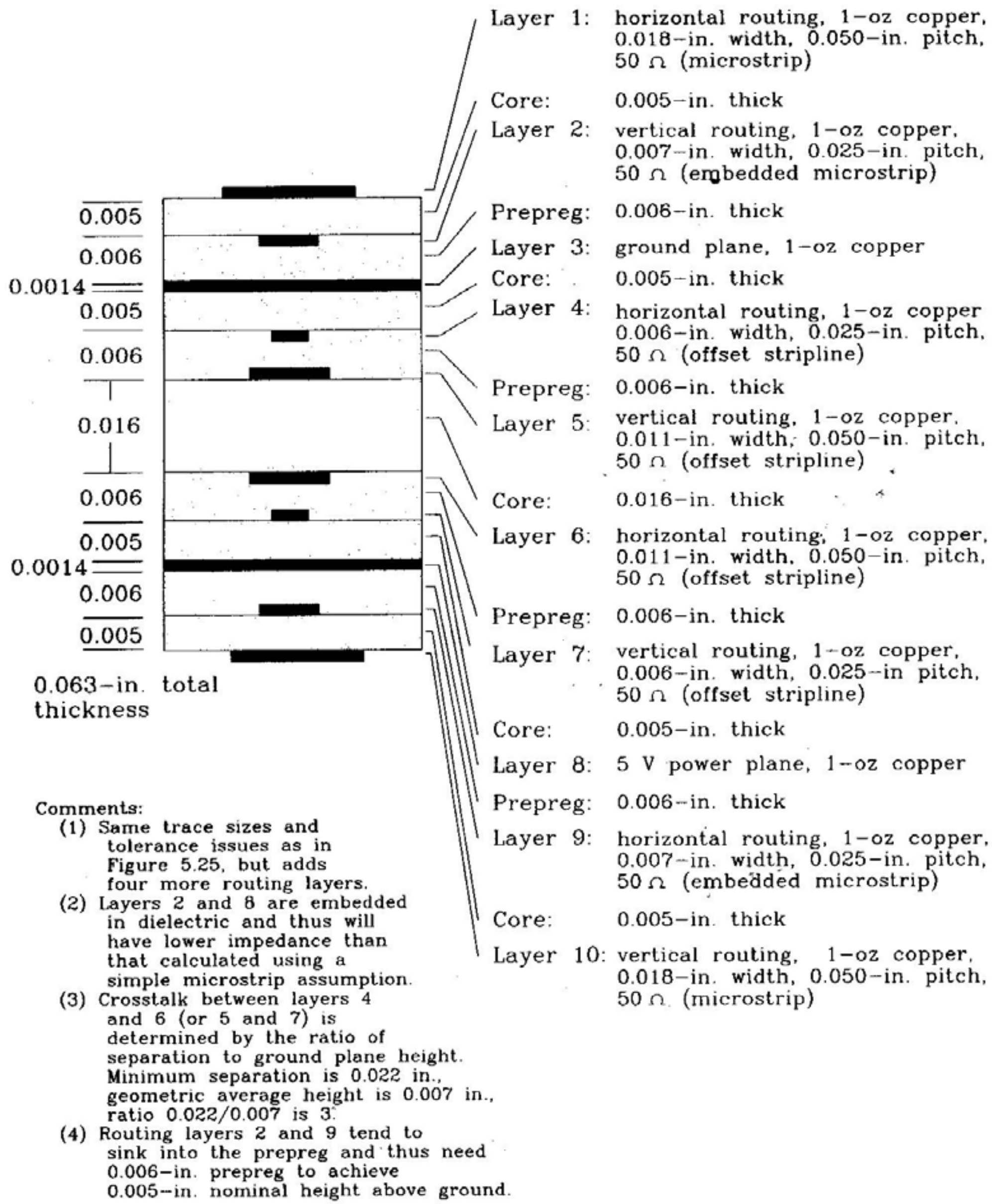


Four-layer stack.



Comments:
(1) Trace widths and pitches are thinner than in Figure 5.24. This board is more difficult to produce.
(2) Thin traces, combined with standard trace width tolerances make impedance more difficult to control.
(3) The trace pitch is twice as dense as in Figure 5.24, and this board has twice the number of routing layers. Overall, this board has four times the routing capacity in Figure 5.24.
(4) The D/H ratio for crosstalk is 5.0.

Six-layer stack.



Ten-layer stack.

Points to remember

- Core and prepreg layers alternate
- Outer layers, if plated, have greater trace width variation than inner traces
- Traces on routing layer tend to sink into the prepreg mixture. Their thickness doesn’t add to the total board thickness
- The thickness of the solid ground always adds to the total board thickness
- For high speed,
  - keep ground and power planes directly adjacent
  - Use extra ground planes, not power planes, to isolated routing layers.