University of San Carlos | Department of
COMPUTER ENGINEERING

CpE 2303L
CpE Drafting and Design

Electrical Design Considerations

Resistance

- The copper printed tracks on a PCB have finite resistance which introduces a voltage drop proportional to the current flowing in that particular conductor.

Conductor Dimensions

- In general, conductor width is determined by:
  - Component Packing
  - Minimum spacing between conductors and components
  - Geometrical constraints

Resistance

- The resistance of a conductor is considered as a metal section having a rectangular cross-section depends upon the specific resistivity of copper, which is 1.724 x $10^{-6}\,\Omega$ at 20˚ C.
- It would be useful to know the resistance of a 1 mm wide copper conductor per cm of length.

Resistance

- A standard copper foil of 35 um thickness (without any plating) may be assumed.

$$R = \frac{\rho L}{A}\,\Omega$$

ρ = resistivity (Ω-cm x 10^{-6})

L = conductor length (cm)

A = area of cross section of the conductor (cm^2)

Resistance

Converting units to cm

$$Thickness = 35\ um = 0.0035\ cm$$
$$width = 1\ mm = 0.1\ cm$$
$$L = 1\ cm$$

Calculating resistance:

$$\rho(copper) = 1.74x10^{-6}\ (at\ 20^{\circ}\ C)$$
$$A = 0.0035\ x\ 0.1 = 3.5x10^{-4}\ cm^2$$
$$R = \frac{1.724x10^{-6}\ x\ 1}{3.5x10^{-4}}$$
$$= 0.0049\ \Omega$$
$$\approx 0.0050\ \Omega \qquad \approx 5\ m\Omega$$

Specific Resistivity

- The specific resistivity of ρ copper can be assumed to be:

  ρ = 1.78 x 10^{-6} Ω-cm for a temperature at 25˚ C

Specific Resistivity

- The resistance (at 25 ^{o}C) of a copper conductor with 0.3 mm conductor width and 35 um copper thickness and 500 mm length will be:

$$R = (1.78x10^{-6}\ \Omega\text{-}cm)\ x\ \frac{Length}{A}$$

where: R in Ω

Length and width in mm

Thickness in microns

Specific Resistivity

Converting units to cm

$$Thickness = 35\ um = 0.0035\ cm$$
$$width = 0.3\ mm = 0.03\ cm$$
$$L = 500\ mm = 50\ cm$$

Solving for the conductor cross-section area

$$A = 35x10^{-3}\ cm\ x\ 0.03\ cm = 105x10^{-6}\ cm^2$$

Solving for the resistance

$$R = 1.78x10^{-6}\ \Omega\text{-}cm\ x\ \frac{50\ cm}{105x10^{-6}\ cm^2}$$
$$R = 0.85\ \Omega$$

Resistance and Temperature

- Its is known that when current flows through a conductor, its temperature rises due to the joule effect (The copper resistance increases with temperature).
- It is known that

$$R_t = R_o[1 + \alpha(T_1 - T_0)$$

where:

R_t = resistance at temperature T_1

R_0 = resistance at temperature T_0

α = temperature coefficient of conductivity

Resistance and Temperature

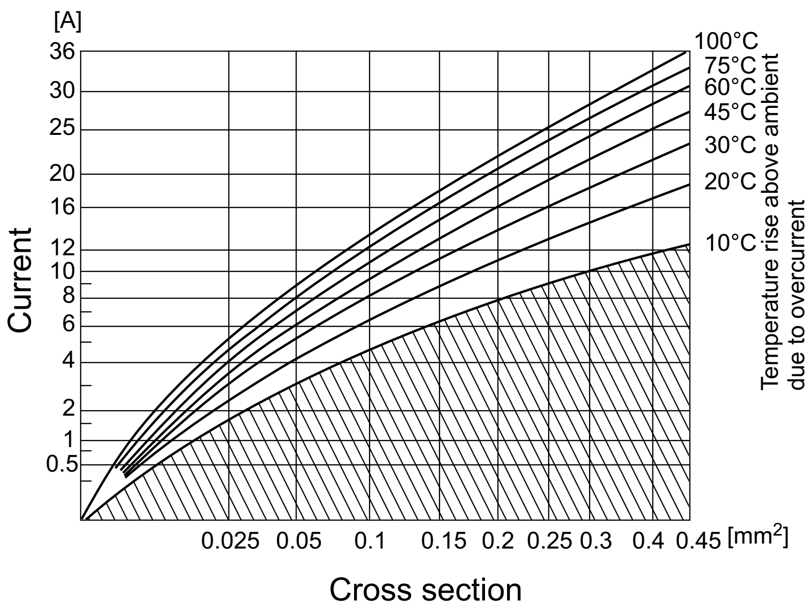
- To illustrate the theory, assume that the inside temperature in an electric equipment is 80˚ C while the outside temperature is 20˚ C. Taking temperature coefficient for copper conductivity as +0.0039, the resistance of 0.5 mm conductor, 10 cm long (w/ 100 Ω resistance would be:

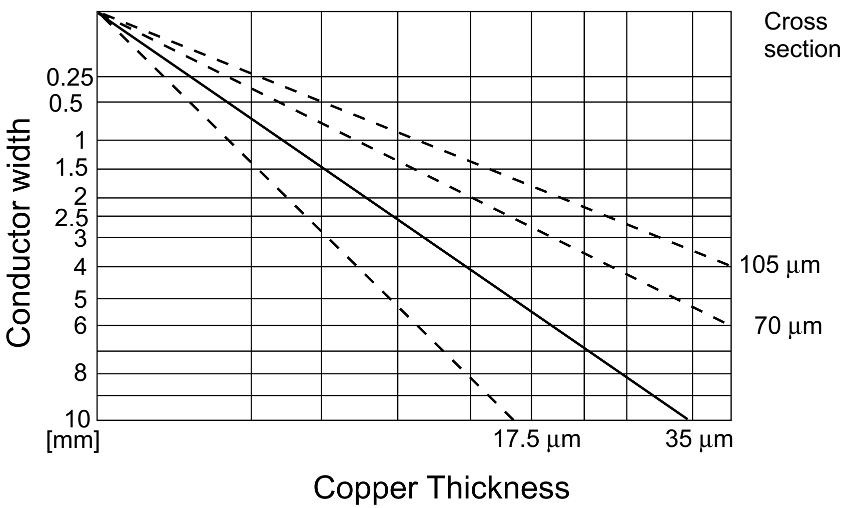
Resistance and Temperature

$$R_t = 100\ [1 + 0.0039\ (80 - 20)]$$
$$R_t = 100\ [1 + 0.0039\ x\ 60] = 123.4\ m\Omega$$

- This shows that the resistance has increased by 23.4 percent.
- All conductors which carry current will be at a temperature higher than temperature rise which depends upon the current, conductor width and copper thickness.

Resistance and Temperature





Conductor widths for safe operating temperatures.

Recommended Current Carrying Capacity of Traces

Conductor Width (in)	1/2 oz	1 oz	2 oz	3 oz
0.005	0.13	0.50	0.70	1.00
0.01	0.50	0.80	1.40	1.90
0.02	0.70	1.40	2.20	3.00
0.03	1.00	1.90	3.00	4.00
0.05	1.50	2.50	4.00	5.50
0.07	2.00	3.50	5.00	7.00
0.10	2.50	4.00	7.00	9.00
0.15	3.50	5.50	9.00	13.00
0.20	4.00	6.00	11.00	14.00

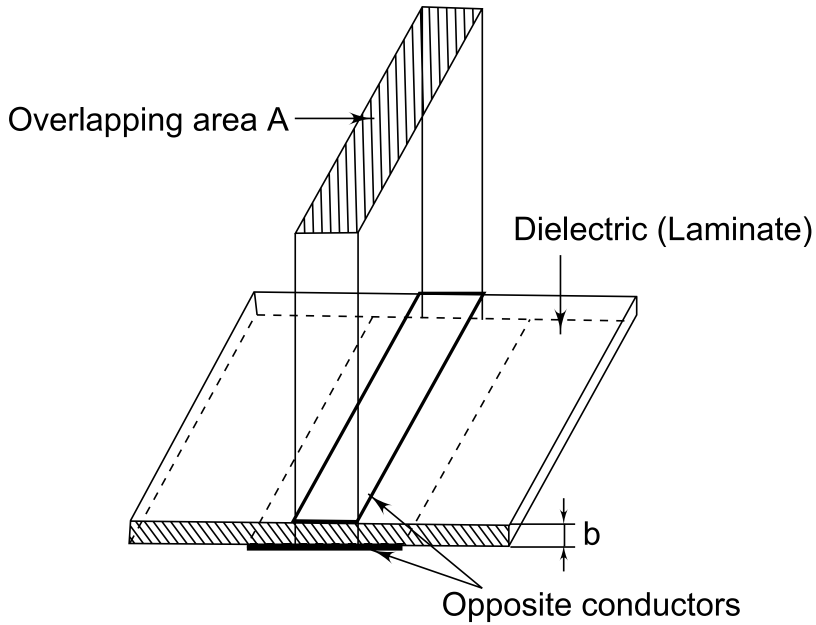
* copper thickness measured in 1 ounce per 1 square foot

Capacitance Consideration

- Capacitance is a parameter of considerable importance, particularly in the design of PCBs at high frequency.
- The capacitance comes into play in the following two situations:
  - Capacitance between conductors on opposite sides.
  - Capacitance between adjacent conductors.

Capacitance Between Conductors on Opposite sides of the PCB

- Two PCB conductors lying one above another and separated by a dielectric (laminate) form a capacitor whose approximate capacitance can be calculated from the basic capacitor formula:
  - C = 0.886 x ε x A/b (pF)
  - A = total overlapping area (cm^2)
  - b = thickness of dielectric
  - ε = relative dielectric constant, whose value is normally available from the manufacturers of the the PCB laminates



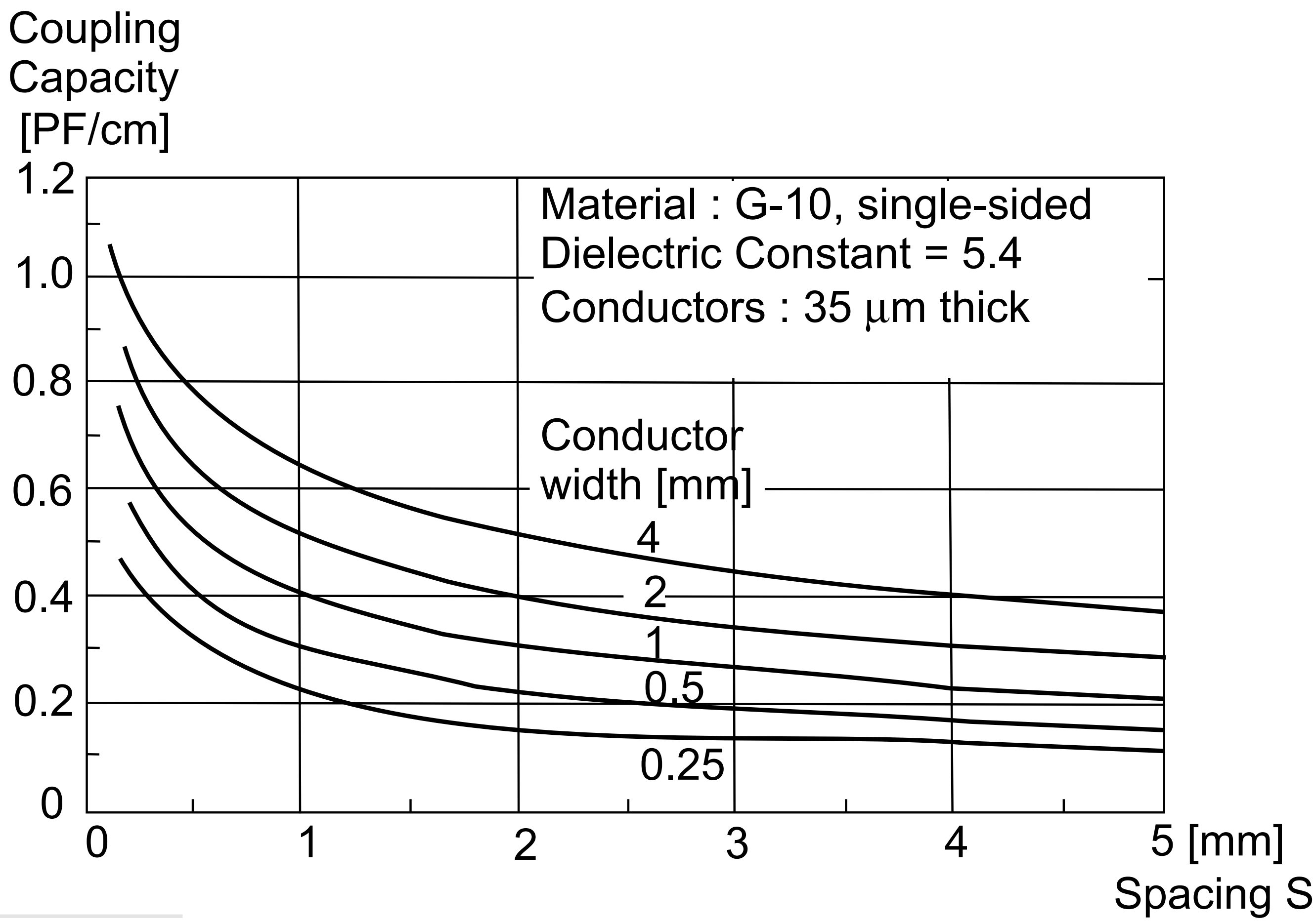
Capacitance between two conductors separated by a dielectric (conductors on opposite sides of the PCB).

Power Supply Spikes

- In order to reduce the magnitude of spikes on supply and the ground lines of fast switching circuits, it is advisable to run these lines exactly on <u>opposite sides of the PCB.</u>
- The supply and ground lines form a distributed de-coupling capacitor across the PCB.
- The TTL circuit on a PCB will give narrow current spikes on the supply line which will be drawn from the de-coupling capacitor without disturbing the supply voltage.

Capacitance Between Adjacent Conductors

- It is a function width, thickness and spacing as well as the dielectric constant of the board material.
- For all practical purposes, the value of coupling capacitance, (pF/cm) for a G-10 laminate with dielectric constant of 5.4 and conductor thickness of 35 um is given on the next figure.



Capacitive Coupling between adjacent conductors as a function of spacing.

Capacitance Between Adjacent Conductors

- To Illustrate a practical case, let us assume two conductors of 1 mm width which are running parallel with 1 mm spacing for a length of 100 mm. From the figure, it is found that the coupling capacitance comes out to be 0.4 pF/cm. This gives a total effective capacitance as 0.4 x 10 = 4 pF.

Capacitance Between Adjacent Conductors

- The capacitance coupling between adjacent tracks is normally undesirable.
- To reduce coupling:
  - Keep critical conductors narrow and provide sufficient spacing between them.
  - Run a ground line between the critical conductors, if possible. The broader the ground line, the better would be the result.

Capacitance Between Adjacent Conductors

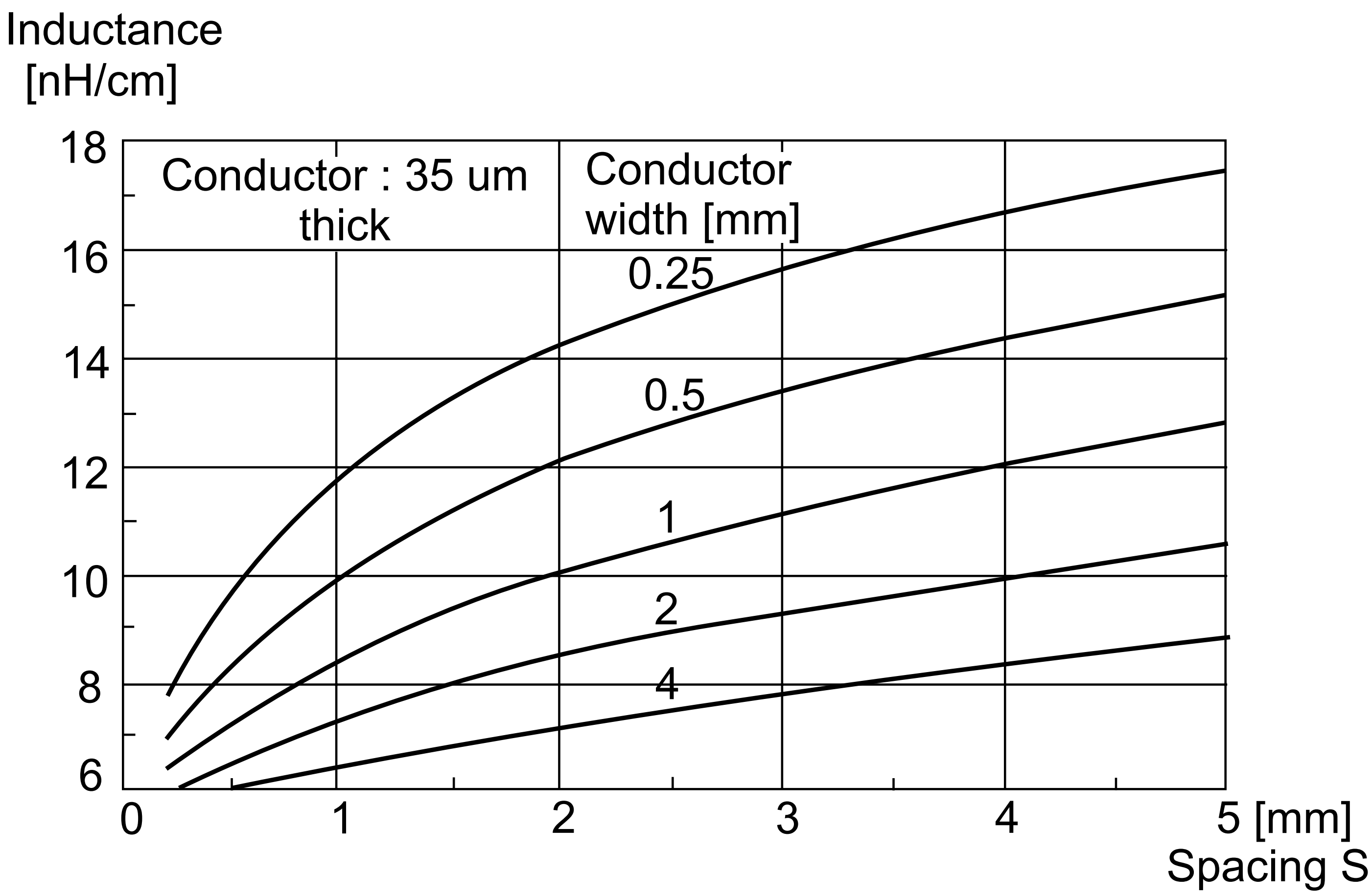
- If a ground line is present, the two signal conductors should run as close to the ground line as possible.
- This would keep the capacitance coupling to ground high, while coupling between signal lines at the same time becomes less.

Inductance of PCB conductors

- In logic circuits with a clock rate of only 10 kHz, high frequency components of the rectangular shaped signals can often cause problems.
- In such situations, it is important to know the inductance of the conductor arrangement.

Inductance of PCB conductors

- For a given type of copper clad board having 35 um copper conductor thickness, the inductance of parallel running conductor widths can be calculated using the next figure.



Inductance of parallel running conductors.

High Electrical Stress

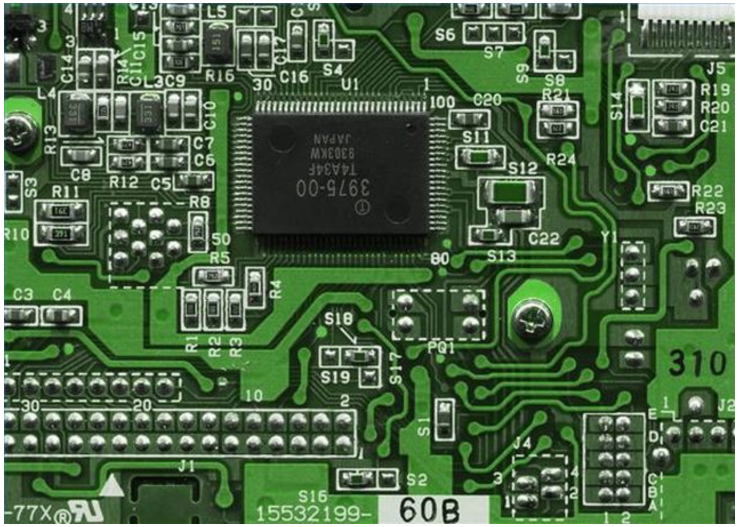
- The increasing density of interconnection in printed circuit boards demands that the designer should progressively decrease:
  – spacing and the size of conductor paths
  – PTH diameters
  – pad areas etc.
- Increasing level of integration is naturally accompanied by an increase of electrical stress.

Component Placement Rules

- In a highly sensitive circuit, the critical components are placed first and in such a manner as to require minimum length for the critical conductors.
- In a less critical circuit, the components are arranged exactly in the order of signal flow. This will result in a minimum overall conductor length.

Component Placement Rules

- In a circuit where a few components have considerably more connecting points than the others, these key components have to be placed first and the remaining ones are grouped around them.



ICs like microprocessors which has a lot of connecting usually placed first then surrounded by other components.

Component Placement Rules

- The general rule is to place first components, whose position is fixed for the final fitting and interconnections, e.g. connectors, heat sinks, etc. Then place the components which are connected to these fixed components.
- Components should be placed on the grid of 2.5 mm.

Component Placement Rules

- Among the components, larger components are placed first and the space in between is filled with smaller ones.
- All the components should be placed in such a manner that disordering of other components is not necessary if they have to be replaced.

Component Placement Rules

- Components should be placed in a row or a column, so that it gives a good overview.

Conductor Spacing

- Conductor Spacing is generally based on voltage breakdown or flashover between adjacent conductors.
- The conductor spacing is determined by the peak voltage difference between adjacent conductors, capacitive coupling parameters and the use of coating (mask).

Conductor Spacing

- Recommended conductor spacing:
  - General designs: 0.025 mm voltage limitation 400 VDC high density
  - Preferred AC or peak minimum is 0.015 mm, voltage limitation 50 VDC or AC peak.
  - The minimum by exception is 0.010 mm.

Additional Considerations for Conductor Spacing

- Critical-Impedance or high frequency components should be placed very close together to reduce critical stage delay.
- Transformers and inductive elements should be isolated to prevent coupling.
- Inductive signal paths should cross at right angles.
- Components which produce any electrical noise from movements of magnetic fields should be isolated.

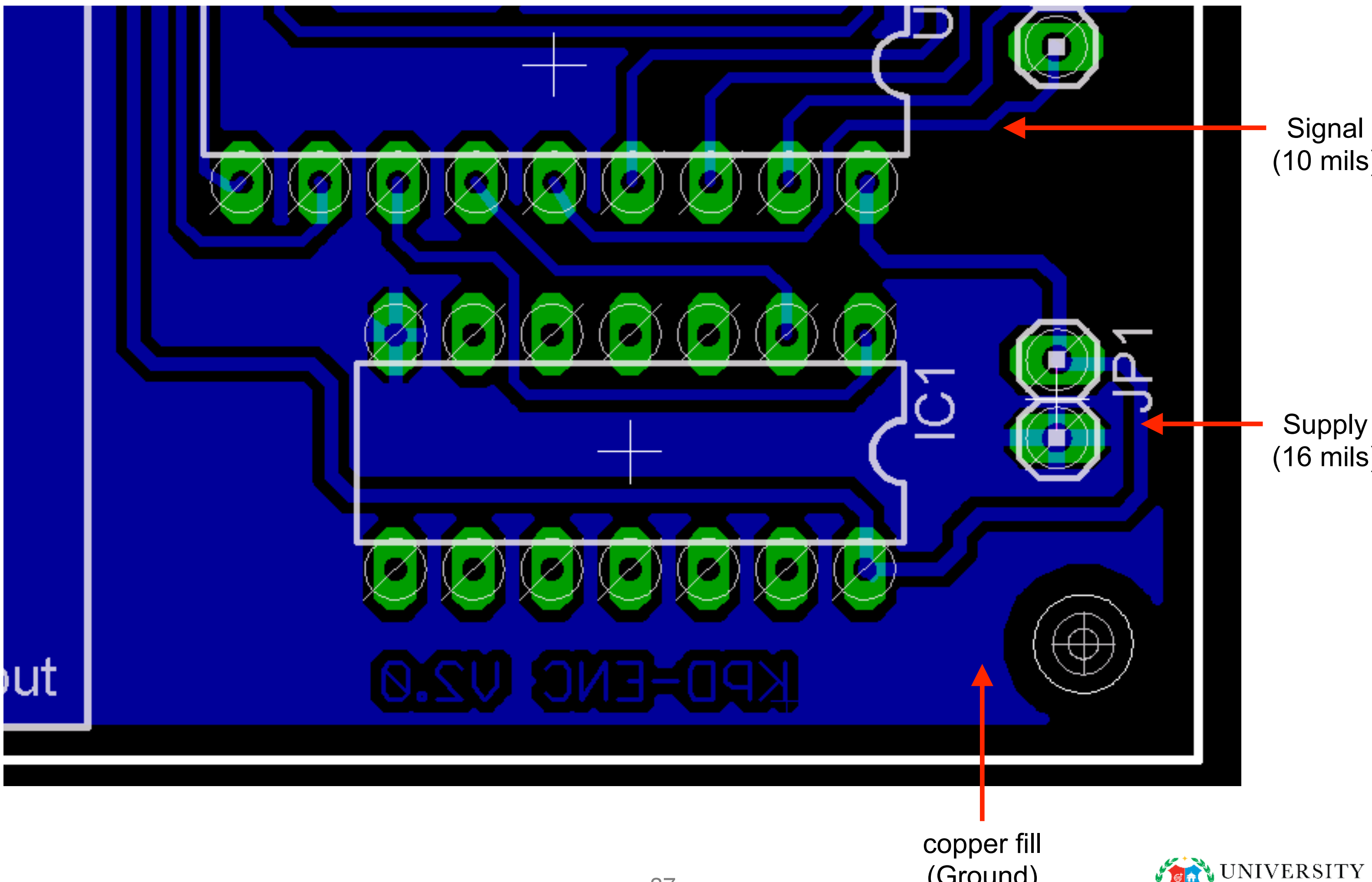
Conductor Width

- General reference for conductor width is:
  - $W_{ground} > W_{supply} > W_{signal}$
    - Where
      - W_{ground} is ground conductor
      - W_{supply} is supply (VCC, VDD) conductor
      - W_{signal} is signal conductor (bus or net)
  - The ground conductor should the most copper volume to provide good grounding and protecting the circuit from power surges.

Conductor Width

- For TTL circuits:
  - $W_{ground} > 2 \times W_{supply}$
  - $W_{supply} > 2 \times W_{signal}$

Conductor Width



University of San Carlos | Department of
COMPUTER ENGINEERING

CpE 2303L
CpE Drafting and Design

End of Lecture

This material is prepared by Van B. Patiluna with contents from the reference textbook and other sources.

References:

R.S. Khandpur. Printed Circuit Boards: Design, Fabrication, Assembly and Testing. New York: McGraw-Hill, 2006.

EAGLE ver 4.16 Manual (2003).