EMBEDDED SYSTEMS(EE30004)
Homework 11

Submitted by:

Pratyush Jaiswal
18EE30021

Q1. Briefly discuss designing of multilayer PCB.

Multilayer PCB is a Printed Circuit Board with more than 2 layers. A Double-Sided PCB has two conductive layers on top and bottom of the PCB substrate. A Multilayer PCB must have a minimum of 3 conductive layers of conductive material or copper layer. All the layers are interconnected with copper plated holes. The layers can be 4, 6, 8…upto 40 layers.

A multilayer PCB is complex by design. The top and bottom layers look same as a double sided PCB but has staked layers on both sides of the core. All the layers are compressed to form a single Multilayer PCB where all the layers are interconnected via copper plated holes.

All the active and passive electronic components are assembled on top and bottom layers. All the inner stacked layers are meant for routing. Both through-hole electronic components and Surface Mount Components (SMD) can be soldered on either side of this type of PCB. SMD components can be soldered with Surface Mount Technology and other PCBA Tools. A general **Multilayer Printed Circuit Board** has following layer stacking:

- Top Layer (*Electronic Components*)
- Inner Layer-1 (*Routing*)
- Inner Layer-2 (*Routing*)
- Inner Layer-3 (*Routing*)
- Bottom Layer (*Electronic Components*)

Multilayer PCB manufacturing process needs special precaution as there are more chances of cross connections, overlaps, tracing in copper areas, etc. The whole process needs to be done in an ESD-Safe and Cleanroom environment. Special manufacturing setup and equipment are needed for 8+ layer PCB manufacturing process.

Following are the steps involved in Multilayer PCB Manufacturing Process:

1. The process starts with designing Layout of the PCB using any PCB designing software / CAD Tool (Proteus, Eagle, OrCAD).
2. Next step is to make the Inner Layer Core. Laminate of desired thickness is treated with Copper foil, dry film resist and UV light to make the inner layer core.
3. Next step is lamination. *This process include*: Inner layer core, prepeg sheets and copper foil sheets. The sheets of material are staked on each other and holes are used to align them as they are stacked up.
   For a 4 layer board, Staking of layers is as follows: Bottom Layer of Copper Foil – Prepeg Sheets – The Inner Layer Core – More Prepeg Sheets – Finally Copper Foil Sheet on Top.
4. Next step is to apply pressure, heat and vacuum using a heated hydraulic press. Vacuum is important to make sure there is no air trapped between the layers. This process ends over 2 hours depending on number of layers.
5. Once cured, resins from the prepegs join the sheets, core, and foil together forming a multilayer PCB.

Q2. Briefly describe what are differential line drivers.

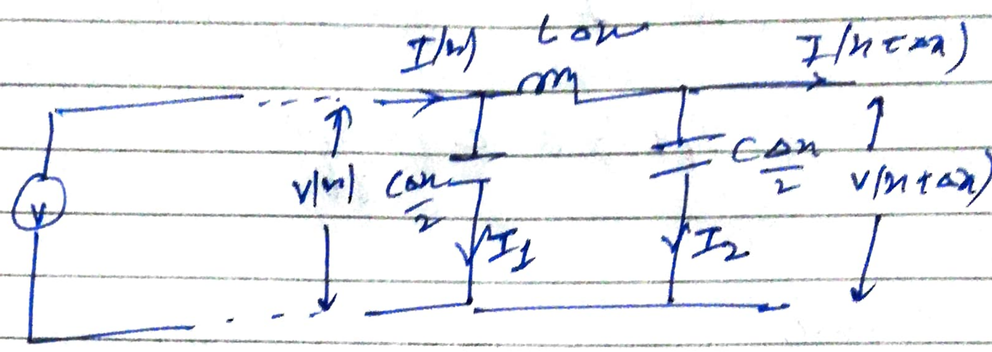
Differential line drivers are electronic amplifier circuits that are used for driving loads such as transmission lines using the differential signalling technique. The technique sends the same electrical signal as a differential pair of signals, each in its own conductor. The pair of conductors can be wires (typically twisted together) or traces on a circuit board. The receiving circuit responds to the electrical difference between the two signals, rather than the difference between a single wire and ground. Differential pairs are usually found on printed circuit boards, in twisted-pair and ribbon cables, and in connectors. A differential driver can convert a single-ended signal to differential, but it can equally merely buffer another differential signal. Differential signals are commonly generated and received on some inputs, outputs and IOs of large devices such as microprocessors. Differential line drivers are used because:

1. They resist more electromagnetic noise compared to one conductor since the receiving circuit only detects the difference between the wires.
2. They are more resistant to noise (for a given supply voltage, they provide twice the noise immunity of a single-ended system)
3. They are capable of carrying high-bitrate signals more reliably.

Pratyush Jaiswal
18EE30021

Date: ___/___/___
Page: ______

Q3)



Here, we choose the x-axis along the line or cable, and divide it by the means of sections perpendicular to the z-axis into short segments of $\Delta z << \lambda$, where λ is the wavelength.

The current i and v is across the conductors in one segment will be practically constant, while both i & v will vary with x and t down the wire.
We assume that through one cross-section the two conductors carry equal currents in magnitude but opposite in magnitude.

Applying Kirchoff's current law to the circuit,

$$I(x+\Delta x) = I(x) - \Delta I \quad -(1)$$

Expanding using Taylor's Series,

$$I(x+\Delta x) = I(x) + \frac{\partial I}{\partial x}\Delta x \quad -(2)$$

Using Ohm's Law and relations between the charge, current & voltage in a capacitor $\left(i = \frac{\partial Q}{dt}\right)$ and $Q = CV$),

Date: __/__/__
Page:

$$\therefore I_1 = \frac{C\Delta x}{2}\frac{\partial V(x)}{\partial t}, \quad I_2 = \frac{C\Delta x}{2}\frac{\partial V(x)}{\partial t}.$$

$$I_1 + I_2 = C\Delta x \frac{\partial V(x)}{\partial t}$$

Here,

$$I(x+\Delta x) + I_1 + I_2 = I(x)$$

$$I(x+\Delta x) = I(x) - (I_1 + I_2) \quad \text{--- (3)}$$

Using (1) & (3),

$$\Delta I = I_1 + I_2$$

$$\Delta I = C\Delta x \frac{\partial V(x)}{\partial t} \quad \text{--- (4)}$$

Using (2) & (4)

$$\boxed{-\frac{\partial I}{\partial x} = C\frac{\partial V(x)}{\partial t}}$$

↳ first telegrapher's Equation

Using Kirchoff's Voltage law,

$$V(x+\Delta x) = V(x) - \Delta V = V(x) + \frac{\partial V}{\partial x}\Delta x$$

$$= V(z) - L\Delta x \frac{\partial I}{\partial t}$$

$$\boxed{\frac{-\partial V}{\partial x} = L\frac{\partial I}{\partial t}}$$

↳ Second Telegrapher's Equation