Name : Aanchal Naman
Roll No : 16010120110
Batch : B2
Aanchal.

PAGE NO. 1
DATE 7 5 2021

Assignment No. 6

PCB Workshop

To understand Design and Manufacturing of Printed Circuit Board [PCB].

Q1) Describe the function of double-sided UV Exposure Unit.

A) A double-sided UV vaccum exposure unit is an essential tool for the production of parts such as double-sided PCBs. The vaccum maker these UV exposure units suitable for processing UV sensitive flexible materials. For the ultimate in reproduction definition, a vaccuum unit has to be the choice. It uses a vaccuum to ensure the art work and material to be exposed are held in perfect contact. When fine tracks are required on a PCB or detailed difinition on a graphic reproduction, a vaccuum UV exposure unit should be chosen. Double sided can be carried out by switching mode on the unit.
All units feature the rubber frame and glass plate vaccuum system for the ultimate exposure resolution. An adjustable selectable 0-600 seconds or 0-100 mins timer is fitted to give the operator greater flexibility. A vaccuum gauge is also fitted in the machine enabling the operator to check the level of vaccuum before activating the UV Tubes. A vaccuum down to 0.2 bar can be created. All units come complete with an IEC socket and moulded 13 amp plug. The fluorescent lamps emit a wavelength of 350-400 nm.

Q2) List the softwares used for PCB layout design and explain how to design layout using DIPTRACE or EAGLE software.

A) Some softwares used for PCB layout design include Altium designer, Fusion 360, NI Multisim, Kilad EDA, Autodesk EAGLE, Dip Trace, CAM 350, Ultiboard, PCB artist by Advanced circuits, Solid works PCB, X circuit etc.

Name: Aanchal Thaman
Roll No: 16010120110
Batch: B2
Aanchal.

PAGE NO: 2
DATE: 7 5 2021

Creating a PCB layout using EAGLE software:

→ Getting software in the system
→ Planning phase and setup.
→ Library setup.
→ Part placement (1) → Placing resistors
→ Part placement (2) → Placing LEDs
→ Part placement (3) → Placing switches
→ Part placement (4) → Placing Connectors
→ Adding resistor to values
→ Renaming connectors
→ Adding LED grounds
→ Flipping connectors
→ Connecting the parts
→ Adding a few extras and finishing connections
→ Adding power to the schematic
→ Error checking
→ Board creation
→ Circuit builder setup
→ Precision part placement
→ Finishing touches and DRC.

Q3) Write and explain in short, the steps for fabrication of PCB.

A) PCB fabrication is the process or procedure that transforms a circuit board design into a physical structure based upon the specifications provided in the design package. This physical manifestation is achieved through the following actions or techniques

→ Imaging desired layout on copper clad laminates
→ Etching or removing excess copper from inner layers to reveal traces and pads.
→ Creating the PCB layer stack up by laminating (heating and pressing) board materials at high temperatures.

Name : Aanchal Tharan
Roll no : 160101 20110
Batch : B2
Aanchal

PAGE NO. 3
DATE 7 5 2021

→ Drilling holes for mounting holes, through hole pins and vias.
→ Etching or removing excess copper from the surface layer(s) to reveal traces and pads.
→ Plating pin holes and via holes.
→ Adding protective coating to surface or solder masking
→ Silkscreen printing reference and polarity indicators, logos or other markings on the surface.
→ Optionally, a finish may be added to copper areas of surface

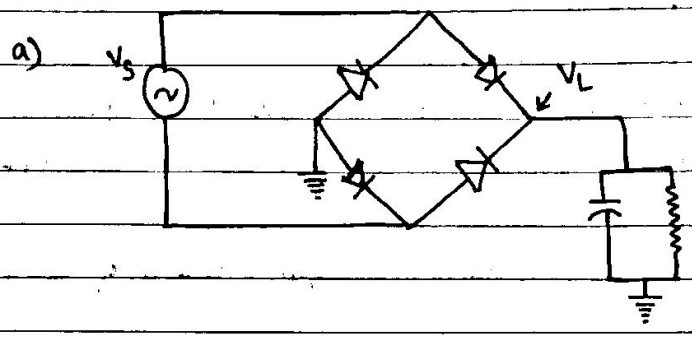
B4) Explain PCB in detail.

A) A printed circuit board (PCB) mechanically supports and electrically connects electrical or electronic components using conductive tracks, pads and other features etched from one or more sheet layers of copper laminated onto and/or between sheet layers of a non-conductive substrate. Components are generally soldered onto the PCB to both electrically connect and mechanically fasten them to it. Printed circuit boards are used in all but the simplest electronic boards. They're also used in some electrical products, such as passive switch boxes.
PCBs can be single-sided (one copper layer), double-sided (two copper layers on both sides of one substrate layer), or multi-layer (outer and inner layers of copper, alternating with layers of substrate). Multi-layer PCBs allow for much higher component density, because circuit traces on the inner layers would otherwise take up surface space between components. The rise in popularity of multilayer PCBs with more than two, and especially with more than four, copper planes was concurrent with the adoption of surface mount technology. However, multilayer PCBs make repair, analysis, and field modification of circuits much more difficult and usually impractical.

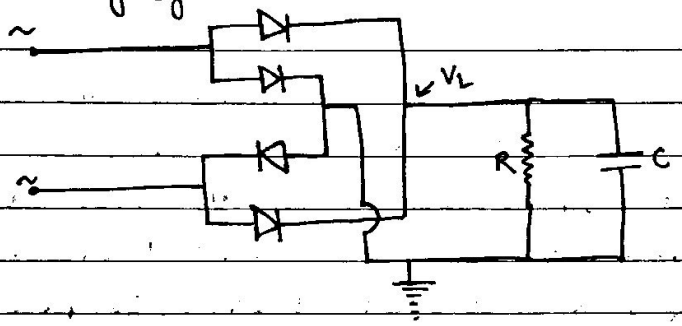
Name : Aanchal Thaman
Roll No : 160101 20110
Batch : B2
Aanchal.

PAGE No. 4
DATE 7 5 2021

Q5) Draw any one of the following schematic diagrams with its PCB layout.

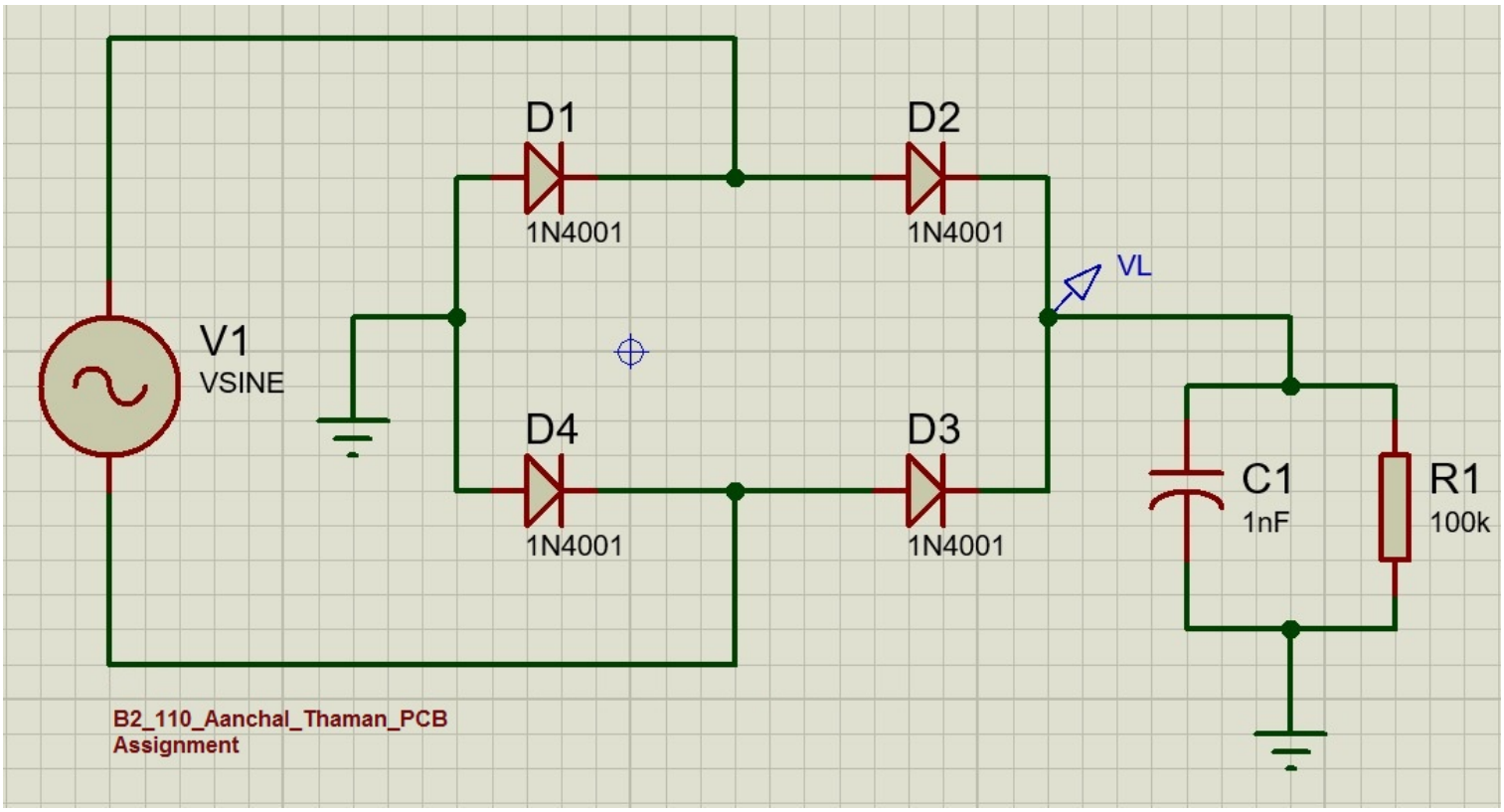
a) 

A) Plan drawing by hand -



Plan drawing in software -

Question 5)


D1
1N4001
D2
1N4001
VL
V1
VSINE
D4
1N4001
D3
1N4001
C1
1nF
R1
100k
B2_110_Aanchal_Thaman_PCB
Assignment