NARAYANA ENGINEERING COLLEGE::NELLORE

AUTONOMOUS

**DEPARTMENT OF ELECTRICAL & ELECTRONICS ENGINEERING**

# REPORT ON VALUE ADDED COURSE

A value added course on **"A SIMPLISTIC PRINTED CIRCUIT BOARD LAYOUT DESIGN WITH PROTEUS"** has organized by the department of Electrical & Electronics Engineering in association with Pantech e-Learning Solutions, Chennai during 02.02.2022 to 06.02.2022.

No. of students participated: 63

## COURSE CONTENTS:

In theory sessions the resource persons have gave the following details about the software and the circuits to be designed.

**Introduction to Proteus:**

Proteus professional is a software combination of ISIS schematic capture program and ARES PCB layout program. This is a powerful and integrated development environment. Tools in this suit are very easy to use and these tools are very useful in education and professional PCB designing. As professional PCB designing software with integrated space based auto router, it provides features such as fully featured schematic capture, highly configurable design rules, interactive SPICE circuit simulator, extensive support for power planes, industry standard CADCAM & ODB++ output and integrated 3D viewer.

Up to now we have discussed about the basics and software description. Now we are entering into the designing section. Run the ISIS professional program by clicking the icon on the desktop, then this splash screen will appear.

Next, a work space with interface buttons for designing circuit will appear as shown in figure below. Note that there is a blue rectangular line in the workspace; make sure that whole circuit is designed inside the rectangular space.

Next step is selecting the components to our required circuit. Let us take one example is designing of 38 kHz frequency generator by using 555 timer IC. The circuit diagram is shown in below image. There is another way to select the components. In work space left side there is a tool bar. In that tool bar click the component mode button or pick from library.

Select the all components from library, that components are added to devices list. Click on the device and change the angle of the device by using rotate buttons. Then click in the work space then the selected component is placed in work space. Place all the devices in work space and put the curser at the component pin end then draw the connections with that pen symbol. Connect all the components according to circuit then that designed circuit is show in below image. If any modifications want to do to the component place the mouse point and click on right button then option window will open. That is shown in below figure.

After completion of designing save with some mane and debug it. This is virtual simulation means without making circuit we can see the result in virtually through this software and we can design the PCB layout to our required circuit with this software.

# PCB Layout Printing

1. Top copper layer printout. It is no need to this because this is single layer PCB. It is only for duel layer PCB.

2. Bottom copper layer printout. While printing this layer except bottom copper and board edge remaining all boxes will unselect position in layers/artwork part. Next select the scale as 100%, select the rotation as X horizontal and the important thing is reflection should be select mirror. Because after printing this layer on paper it is placed on the copper board in opposite direction means the printed side should be faced to copper layer. That's why we are selecting reflection as mirror.

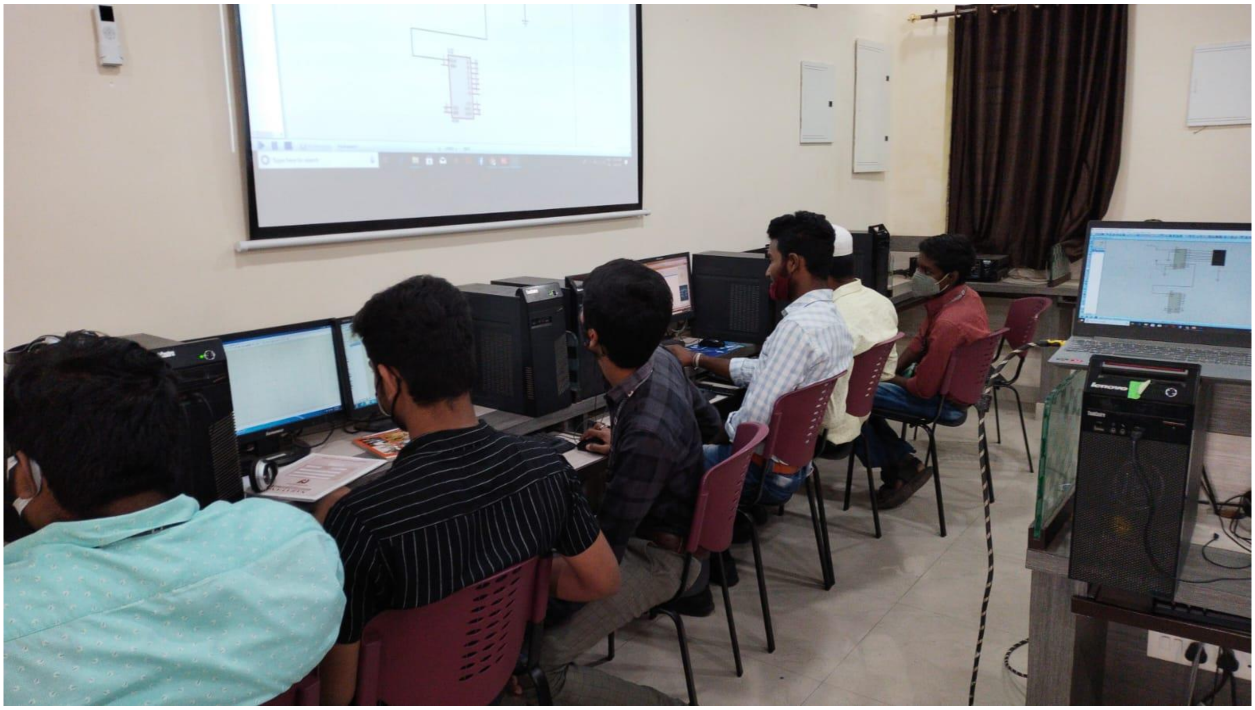
3. Top silk layer printout. This combination with bottom copper. In single layer PCB we are using only bottom copper. That means components are present in top side. So top silk layer prints the components view. This will prints the place of the components. While printing this layer except top silk and board edge remaining all boxes will unselect position in layers/artwork part. Remaining all selections is same except reflection. Reflection should be selected in normal mode.

4. Bottom silk layer printout. This is for duel layer PCB.

5. Solder resist layer printout. This is for preventing from short circuits. To print this layer select the mode as solder resist and click the bottom resist box and board edge and reflection mirror mode. Because this is also same like bottom copper layer printing.

6. SMT mark is not for this because in our circuit we don't use the SMT modules.

7. Drill plot layer printing. This layer indicates the drill place and drill hole size. While printing this layer only drill and board edge boxes is in selected position. Reflection is normal mode.



After printing the all layers next thing is PCB itching. That means copper tracks designing with layer printed paper.

- Take the copper layered PCB board and cut the board according to our requirements.
- Place the bottom copper layer printed paper on the copper PCB board by facing the printed surface to copper layer.
- Fix the paper and board without moving.
- Apply heat by iron box or any other sources to the white printed paper.
- At the printing machine carbon powder sprayed on white paper and applying heat, the model is scanned, according to that model carbon powder is stick on the paper and remaining powder on the paper is cleaned.
- Here we are applying same process. Place the face of printed paper to copper layer and applying heat to it then the carbon power stick to copper layer. So the paper is merged with board.
- Drop the board into water and remove paper slowly then the carbon layer fixed on copper board.

- Then drop the board into ferric-chloride liquid. Then the copper react with ferric-chloride and the copper which is not having carbon layer is dissolved in ferric-chloride and remaining part which is having the carbon layer that is not dissolved in ferric-chloride.
- Next clean the board with sand paper. Erase or remove the carbon layer by scratching the board with sand paper. Then the carbon layer completely removed and copper layer shown out.
- Make the holes by drilling according to drill position layer.
- Then finally place the proper components in correct places and by using soldering kit fix the components to the board.
- Then finally cut the extra pins come from holes by cutter. Then our circuit is ready.

The session was ended with a valedictory function to felicitate and to present vote of thanks to the deserved deligates.

**HOD-EEE**