**DAILY ASSESSMENT**

|  |  |  |  |
| --- | --- | --- | --- |
| **Date:** | **09/06/2020** | **Name:** | **Dhavala** |
| **Course:** | **Learn KiCad. Printed circuit Board Design** | **USN:** | **4AL17EC027** |
| **Topic:** | **Up and running** | **Semester & Section:** | **6TH SEM & A Section** |
| **Github Repository:** | **Dhavala27** |  |  |

|  |
| --- |
| **FORENOON SESSION DETAILS** |
|  |

|  |
| --- |
| **Report**  THE DESIGINING PROCESS  Designing a circuit board consists of four main parts:  •Draw the schematic (circuit diagram)  •Generate a netlist for the schematic  •Lay out the circuit board  •Generate Gerber files that are sent to the PCB manufacturer  Drawing the Schematic and Generating the Netlist  •The schematic editor used to draw circuit diagrams in KiCad is called EESchema. Once the circuit diagram is drawn, a netlist is generated from it. This is done by simply clicking a button in EESchema.  •The netlist contains information on all the components in the schematic and the  connections between components.  •The Netlist file is a file that contains information about the circuit, it’s components,  associated footprints, labels and pin numbers and many other things.  •Our PCBnew, which is the PCB editor, would read this file and load the appropriate  footprints from the library and that will do the layout and wiring  . |

**DAILY ASSESSMENT**

|  |  |  |  |
| --- | --- | --- | --- |
| **Date:** | **09/06/2020** | **Name:** | **Dhavala** |
| **Course:** | **mysql** | **USN:** | **4AL17EC027** |
| **Topic:** | **Section 4: Outputting And Processing Data**  **Section 5: Dealing With Variables**  **Section 6: Inserting And Using Database Data** | **Semester & Section:** | **6TH SEM & A Section** |
| **Github Repository:** | **Dhavala27** |  |  |

|  |
| --- |
| **FORENOON SESSION DETAILS** |
|  |

|  |
| --- |
| **Report** |