**DAILY ASSESSMENT FORMAT**

|  |  |  |  |
| --- | --- | --- | --- |
| **Date:** | **09/06/2020** | **Name:** | **PRIYA P RAO** |
| **Course:** | **PCB Design** | **USN:** | **4AL18EC041** |
| **Topic:** | * **Introduction to KiCad**  Introduction to PCB design with KiCad | **Semester & Section:** | **4th sem ‘A’ section.** |
| **Github Repository:** | **Priya-Rao** |  |  |

|  |
| --- |
| **FORENOON SESSION DETAILS** |
| **Image of session**  **C:\Users\Pawan\Desktop\kikad.PNG**  **C:\Users\Pawan\Desktop\rrrr.PNG** |
| Chapter 1: Introduction to KiCadA PCB or board describes the layout of the components, holes, pads, traces, connections, mechanical design and silkscreen. Designing a board is the end game of the all PCB design work, and it is this file that gets the sent off to a fab house for manufacturing. **Before we put together a schematic and design on a board, we must learn how to build our own parts. To start, install KiCad, select file then new project and we have to name it accordingly.** Chapter 2: Introduction to PCB design with KiCad **In this tutorial we will learn the very basics of using KiCad for designing a circuit board. It is assumed that you already installed KiCad. If not head over to**[**KiCads download 22**](https://kicad-pcb.org/download/)**page and follow the instructions for your operating system.**  **In today’s session I have learnt about;**   * [**Creating a Project**](https://forum.kicad.info/t/tutorial-introduction-to-pcb-design-with-kicad-version-5-1/20600/2) * [**Designing the schematic**](https://forum.kicad.info/t/tutorial-introduction-to-pcb-design-with-kicad-version-5-1/20600/3)**(Schematic using symbols and Wires)** * [**Lay out the board**](https://forum.kicad.info/t/tutorial-introduction-to-pcb-design-with-kicad-version-5-1/20600/4)**(Visibility options, Footprints, Traces and board outline)** * [**Improve the schematic**](https://forum.kicad.info/t/tutorial-introduction-to-pcb-design-with-kicad-version-5-1/20600/5)**(Power symbols, labels and electrical rule check)** * [**Improve the Layout**](https://forum.kicad.info/t/tutorial-introduction-to-pcb-design-with-kicad-version-5-1/20600/6)**(Updating from schematic changes, copper zones, vias, net selection and exchanging units)** * [**Get ready for manufacturing**](https://forum.kicad.info/t/tutorial-introduction-to-pcb-design-with-kicad-version-5-1/20600/7)**(Design rules and gerber export)** |

|  |  |  |  |
| --- | --- | --- | --- |
| **Date:** | **09/06/2020** | **Name:** | **PRIYA P RAO** |
| **Course:** | **JAVA** | **USN:** | **4AL18EC041** |
| **Topic:** | **Programming core JAVA** | **Semester & Section:** | **4th sem ‘A’ section** |
| **Github Repository:** | **Priya-Rao** |  |  |

|  |
| --- |
| **AFTERNOON SESSION DETAILS** |
| **Image of session**  **C:\Users\Pawan\Desktop\java.PNG** |
| **Chapter 2: Programming Core Java**   * **A Hello World Program** * **Using Variables** * **Strings: Working With Text** * **While Loops** * **For Loops** * **"If"** * **Getting User Input** * **Do ... While** * **Switch** * **Arrays** |