**DAILY ASSESSMENT FORMAT**

|  |  |  |  |
| --- | --- | --- | --- |
| **Date:** | **12/06/2020** | **Name:** | **Lavanya B** |
| **Course:** | **Kicad** | **USN:** | **4al17ec043** |
| **Topic:** | **PCB designing** | **Semester & Section:** | **6th A** |
| **Github Repository:** | **Lavanya-B** |  |  |

|  |
| --- |
| **FORENOON SESSION DETAILS** |
| **Image of session** |
| **Report**  **Creating PCB footprint component**  **Create a PCB footprint**  **A footprint describes the interface between the circuit board and the component it self. This is often called the land-pattern. At the least it needs to contain so called pads. It is also suggested to at least include the part outline and part identifiers on the silk and fab layers. If a footprint is specialized for a single component then include the part number and the manufacturer name. For generic footprints include all identifying parameters.**  **Add footprint search path**    **In KiCad, one can define some paths using an environment variable. A few environment variables are internally defined by KiCad, and can be used to define paths.**  **This is useful when absolute paths are not known or are subject to change. This is the case for "official" libraries built for KiCad:**  **-for the path of these libraries, when installed on your disk**  **-for the path of 3D shapes files used in footprint definitions.**  **For instance, the full path of connect.pretty footprint library is defined like this, when using the KISYSMOD environment variable to define the full path: ${KISYSMOD}/connect.pretty**  **Obviously, one can use a usual full path definition, if this full path is well known, and never changes.**  **Production file**  **The Gerber format is an open 2D binary vector image file format. It is the standard file used by printed circuit board (PCB) industry software to describe the printed circuit board images: copper layers, solder mask, legend, etc.**  **Change to the "PCB" edit mode on your Fritzing. And click the "Export for PCB" ▼ button on the lower toolbar. Make sure you click the arrow for other format options, not the main button, otherwise, Fritzing will export to PDF by default.**  **The data type selection menu will open. Click "Extended Gerber (RS-274x)**  **The "Choose a folder for exporting" dialog will open. Create a new folder and select it (I made the "gerber-files" folder.) The Gerber files and drill file will then be exported into the chosen folder.**  **Open the folder and delete the "Countdown\_pnp.txt". It is not needed for PCB manufacture.** |

|  |  |  |  |  |
| --- | --- | --- | --- | --- |
| **Date:** | **12/06/2020** | **Name:** | **Lavanya B** | |
| **Course:** | **JAVA** | **USN:** | **4al17ec043** | |
| **Topic:** |  | **Semester & Section:** | **6th A** | |
| **AFTERNOON SESSION DETAILS** | | | |
| **Image of session** | | | |
| **Report – Report can be typed or hand written for up to two pages.** | | | |