*Norwich Technical High School - Electronics Technology Date Updated: 01/05/2024*

*7 Mahan Drive*

*Norwich, CT 06360*

**TITLE: Standard Operating Procedure (SOP) for Eagle CAD**

**APPROVALS:**

**Author: Keyremy Manuel Vazquez Malave Date: January 5th, 2024**

**Review: Date:**

1. **Revision History:**

| **Initials** | **REV** | **DATE** | **SUMMARY OF CHANGES** |
| --- | --- | --- | --- |
| **K.M.V.M** | **A** | **1/05/2024** | **Initial Release** |
|  |  |  |  |

1. **Purpose:**

* **The purpose of this SOP is to guide users in utilizing Eagle CAD for designing printed circuit boards (PCBs) effectively and in a non-costly manner.**

1. **Safety:**

* **Ensure proper handling of electronic components and adherence to safety guidelines given by OSHA (The Occupational Safety and Health Administration) during the design and fabrication process.**

1. **Materials:**

* **Computer with Eagle CAD software**

**Installed.**

* **Schematics and component datasheets.**
* **Access to a printer for documentation.**
* **Access to the Internet.**
* **Access to a Thumb Drive.**

1. **Procedure:**

| **Step** | **Procedure** | **Picture** |
| --- | --- | --- |
|  | **Launching Eagle CAD:**   * **Open the Eagle CAD software on your computer.** |  |
|  | **Creating a New Project:**   * **Start a new project file and save it in an organized directory.** |  |
|  | **Designing Schematics:**   * **Create the schematic diagram by adding components and connecting them appropriately.** * **Label components and use nets to define connections.** |  |
|  | **PCB Layout:**   * **Transfer the schematic to the PCB layout by clicking on the 'Switch to Board' button.** * **Arrange components on the PCB layout, considering space and signal paths.** |  |
|  | **Routing:**   * **Use the autorouter or manually route traces to connect components on the PCB.** * **Ensure proper clearance and follow design rules.** |  |
|  | **Design Rule Check (DRC):**   * **Perform a Design Rule Check to identify and correct any violations.** * **Resolve errors related to clearances, trace widths, and other design rules.** |  |
|  | **Generating Gerber Files:**   * **Generate Gerber files for manufacturing, including layers for copper, silkscreen, and solder mask.** |  |
|  | **Documentation:**   * **Create comprehensive documentation, including assembly drawings, bill of materials (BOM), and fabrication notes.** |  |
|  | **Review:**   * **Have a peer review the design for feedback and improvements.** |  |
|  | **Finalization:**   * **Make necessary adjustments based on the review.** * **Save the final design files and documents. Step 11: Project Archiving** |  |
|  | **Project Archiving:**   * **Archive the project files for future reference.** * **Save a copy of the design files in a secure location.** |  |