*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** |
| **K.M.V.M** | **B** | **2/8/2024** | **Updated Version 1** |

1. **Purpose:**

* **The purpose of this SOP is to provide a systematic guide for utilizing Eagle CAD effectively and economically in the design of printed circuit boards (PCBs), ensuring consistency, quality, and adherence to best practices throughout the process.**

1. **Safety:**

* **To guarantee the safety of personnel and equipment, it is imperative to strictly adhere to safety guidelines provided by OSHA during all stages of the design and fabrication process. This includes proper handling of electronic components, use of appropriate protective equipment, and adherence to electrical safety protocols.**

1. **Materials:**

* **Computer with Eagle CAD software installed: Ensure the software is properly installed and updated to the latest version to leverage new features and bug fixes.**
* **Schematics and component datasheets: Gather necessary schematics and datasheets for all components involved in the design to ensure accurate representation and proper integration.**
* **Printer for documentation: Utilize a printer to produce hard copies of documentation, schematics, and design specifications for reference and review purposes.**
* **Internet access: Access to the internet is essential for obtaining component libraries, software updates, and troubleshooting assistance.**
* **Thumb Drive: Maintain a backup of project files and design documents on a thumb drive to prevent data loss and facilitate easy sharing and transfer between devices.**

1. **Procedure:**

|  |  |  |
| --- | --- | --- |
| **Step** | **Procedure** | **Picture** |
|  | **Launching Eagle CAD:**   * **Open the Eagle CAD software on your computer by double-clicking the application icon or selecting it from the Start menu.** |  |
|  | **Creating a New Project:**   * **Begin by creating a new project file to organize all design-related files and documents in a structured manner. Save the project file in a designated directory with an appropriate name for easy identification and retrieval.** |  |
|  | **Designing Schematics:**   * **Develop the schematic diagram by adding components from the Eagle CAD library and connecting them according to the circuit requirements.** * **Ensure accurate representation of components and connections, and label components with descriptive names and values for clarity.** * **Utilize nets to define connections between components and ensure proper signal flow throughout the circuit.** |  |
|  | **PCB Layout:**   * **Transfer the schematic to the PCB layout editor by selecting the 'Switch to Board' option from the toolbar or menu.** * **Arrange components on the PCB layout to optimize space utilization and minimize signal interference.** * **Consider factors such as component placement, routing complexity, and thermal management when arranging components on the board.** |  |
|  | **Routing:**   * **Use the autorouter or manually route traces to connect components on the PCB, ensuring proper signal integrity and impedance control.** * **Pay attention to signal paths, avoiding crossing traces, and maintaining adequate clearance between traces to prevent signal crosstalk and interference.** * **Follow design rules and guidelines provided by the manufacturer to ensure compliance with industry standards and best practices.** |  |
|  | **Design Rule Check (DRC):**   * **Perform a comprehensive Design Rule Check to identify and rectify any violations or errors in the design.** * **Verify compliance with design rules related to trace width, clearance, spacing, and other parameters to ensure manufacturability and reliability of the PCB.** |  |
|  | **Generating Gerber Files:**   * **Generate Gerber files required for manufacturing the PCB, including layers for copper, silkscreen, solder mask, and drill.**   **Archive all project files, including schematic, layout, Gerber files, documentation, and any related materials, for future reference and retrieval. Store a copy of the design files in a secure location, such as a cloud-based repository or dedicated server, to prevent loss or damage to critical data.Verify the accuracy and completeness of Gerber files by reviewing them in a Gerber viewer or CAD software before sending them to the manufacturer.** |  |
|  | **Documentation:**   * **Create detailed documentation to accompany the PCB design, including assembly drawings, bill of materials (BOM), fabrication notes, and any other relevant information.** * **Document design decisions, constraints, and specifications to provide a comprehensive reference for future revisions and modifications** |  |
|  | **Review:**   * **Conduct a thorough review of the PCB design with peers or stakeholders to gather feedback and identify potential areas for improvement.** * **Solicit input from individuals with diverse expertise to ensure the design meets all requirements and objectives.** |  |
|  | **Finalization:**   * **Incorporate feedback and suggestions obtained during the review process into the design as necessary.** * **Make any final adjustments or refinements to the design before proceeding to the next phase of the project.** |  |
|  | **Project Archiving:**   * **Archive all project files, including schematic, layout, Gerber files, documentation, and any related materials, for future reference and retrieval.** * **Store a copy of the design files in a secure location, such as a cloud-based repository or dedicated server, to prevent loss or damage to critical data.** |  |