

Task: Creating a Schematic Symbol and Footprint for the NE555 Timer IC

Objective of the Task

The objective of this task is to help students gain practical experience in PCB design and component library creation by:

- Understanding the pin configuration and functionality of the NE555 Timer IC.
- Learning how to create custom schematic symbols in PCB design software.
- Learning how to create and manage PCB footprints.
- Understanding the relationship between schematic symbols and PCB footprints.
- Developing proficiency in Electronic CAD (ECAD) tools.

Instructions for Performing the Task

Using an open-source PCB design software such as **KiCad** or **EasyEDA**, (may download Altium)perform the following:

1. Schematic Symbol Creation

- Create a custom schematic symbol/library for the NE555 Timer IC.
- Assign all 8 pins with the correct:
 - Pin numbers
 - Pin names
 - Electrical pin types (where applicable)

2. Footprint Creation

- Create a DIP-8 footprint for the NE555 Timer IC using standard package dimensions.
- Ensure proper pad spacing, pad numbering, and package outline.

3. Symbol–Footprint Association

- Link the created schematic symbol with the corresponding DIP-8 footprint.
- Verify that each schematic pin is correctly mapped to its respective PCB pad.

4. Verification

- Check the symbol for correctness and readability.
- Verify footprint dimensions using the manufacturer's datasheet.
- Confirm that all pin mappings are electrically correct.

Requirements for Submission

Students must submit the following:

- 1. Schematic Library File**
 - Custom library containing the NE555 Timer schematic symbol.
- 2. Footprint Library File**
 - Custom footprint library containing the DIP-8 package footprint.
- 3. Symbol Screenshot**
 - Clear screenshot of the completed schematic symbol.
- 4. Footprint Screenshot**
 - Clear screenshot of the completed PCB footprint layout.