

TASK 1 ABHINAY SAMALA 20368668

INTELLIGENT BATHROOM VENTILATION FAN CONTROLLER

AIM OF THE PROJECT

The aim of the project is to develop an intelligent Bathroom Ventilation Fan Controller that enhances indoor air quality and optimizes energy efficiency in residential and commercial buildings. By integrating sensors, a microcontroller, and smart algorithms, the controller will automatically adjust fan operation based on real-time occupancy and humidity levels. Additionally, the project will include the development of a user-friendly web interface to allow remote monitoring and control of the ventilation system, ensuring convenience and improved environmental conditions.

PROBLEM STATEMENT AND SOLUTION

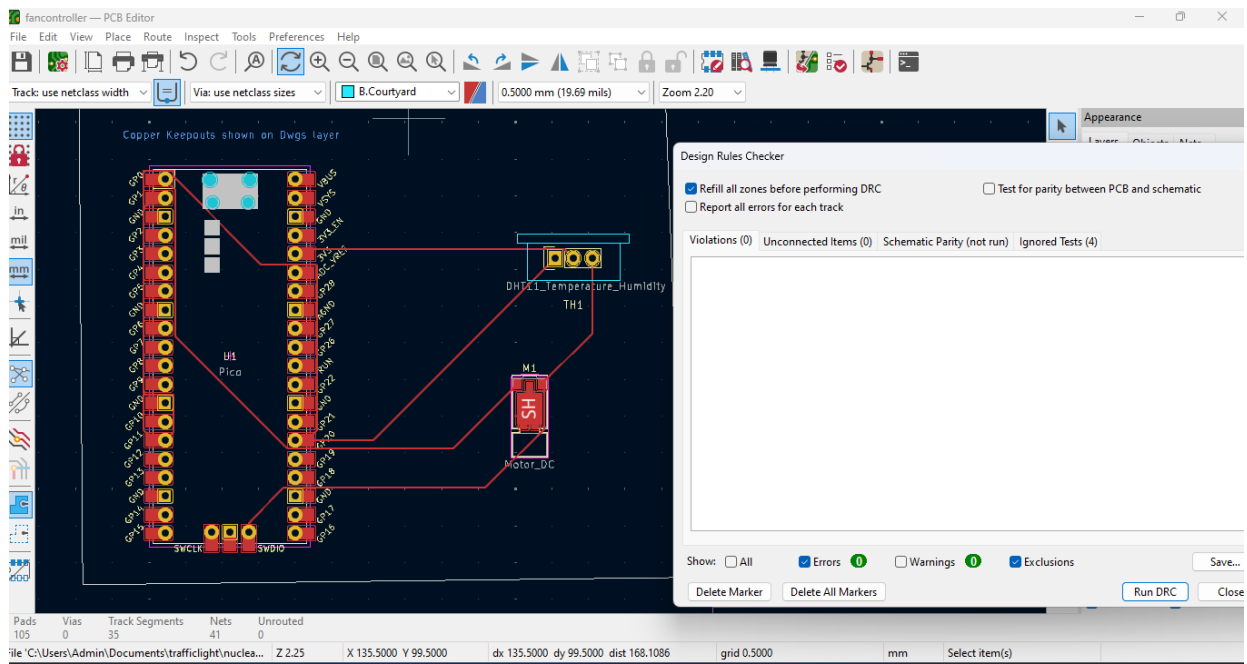
In residential and commercial buildings, maintaining indoor air quality and managing moisture levels in bathrooms is essential to prevent mold growth, structural damage, and to promote a healthy environment. Traditional bathroom ventilation fans, which are controlled manually through on/off switches, are often inefficient, leading to energy wastage and inconsistent ventilation. These manual systems fail to adjust based on real-time conditions, causing potential issues such as prolonged humidity, energy overuse, and suboptimal air quality.

The solution is to create an intelligent Bathroom Ventilation Fan Controller using a microcontroller and sensors to automate fan operation based on real-time occupancy and humidity levels. The system will continuously monitor environmental conditions and adjust the fan accordingly to optimize energy use and air quality. A user-friendly web interface will enable remote monitoring and control of the fan, providing convenience and real-time access to environmental data.

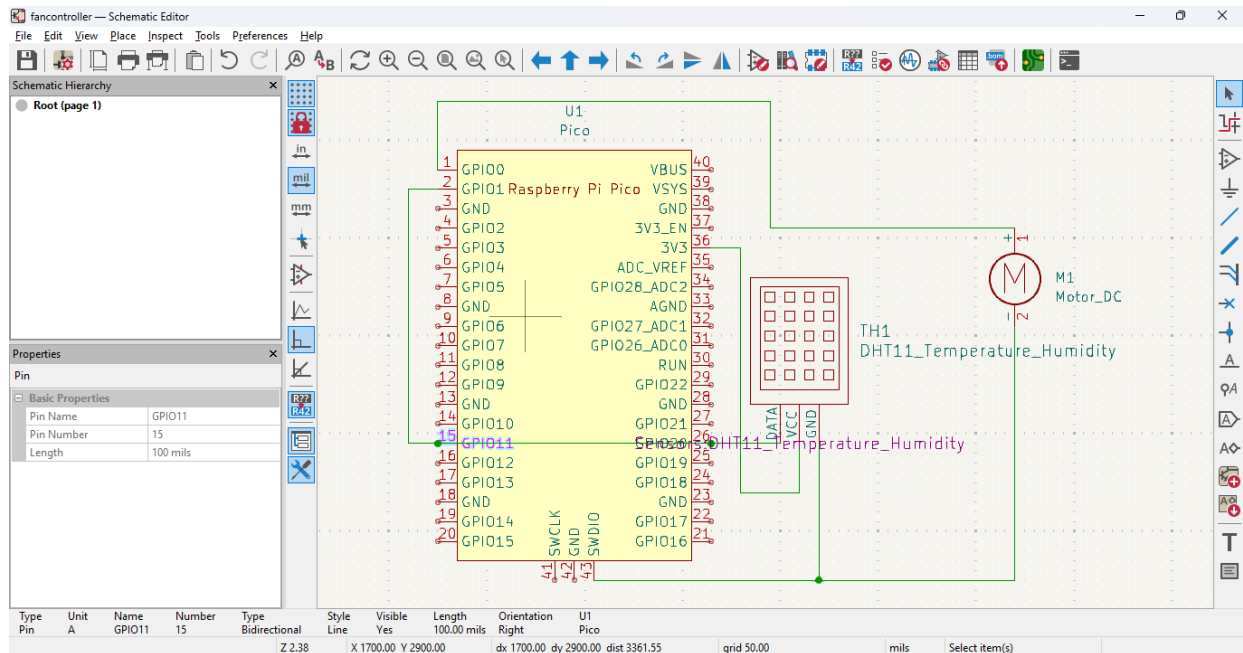
Open kicad and open preferences and manage symbol libraries
And click on folder and kicad rp pico main and click on rp pico libraries and click on mcu
raspberry pi ki cad sym file
And click on folder and kicad lib master and click on 4 lib and click on in kicad

Click on preferences and click on manage footprints libraries
And click on folder and kicad master and click on all pretty files and click ok

DRC:



WIRING DIAGRAM



PCB DESIGN

