**EAGLE Schematic Software**
Cornerstone Electronics Technology and Robotics II

- **Administration:**
  - Prayer
- **EAGLE Light Edition:**
  - The name **EAGLE** is an acronym which stands for **E**asily **A**pplicable **G**raphical **L**ayout **E**ditor.
  - The EAGLE Light Edition is offered as a freeware version at: http://www.cadsoft.de/freeware.htm
  - The following limitations apply to the EAGLE Light Edition in general:
    - The useable board area is limited to 100 x 80 mm (4 x 3.2 inches).
    - Only two signal layers can be used (Top and Bottom).
    - The schematic editor can only create one sheet.
- **Labs**:
  - This lesson is presented as a lab with step by step instructions to make a schematic with the EAGLE Light Edition.
  - Download the freeware version before proceeding to Lab 1.
  - Perform EAGLE Tutorial Lab 1 – Making a Schematic
- **References:**
  - See the web site tutorial: http://myhome.spu.edu/bolding/EE4211/EagleTutorial4.htm
  - Also see a Guided Tour through EAGLE at the CadSoft web site: http://cadsoftusa.com/Tour/tour00.htm

- **Purpose:** The purpose of this lab is to acquaint the student making a simple schematic drawing.

- **Apparatus and Materials:**

  - 1 – PC or Laptop with Internet Connection

- **Procedure:**
  - Start up the EAGLE software.
  - Click the Options/Directories and change the Project default to a location you will use for your files
  - Click on File/New/Project to create a new project.
  - Click on File/New/Schematic to open up a new schematic file within your project.
  - Use File/Save As to assign a name to the schematic.
  - Note on Backup Files: Whenever you save a library, schematic, or PCB design, EAGLE copies the previously-saved design to a backup file (e.g. led_flasher.s#1 for led_flasher.sch). Up to ten previously-saved versions are kept, named #1 through #9. These files can help if you make a mistake and need to revert to an earlier version.
  - Right click on the light gray tool bar at the top of the window. In the menu that appears, Actions, Parameters, Command buttons, and Command texts should be checked.
  - This lesson will develop the following LED Flasher schematic:

LED Flasher
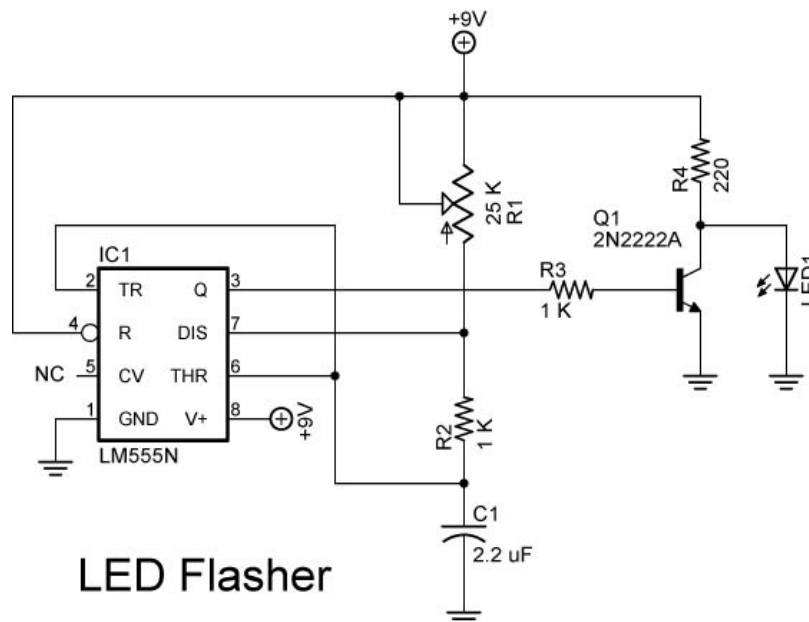
o **Adding Parts to the Schematic:**
  - From the Command buttons on the left side of the screen, click on the Add button ⬚ to select a part.
  - 555 Timer: Select the LINEAR library and then choose LM555N from the 555 Timer part list library. Left click twice on the part to select the part, and then left click once to place the part. You may rotate the part around by right-clicking.
  - Potentiometer R1: From the POT library, select TRIM_US. We then selected the TRIM_US-B25U package. Left click the TRIM_US-B25U twice and place it on the schematic.
  - Resistor R2: From the RCL library, select R-US. Double click on R-US_0204/7 and place the resistor R2 on the schematic.
  - Capacitor C1: From the RCL library, select C-US. Double click on C-US025_025X050 and place the capacitor C1 on the schematic.
  - To this point, you should have something that looks like this:
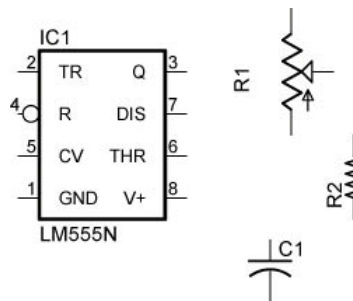
o **Resizing the Schematic:**
  - If the drawing becomes too crowded, click on the Out button in the Action tool bar at the top of the window. Add an additional part to the schematic, then click on the Fit button to have the drawing fit the screen.
o **Deleting Parts:**
  - Select the Delete button and left click on the part to be deleted.
o Adding the Remaining Parts:
  - Transistor Q1: From TRANSISTOR-NPN library, select 2N2222 and then double click on 2N2222A. Place the transistor in the schematic.
  - LED1: From LED library, select LED and then select LEDSFH486. Place the LED1 in the schematic.
  - +9V: From SUPPLY2, select +9V and add it to the schematic.
  - Ground: From SUPPLY2 select PE and add it to the schematic.
  - Click on the Stop Sign 🛑 to go back to select mode.

- o **Copying Parts:**
  - ▪ Resistors R3 and R4: Select the Copy button and left click once on R2. Move the new resistor to the side and left click again making a copy, R3. Click on R3 to make a copy which becomes R4.
  - ▪ +9V: Copy +9V symbol and place it close to pin 8 of the 555 timer.
  - ▪ Ground: Copy the ground symbol three times and place one by in 1 of the 555 timer, one under Q1 and the other under LED1.
- o **Moving Parts:**
  - ▪ Your schematic should have all of the following parts: The parts should be in these approximate positions. To move parts around, select the Move button and left click and hold on the part. Drag the part to the new position and release.
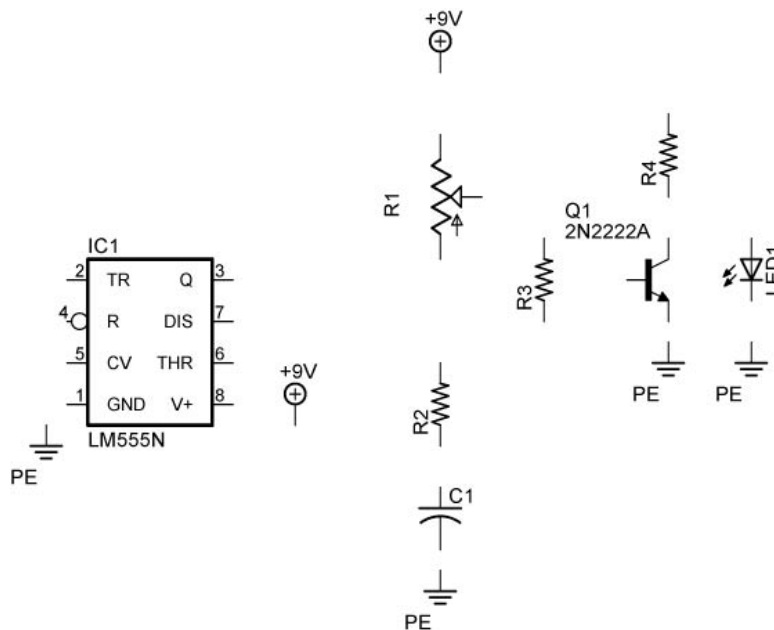
- o **Rotating Parts:**
  - ▪ Rotate R3: Select the Rotate button from the tool menu and then left click on R3. Repeated clicks on R3 will rotate R3 an additional 90 degrees. Rotate R3 into the horizontal position. When using the Rotate tool, place the cursor over the + in the part to be rotated.
  - ▪ Rotate +9V: Rotate the +9V adjacent to pin 8 of the 555 timer. The line (wire) should face pin 8.
- o **Creating a Mirror Image of Parts:**
  - ▪ Mirroring R1: Select the Mirror button and single click on the + in R1. This flips R1 creating the mirror image.
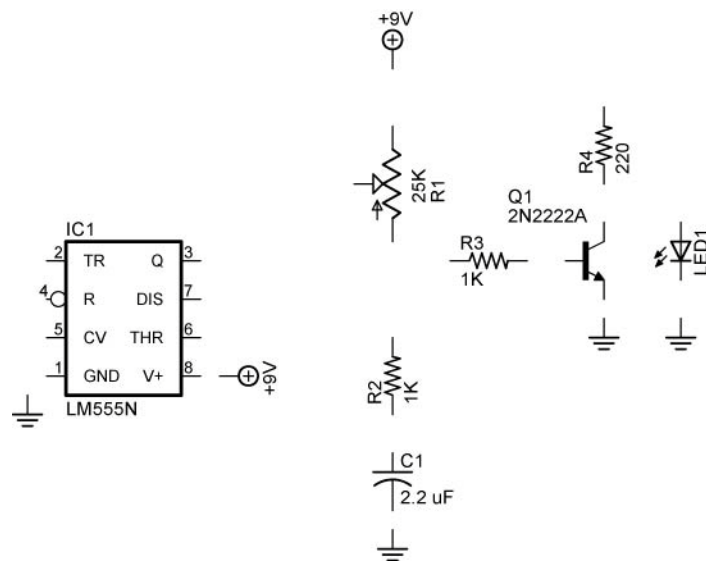
- o **Adding Values to Parts:**
  - Add values to R12, R2, R3, R4, C1, PE: Select the Value tool button, click on the part and then assign the new values shown below:

 | | |
    |---|---|
    | R1 | 25K |
    | R2 | 1K |
    | R3 | 1K |
    | R4 | 220 |
    | C1 | 2.2uF |
    | Grounds | When the warning comes up and asks you "Do you want to change it anyway?" answer "Yes". In the Value window, delete the PE and press "OK". |

- o Final Parts Layout:

- o **Moving Groups of Parts:**
  - If you want to move a group of parts, select the Group button and outline the group of parts that must be moved. The parts selected will change color. Now select the Move button and right click on the group of objects to be moved and relocate to the new position.
- o **Connecting the Parts:**
  - To draw nets (wires) in, select the Net button (this is the one on the right side of the toolbar). Do not use the wire function – it just draws lines, but does nothing electrically! Left-click to begin drawing; click again when the net is done. If the Net function does not place the junction dots at connections, use the junction button to add the connections. Use the stop sign to end the Net function.

o **Adding Text:**
  ▪ Click on the Text button and the parameters appear on the Parameters tool bar. Type in the desired text in the Text window and press "OK". At this point you can change the parameters such as size or orientation. Add the text to the schematic. This completes the schematic LED Flasher:

LED Flasher
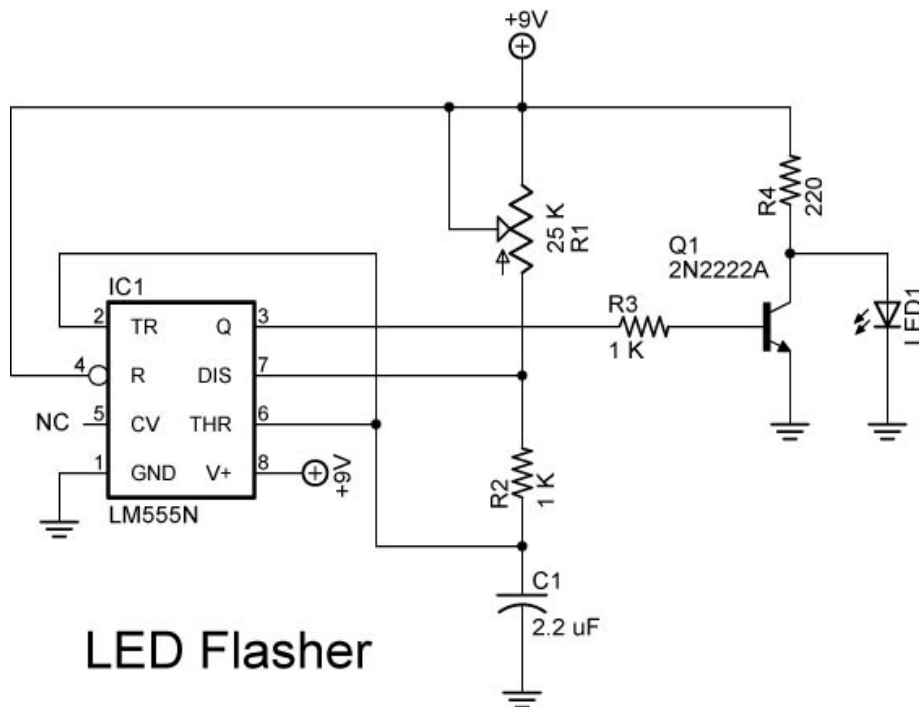
o **Grid System:**
  ▪ Grid is listed in the Command Text menu on the right side of the screen. Click on Grid which activates the Grid window. From the Grid window you can make adjustments to the grid properties such as Display, Style, Size, etc.