Eagle Schematic Capture Tutorial

Introduction

In this project, we will learn about graphical representation of circuits called schematics. A short tutorial will be presented to introduce the students to basics of schematic capture.

Discussion Overview

When designing a circuit that is to be eventually turned into a circuit board, it is much more convenient and easier to understand when the design is represented graphically. For example, for a simple LED circuit with a current limiting resistor, one might have to create the following verbal instructions:

- The design used a 9V battery, 1KΩ resistor R1, and an LED D1
- Connect the positive side of the battery to one side of R1
- The other side of R1 is connected to the positive lead of D1
- The negative side of D1 is then connect back to the negative side of the battery
- No other connections exists in this schematic

As one can see, a verbal description can easily become cumbersome and very complex for even a slightly more complicated circuit.

Graphical representations, called schematics, can convey the same information in a much more succinct way. For the circuit above, for example, the following schematic captures the information simply and without any ambiguity.
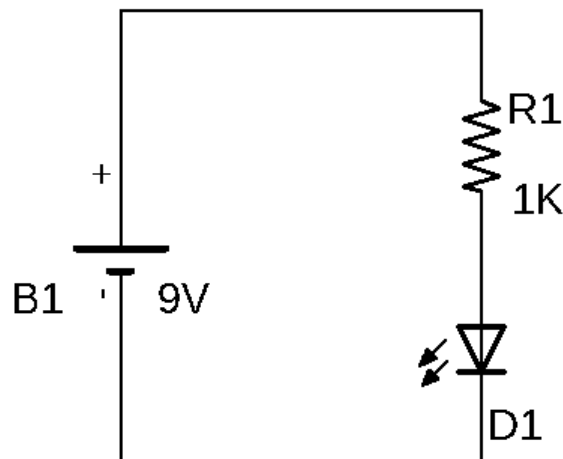

Figure 1 - LED Circuit Schematic

One of the popular and freely available schematic capture programs is Eagle by Autodesk. In the rest of this worksheet, we will go through a simple tutorial of Eagle commands to capture the schematics for the design of your choosing.

Procedure

The following steps will guide you through capturing the schematic for the Electronic Piano circuit shown below.
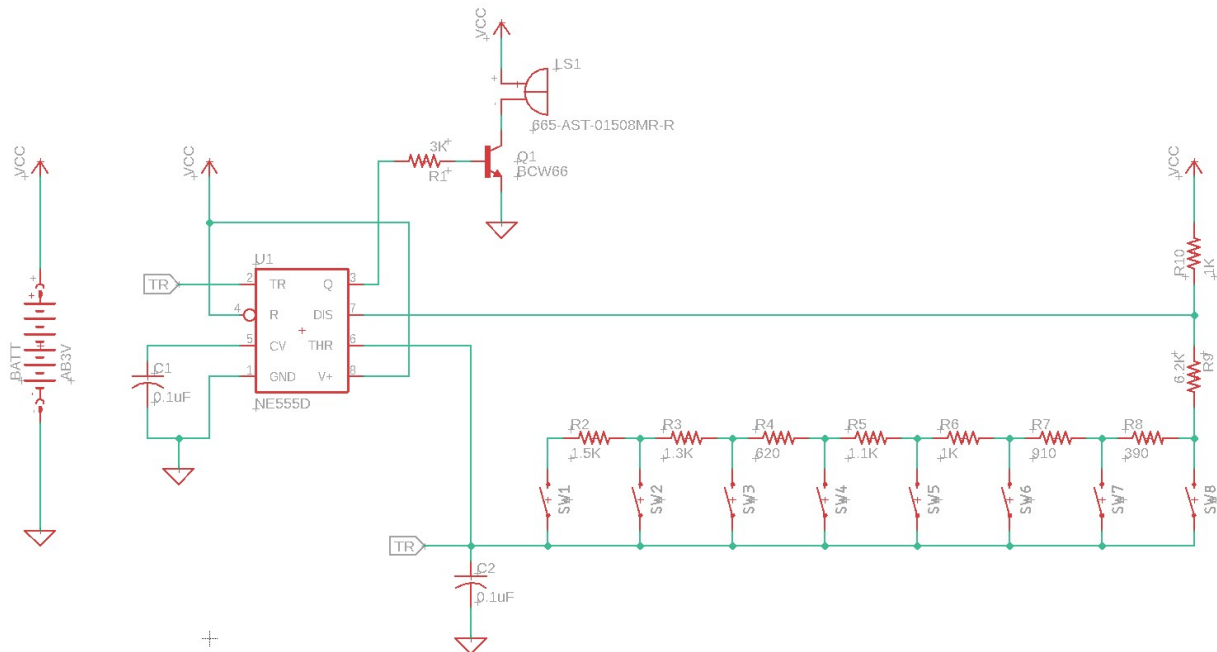

Figure 2 – Electronic Piano Schematic

1. On Github site, fork https://github.com/League-EE/EE-Workshop to your account and then clone it on your desktop.
2. Open the EE-Workshop schematic in "EE-Workshop/Lesson 4" folder.
   a. This will launch the Eagle schematic capture program and load the EE-Workshop schematic.
   b. The schematic has the battery symbol already placed for you. You will go through placing the rest of the components for your design.

3. Placing parts

    a. To place a part, click on the "Add Part" icon ⊕ ▾, or choose "Add…" from the "Edit" menu.

    b. The "ADD" window displays the available libraries to choose parts from.
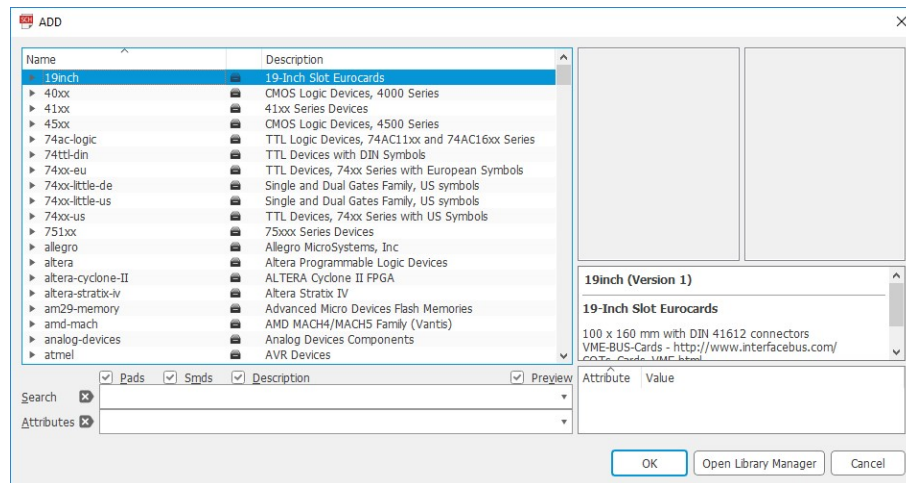
    c. In the "Search" field type "NE555*". This will bring up all the parts starting with "NE555".

Hint: You can use the wildcard "" for any combination of characters at the beginning, middle or end of your search string. For example, to search for all devices that have the string "555" in their names, type "*555*" in the search window.*

    d. Under the "linear" library, click on *555 folder, select NE555D and click "OK".
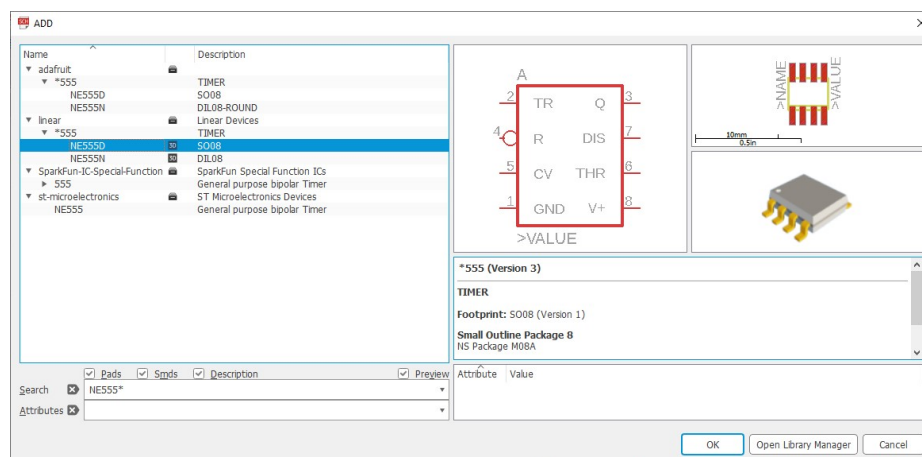
    e. This will select the NE555D part for you and allows you to place it in your schematic.

f. Place the part on your schematic as seen below.
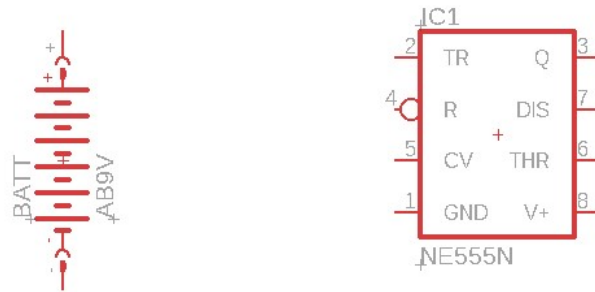
You will notice that after placing the part, the tool will automatically show another part of the same kind ready to be placed. This will be very useful when you are placing multiple of the same kind, for example, resistors. For this part, press "Esc" to end placement of the NE555N part. The tool will return to the "ADD" window.

g. Clear the search by clicking on the "x" next to "Search".

h. This time we will search for a resistor to place by browsing through the libraries

    i. Scroll down to the "rcl" (Resistor, Capacitor and Inductor) library and click on the small triangle next to it to open it.

    ii. Scroll down to "R-US_" library and click to open it.

    iii. Select the part "R-US_0805" and click OK to select it for placement.

i. You will notice that resistor symbol is in a horizontal configuration. Right click to rotate the symbol by 90°.

j. Place as many instances of the resistor as you need on your schematic by left clicking the mouse at the desired location. Your schematic should look like the one below.

k. Place the rest of the parts in your schematic following the steps outlined above.

  i. Consult the parts list in **Error! Reference source not found.**"Appendix B – Parts List" to look up part names and libraries.

  Adding a Library
  When placing parts, if you cannot find the library you are looking for, you can add it by following the steps below

  1. From the add parts "ADD" window, click on "Open Library Manager".

  2. In the "Library Manager" window, click on the "Available" tab.

  3. Search for the library you are looking for, highlight it and then click on "Use". This will make the selected library available for use in your design.
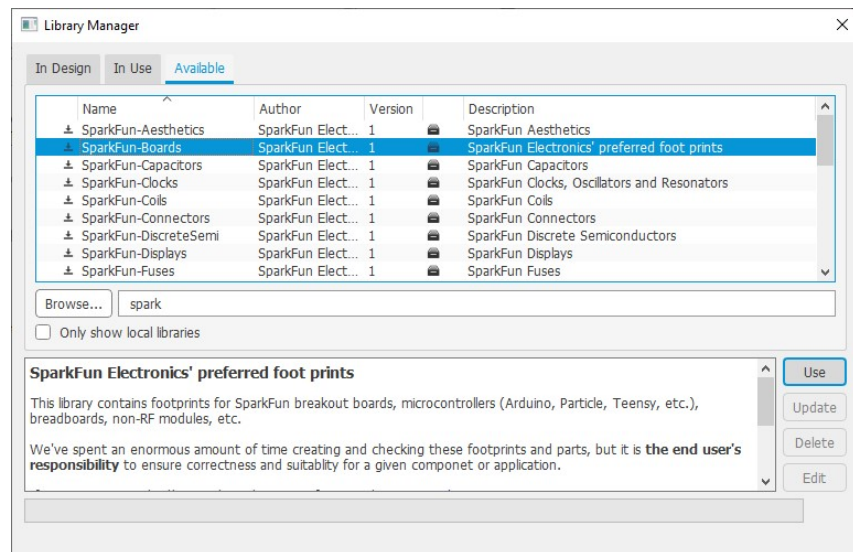

  ii. After placing all your parts, the part placement on your schematic should look similar to the placement seen in Figure 2.

4. Setting Values

  a. In order to change the value of a part in your schematic, you can either choose the "Value…" item from the "Edit" menu, click on the "Value" icon , or right click on a part and select "Value" from the dropdown menu.

b. To change the value of R1, click on the "Value" icon and then click on R1.

   i. This will open the "Value" window where you can enter the desired value:
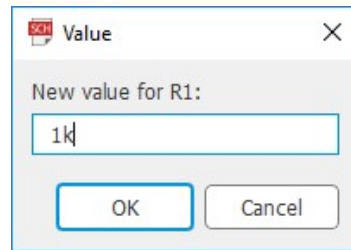
    ii. Enter the value 1k, and click on OK.

c. Change the values for all the parts in your schematic to those specified in Figure 2.

5. Connecting the parts

a. In order to connect the parts on the schematic with nets, either select "Net" from the "Draw" menu or click on the "Net" icon ⌐.

b. To connect a net to a part, click on the lead that you'd like to connect a net to. This will create a connection to the lead that was just clicked on. (In our example below, we are connecting a net from pin 5 of U1 to the top lead of C2.)
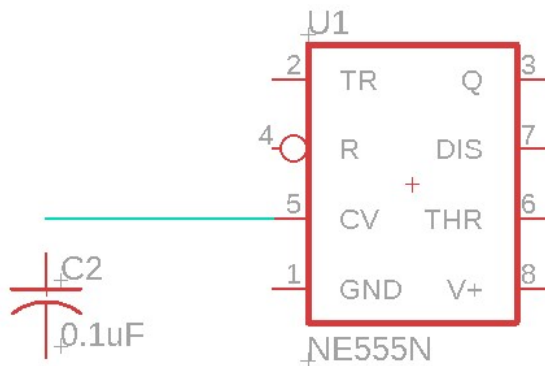
c. Drag the other end of the net to a location right above C2 and click. This will create a junction point where you can move the net in the direction perpendicular to the original direction.
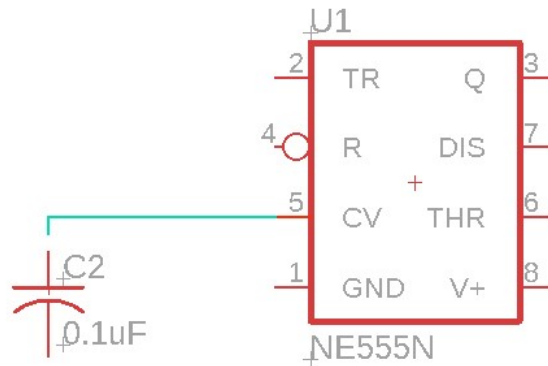

d. Once you have dragged the net to the part that it needs to be connected to, click on the lead that the net should be terminated on. This will end the current net on the lead just clicked on.
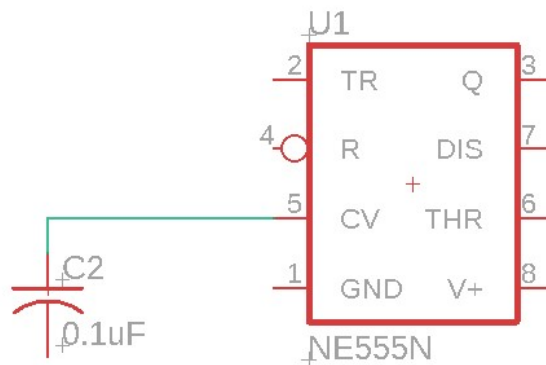
e. Complete connecting all the nets for your schematic. Your final schematic should look similar to the one in Figure 2.

6. Adding GND

a. Adding a ground to your schematic is the same as adding a part.

b. Click on the "Add Part" icon and enter "GND" in the search field.

c. Select the GND from the "ngspice-simulation" library and place it on your schematic.

d. Connect the ground to the appropriate nets as indicated in your design.

e. When connecting ground, the tool will ask if you are intending to merge the current net name for the selected net with "GND". (See the example below.)

Warning ✕

Merge net segment 'N$9' into supply net 'GND'?

Yes　No
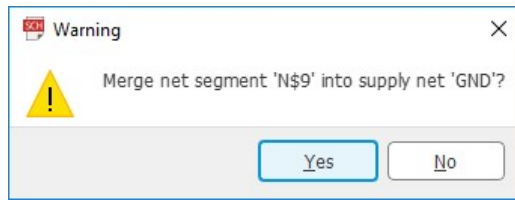
f.　Click "Yes" to complete the process.

7.　Adding Labels and Cross References (Xref)

a.　A "Cross Reference" helps with connection of nets on different pages. Even though our schematic has only one page, we will add a cross reference to show connectivity between two nets on the same page.

b.　To add a cross reference, first either select "Label" from the Draw menu or click on the "Label" icon .

　i.　Click on the "Xref On" icon on the ribbon above the schematic. This will turn on the cross reference graphics for the net names (labels).

　ii.　Click on the net that you'd like to add a label to. For our example, we click on the "TR" pin of "U1".

　iii.　This will display the current name of the net that you can drag around and place on the schematic.

　iv.　Place the Xref at the end of the net as shown in the example below. (If you need to rotate the Xref, repeatedly right click on the mouse.)
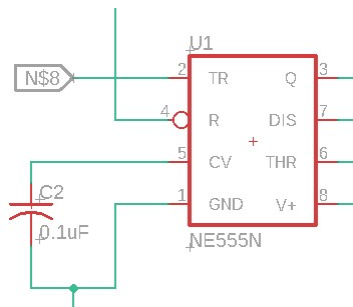

　v.　Right click on the "+" reference point of the Xref and click on "Name" from the dropdown menu. This will open a dialogue window that will allow you to change the net name. Change it to "TR" and click "OK".

vi. Now the Xref should display "TR" as shown below.
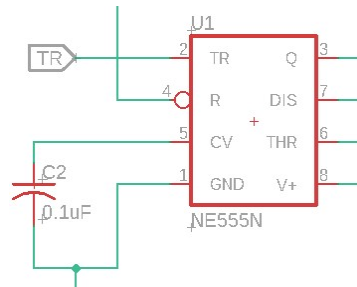
vii. Add another cross reference with the same name "TR" to the net connected between C1 and pin 6 of U1 as shown below.

1. The tool will ask you whether you would like to merge the current name with "TR". Click on "Yes".
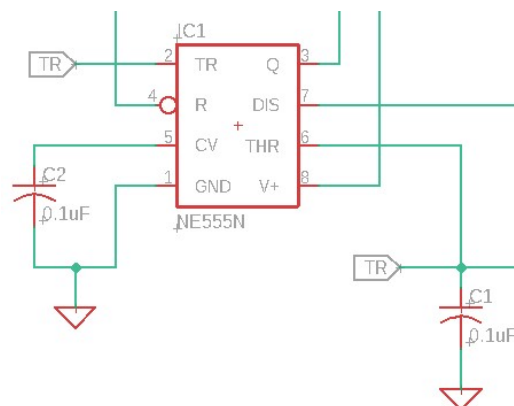
viii. The two nets labeled "TR" are now treated as the same net by the tool.

ix. Add any other Xref indicated in your design.

Your final schematic should look similar to the one in Figure 2.

Make sure you commit and push your final files to github.

Appendix – Useful Links

Schematic capture in Eagle:
https://www.youtube.com/watch?v=1AXwjZoyNno

Laying out a board in Eagle:
https://www.youtube.com/watch?v=CCTs0mNXY24

Appendix B – Parts List

Part	Value	Device	Package	Library
BATT	AB9V	AB9V	AB9V	battery
C1	0.1uF	C-USC0805	C0805	rcl
C2	0.1uF	C-USC0805	C0805	rcl
IC1	NE555D	NE555D	SO08	linear
LS1		F/HMB	F/HMB	buzzer
Q1	BCW66	BCW60-NPN-SOT23-BEC	SOT23-BEC	transistor
R1	3K	R-US_R0805	R0805	rcl
R2	6.2K	R-US_R0805	R0805	rcl
R3	1K	R-US_R0805	R0805	rcl
R4	390	R-US_R0805	R0805	rcl
R5	1.5K	R-US_R0805	R0805	rcl
R6	1.3K	R-US_R0805	R0805	rcl
R7	620	R-US_R0805	R0805	rcl
R8	1.1K	R-US_R0805	R0805	rcl
R9	1K	R-US_R0805	R0805	rcl
R10	910	R-US_R0805	R0805	rcl
SW1	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	TACTILE_SWITCH_SMD_6.2MM_TALL	SparkFun-Switches
SW2	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	TACTILE_SWITCH_SMD_6.2MM_TALL	SparkFun-Switches
SW3	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	TACTILE_SWITCH_SMD_6.2MM_TALL	SparkFun-Switches
SW4	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	TACTILE_SWITCH_SMD_6.2MM_TALL	SparkFun-Switches
SW5	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	TACTILE_SWITCH_SMD_6.2MM_TALL	SparkFun-Switches
SW6	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	TACTILE_SWITCH_SMD_6.2MM_TALL	SparkFun-Switches
SW7	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	TACTILE_SWITCH_SMD_6.2MM_TALL	SparkFun-Switches
SW8	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	MOMENTARY-SWITCH-SPST-SMD-6.2MM-TALL	TACTILE_SWITCH_SMD_6.2MM_TALL	SparkFun-Switches