TEAMS Academy AT&E: A beginning tutorial to Eagle Schematic Editor

Our goal today

Your goal is to use Eagle to create the following circuit diagram
(a.k.a., schematic) on the attached sheet.

Starting a new project

1. First open a new project in the **Control Panel** window using the
pull-down menus on the top toolbar as shown to the right. Create a
unique project name that starts with your last name.

- Note that the **Control Panel** window contains several
  directories. The most important are the Projects directory (you
  can see several of mine to the right) and the **Library** directory,
  which contains thousands of components organized in
  subdirectories by manufacturer or function.

2. Next create a **new schematic** in your project folder using similar
pull down menus (**File, New, Schematic**). An empty **schematic**
window will appear, waiting for you to create your circuit diagram
using components from the **library**. Before moving on, make sure to
save your work (**File, Save All**). Be sure to save your work every 10
minutes or so.
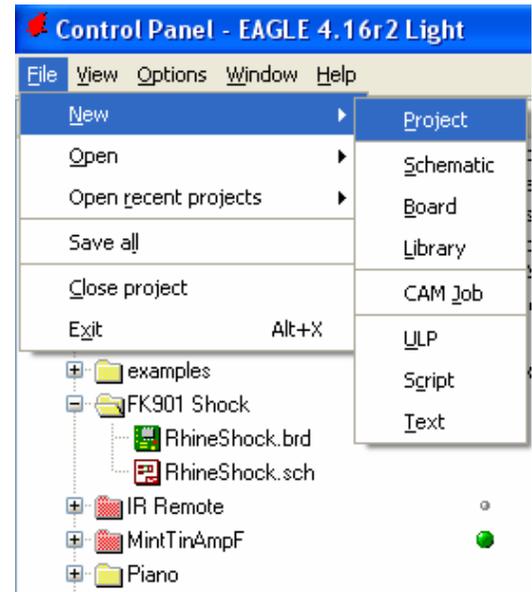
Creating the circuit schematic

Here are the toolbars you will use in the **Schematic** window, with some of the important buttons highlighted:

Info – get useful info about any object	
Move an object or group – can also **Rotate** with right click	
Mirrors an object. Useful to untangle diagram. Does **not** mirror device!	
Selects a group of objects (then you can **Move**, **Copy**, **Format**, etc.)	
Delete a component	
Name –adds or changes the name of a component	
Split: creates a bend in a wire. **Wire**: click once to start a wire, double click or escape to end.	

Open, Save, Print

Fit, In, Out, Refresh, Selection - Use to zoom window in and out

Board - Use to switch between schematic and board views

Show – click on wiring to see if connects are complete/correct

Copy

Rotate – it's easier to rotate using right mouse button with **Move** button

Change – changes text, size, and other attributes of an object

Cut & **Paste** tools – a bit tricky to use! May need right button click.

Add – adds new component from library; escape key will pop you back into library for next component selection.

Value –adds or changes a value, e.g. 100 uF, to a component

Smash tools "separates" the **name** and **value** from a component so you can **move/rotate** them independently and clean up your schematic.

Text – adds text (titles, dates, your name, etc.) to your schematic

Junction – create a "solder ball" to join two wires or a wire to a component. Be consistent in your diagrams and use junctions!

3. **Selecting and Placing Components**: Try adding a new part using the **Add** button on the toolbar. We provided a complete list of components you need on the attached sheet. Let's practice by adding a **three** 555 timers (you can delete them later).

a. <u>Using the search field to find a part</u>: When searching for a part use * expand your search. For instance, when looking for a 555 timer, you should type in *555* to get all parts with a 555 in their name. The * is called a "wildcard" and it acts as a substitute for a string of characters of any length. You may also use the ? as a wildcard to substitute for a single character. To clear your search, delete the search term and hit **OK**.
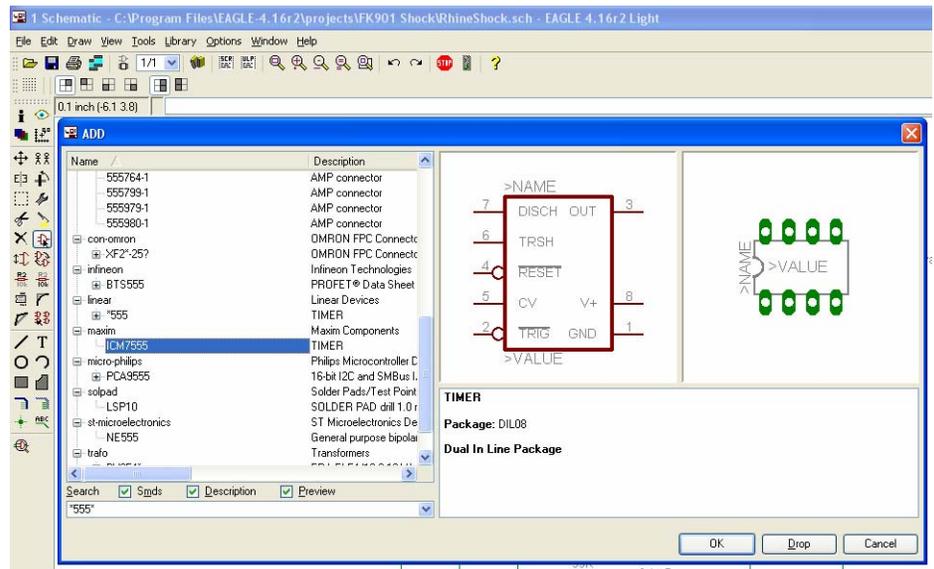
b. You can scroll through the various parts to see which matches. You should adjust your search window as shown to the right to clearly view the logical symbol (middle diagram, reddish-brown, this is what shows up on your schematic) and the physical device (in gray/green, this is a drawing of the actual device that you will solder to your board – make sure it is the correct package!).

c. After you find the correct component, choose **OK** to select the part, but **DO NOT SELECT DROP**! This removes the component From the library and you cannot select it again. If you do drop the library by mistake, you can click into the Control Panel window, right click on the "Libraries" folder, and select "Use All."

d. You can add multiple copies of an object by <u>left clicking</u> over and over again (try it). You may also <u>right click</u> to rotate the object.

e. When you are done, use the ESCAPE key – you'll jump back to the library to look for the next component

f. You can use the zoom toolbar to navigate around the schematic. Click on the left button to auto center the schematic.

4. **Wiring**:

a. After placing all of the parts, you can begin connecting the components together. Use the **Wire** tool to connect your parts. <u>Left click</u> to start the wire and <u>double click</u> or <u>escape</u> to end the wire.

b. You can use the **Split** button to create a bend in a wire.

c. After connecting the wires place **Junctions** at where the wire connects to the part. Don't forget these; they are important to make the connections.

d. **IMPORTANT**: Use the **Show** button to ensure that wire connects to each component! Or drag a component to see if wires move with it!
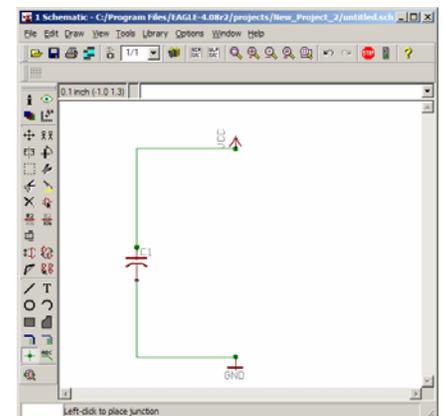
5. At this point you should **Name** all of your components and add correct **Values**. Eagle with auto-name components as you add them, but feel free to adjust the names and values. You might also want to try the **Smash** tool to give you more flexibility with the arrangements of names & values on your schematic.

6. Finally, use the **Text** tool to add a title, your name, date, etc. to the schematic. You can try using the **Change** button to modify the size of the text.

Circuit diagram for the FK901 Electric Shock Low Power Circuit, by Future Kit:
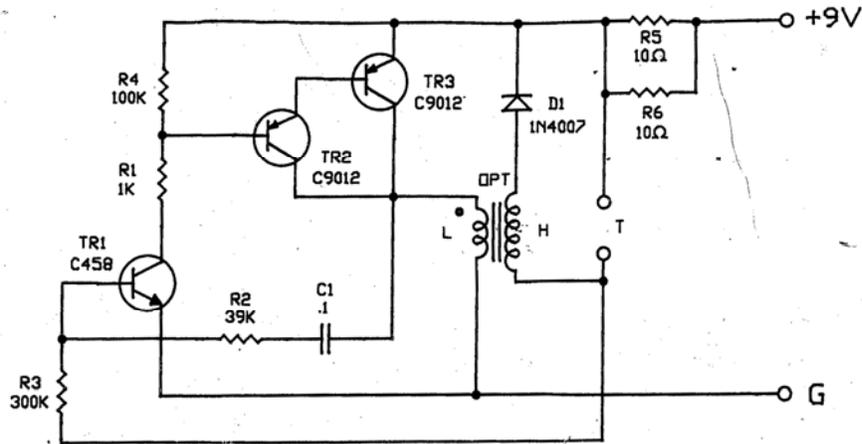

```
Partlist exported from C:/Program Files/EAGLE-4.16r2/projects/FK901 Shock/RhineShock.sch at 3/24/2008 07:48:30a
```

Part	Value	Device	Package	Description
BATTERY	9V	AB9V	AB9V	9-V BATTERY CLIP
C1	0.1 uF	C-US050-025X075	C050-025X075	CAPACITOR, American symbol
D1	1N4004	1N4004	DO41-10	DIODE
Q1	2N3906	2N3906	TO92D	PNP TRANSISTOR
Q2	2N3906	2N3906	TO92D	PNP TRANSISTOR
Q3	2N3904	2N3904	TO92D	NPN TRANSISTOR
R1	1.0K	R-US_0204/7	0204/7	RESISTOR, American symbol
R2	39K	R-US_0204/7	0204/7	RESISTOR, American symbol
R3	300K	R-US_0204/7	0204/7	RESISTOR, American symbol
R4	100K	R-US_0204/7	0204/7	RESISTOR, American symbol
R5	5 Ohm, 1/2 watt	R-US_0617/22	0617/22	RESISTOR, American symbol
THUMB1		WIREPAD1,6/0,8	1,6/0,8	Wire PAD connect wire on PCB
THUMB2		WIREPAD1,6/0,8	1,6/0,8	Wire PAD connect wire on PCB
TR1		U15-TRANSFORMER	U15	TRANSFORMER

Introduction to Eagle

March 24/25, 2008

Finis!

- Here's the final version we came up with in Eagle.

- What changes did we make from the original?

- Why did we make those changes? Are there any functional differences?