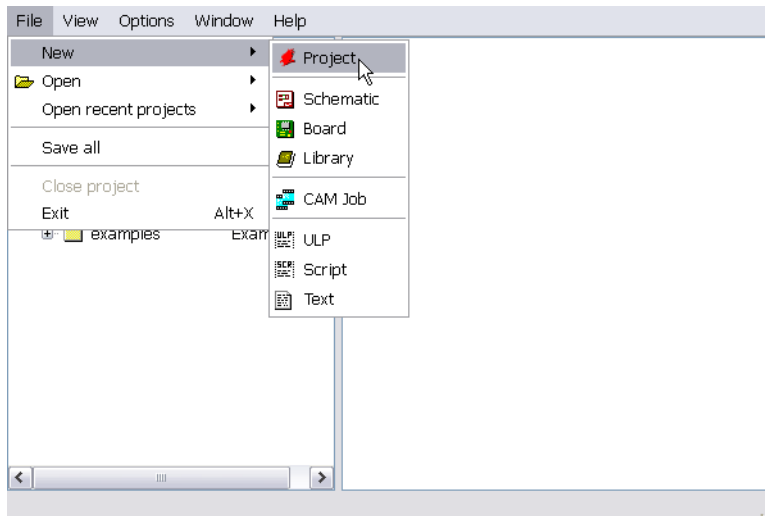
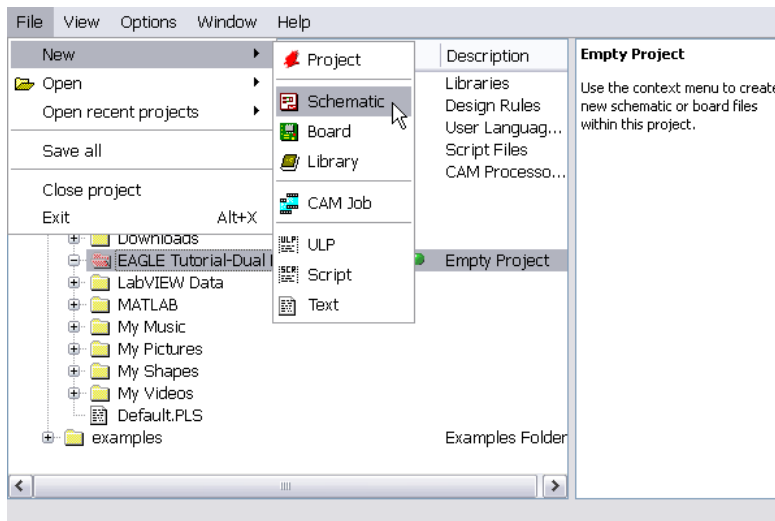


This tutorial will introduce the basic operations available with the EAGLE (Easily Applicable Graphical Layout Editor) PCB software. This tutorial is based on EAGLE Lite version 5.10.0 for Windows. The latest version is available at <http://www.cadsoft.de/download.htm>

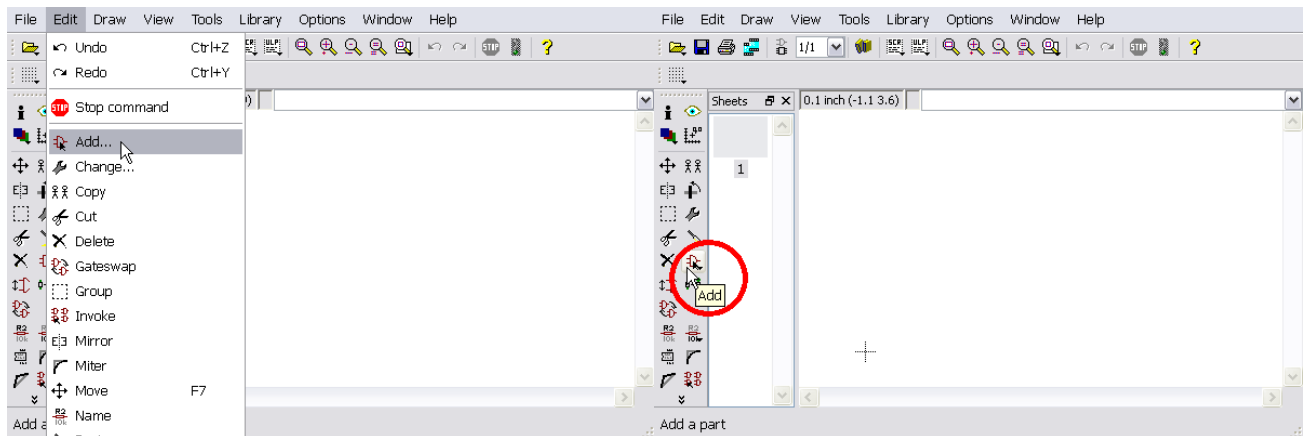
Once you have the latest version of the EAGLE software downloaded & installed, launch the program to create your new project:



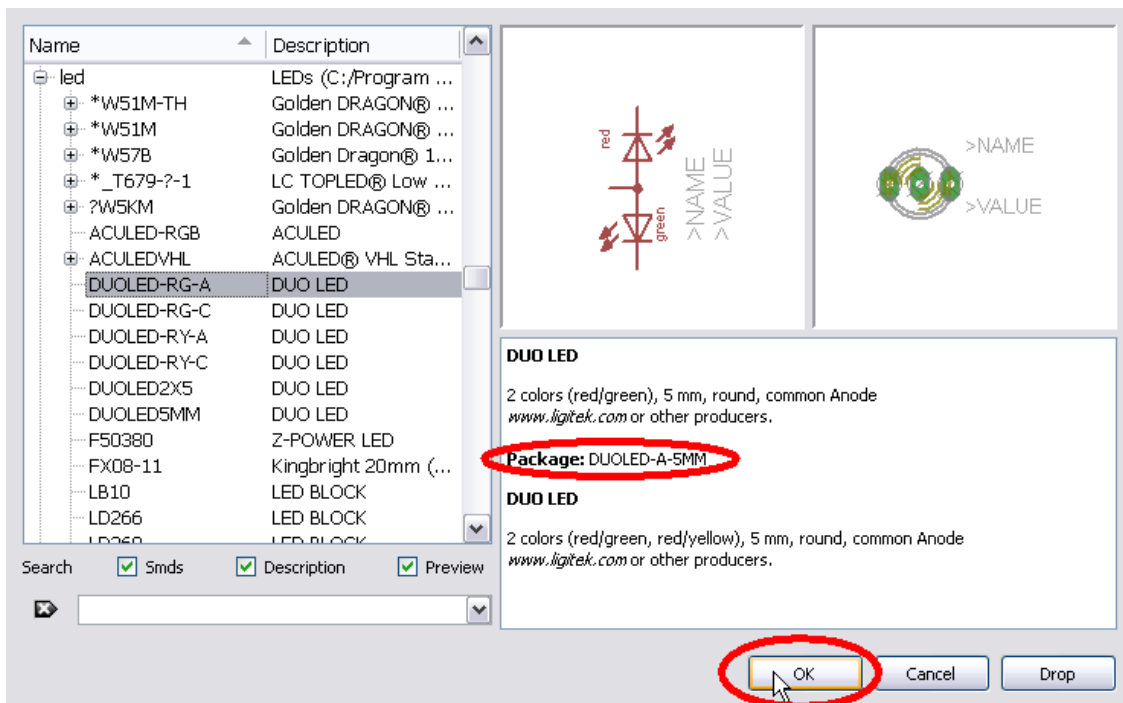
For this tutorial, the project will be named “EAGLE Tutorial-Dual LED blinky thing”. The next step is to create a schematic in the new project, by default the new project is already open, if adding additional schematics, be sure that the proper project is open before adding new schematics from the File Menu (File—New—Schematic):



The schematic editor window will open, this is where the circuit schematic will be entered. Parts can be added to the schematic by either using the Edit Menu (Edit—Add ...) or by using the Add button as shown in the following screenshots:



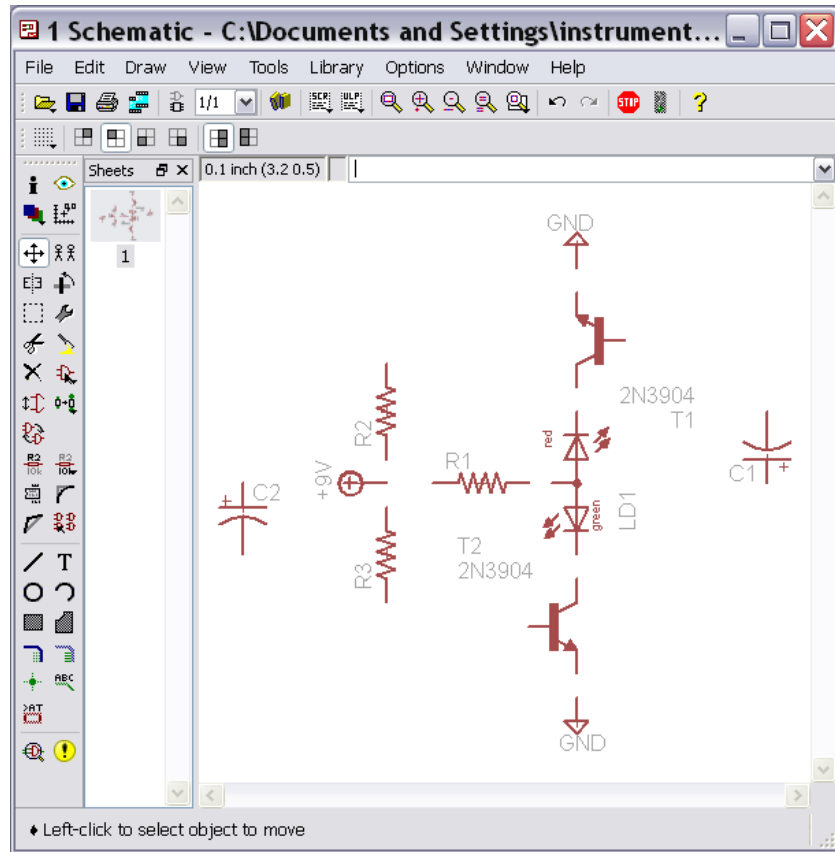
Once the ADD selection box is visible, add the DUOLED-RG-A dual color, common anode LED by selecting it and then placing it in the schematic (led==DUOLED-RG-A):



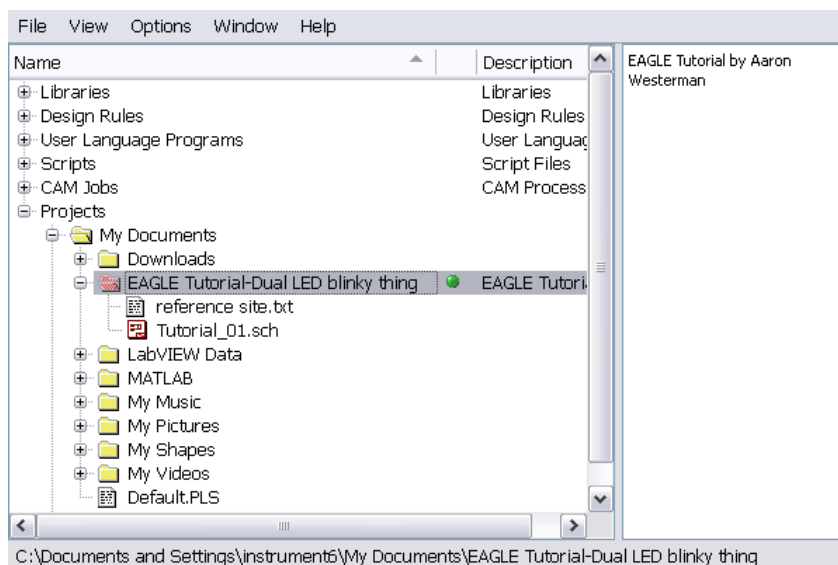
Continue to add:

- 2 – 2N3904 transistors (transistor==2N3904) – T1, T2 on schematic
- 2 – 10 $\mu$ F capacitors (rc1==CPOL-US==CPOL-USE2-5) – C1, C2 on schematic  
Based on Mouser P/N 647-UKW1J100MDD
- 1 – 390 $\Omega$  resistor (rc1==R-US\_== R-US\_207/10) – R1 on schematic  
Based on Mouser P/N 30BJ250-390
- 2 – 100K $\Omega$  resistor (rc1==R-US\_== R-US\_207/10) – R2, R3 on schematic  
Based on Mouser P/N 30BJ250-100K
- 1 – +9V supply (supply2==+9V)
- 2 – ground connections (supply2==GND)

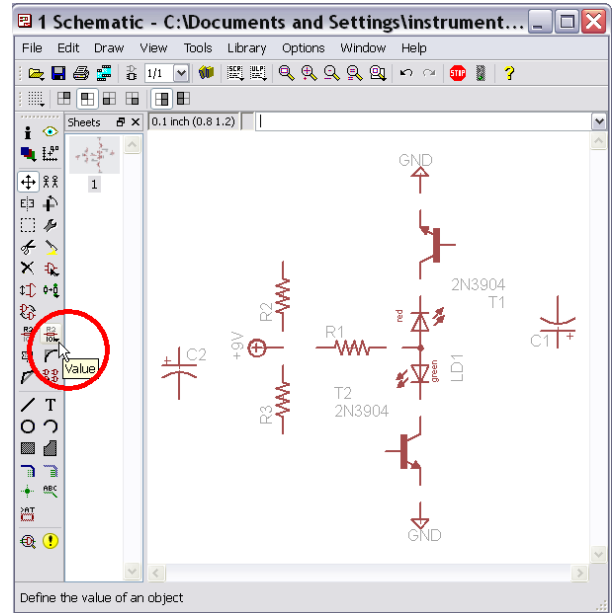
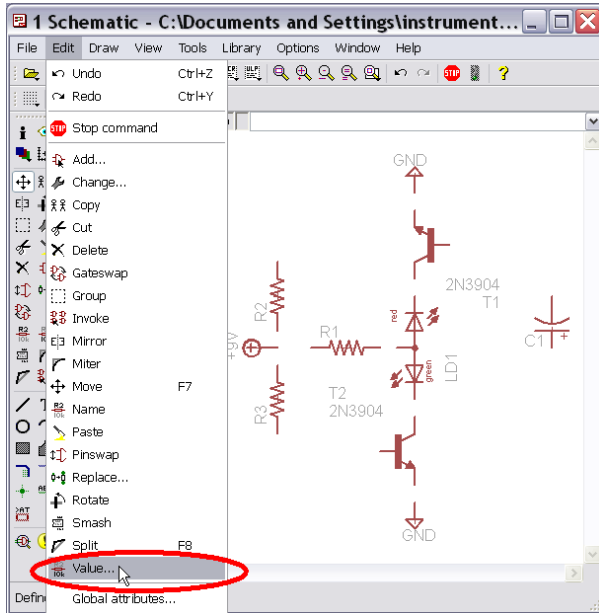
Now the schematic window should look something like this (+9V, R2, R3, C1, T1 and one ground connection have been rotated 90°, the reason will soon become clear):



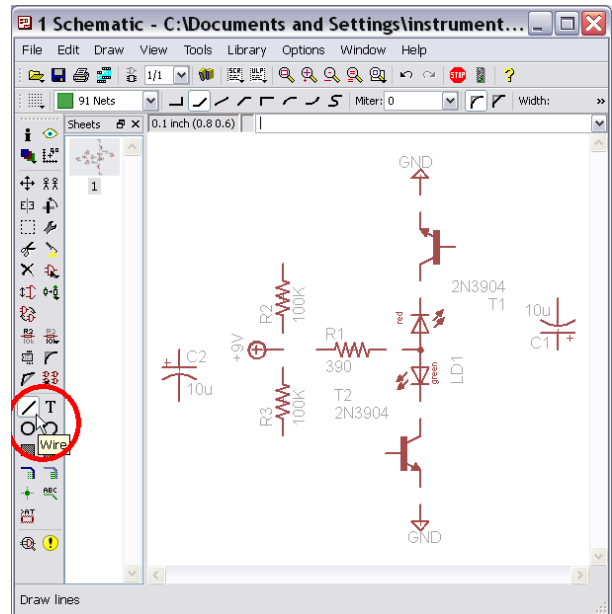
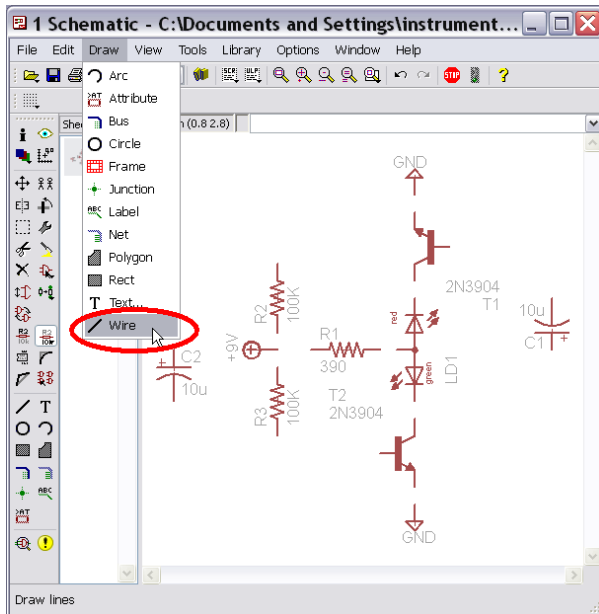
This would be a good time to go ahead & save your schematic (I chose Tutorial\_01 for my filename). If you view the EAGLE Control Panel it should show your schematic under your project (you can also Right-click to edit the description information for the project if desired).



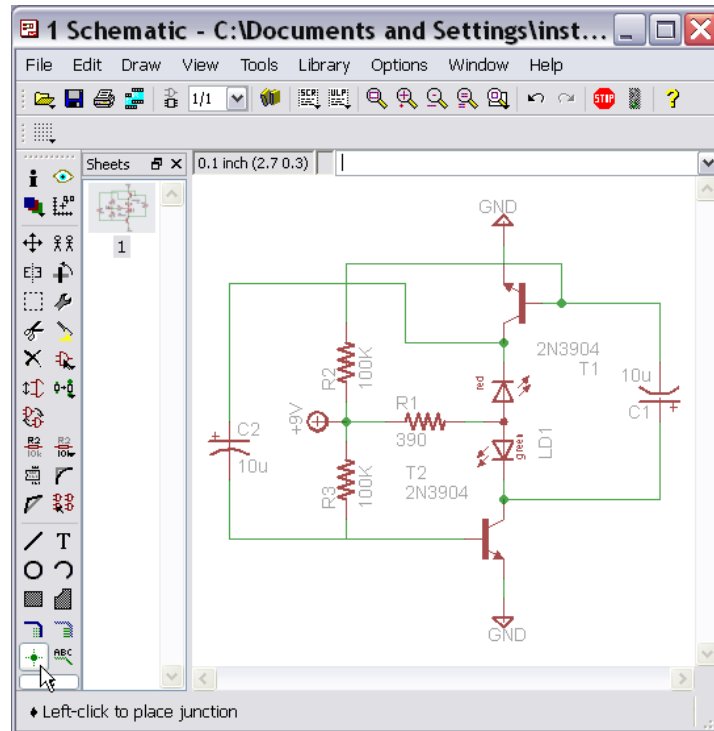
We know the values for the components we used in this circuit, but in a more complex circuit multiple circuit components of the same type may have several different values. To keep this correct in our circuit schematic as well as circuit board layout the values need to be defined for each component. This can be accomplished by either using the Edit Menu (Edit—Value ...) or by using the Value button as shown in the following screenshots:



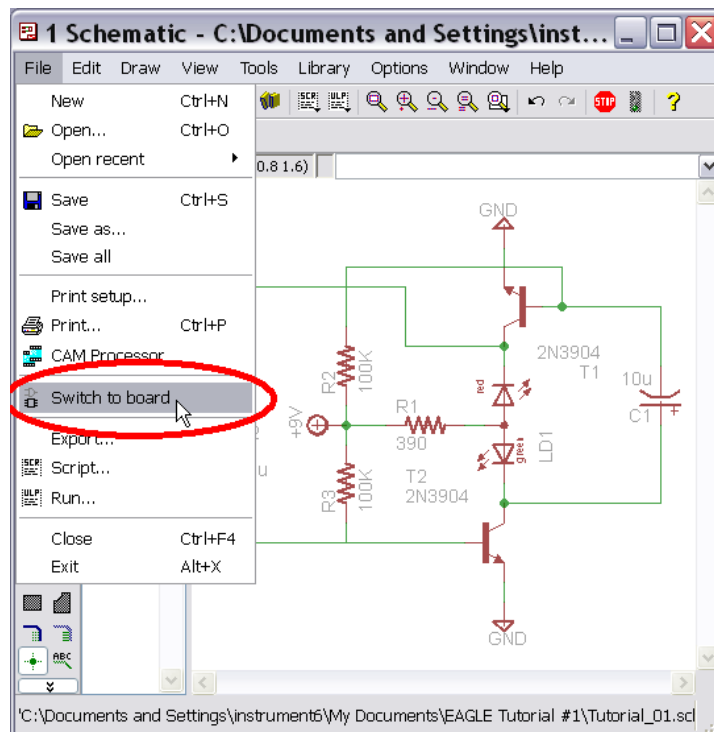
Now the circuit needs to be wired. Wires can be added to the schematic by either using the Draw Menu (Draw—Wire) or by using the Wire button as shown in the following screenshots:



Once the schematic is wired, it should look something like this (note that wires connect only where a junction point is shown):

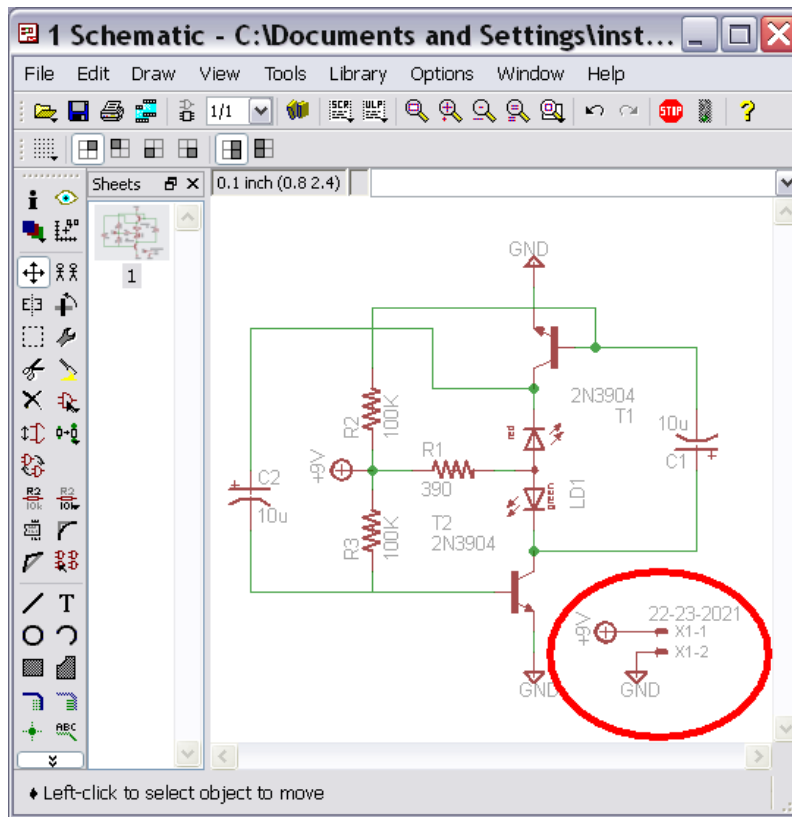


Save the schematic and invoke the Board Layout component of EAGLE (if your file doesn't exist, go ahead & click YES to create it):

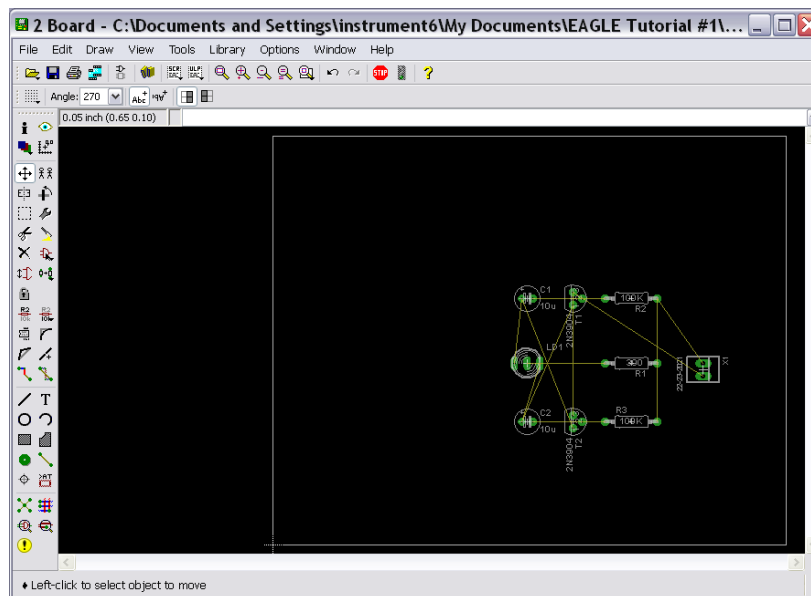




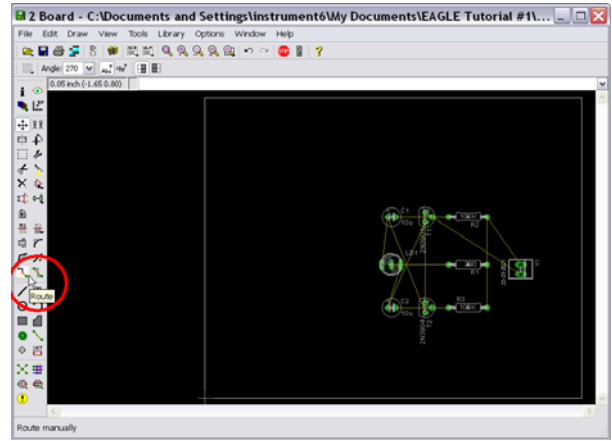
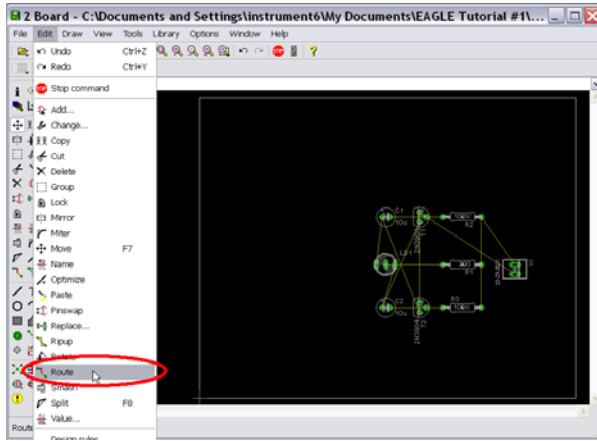
Notice that the power and ground connections are not shown! Save your board file before closing it to return to the schematic editor to add a connector that will accommodate them. In this case a Molex P/N 22-23-2021 (con-molex==22-23-2021) will be used:



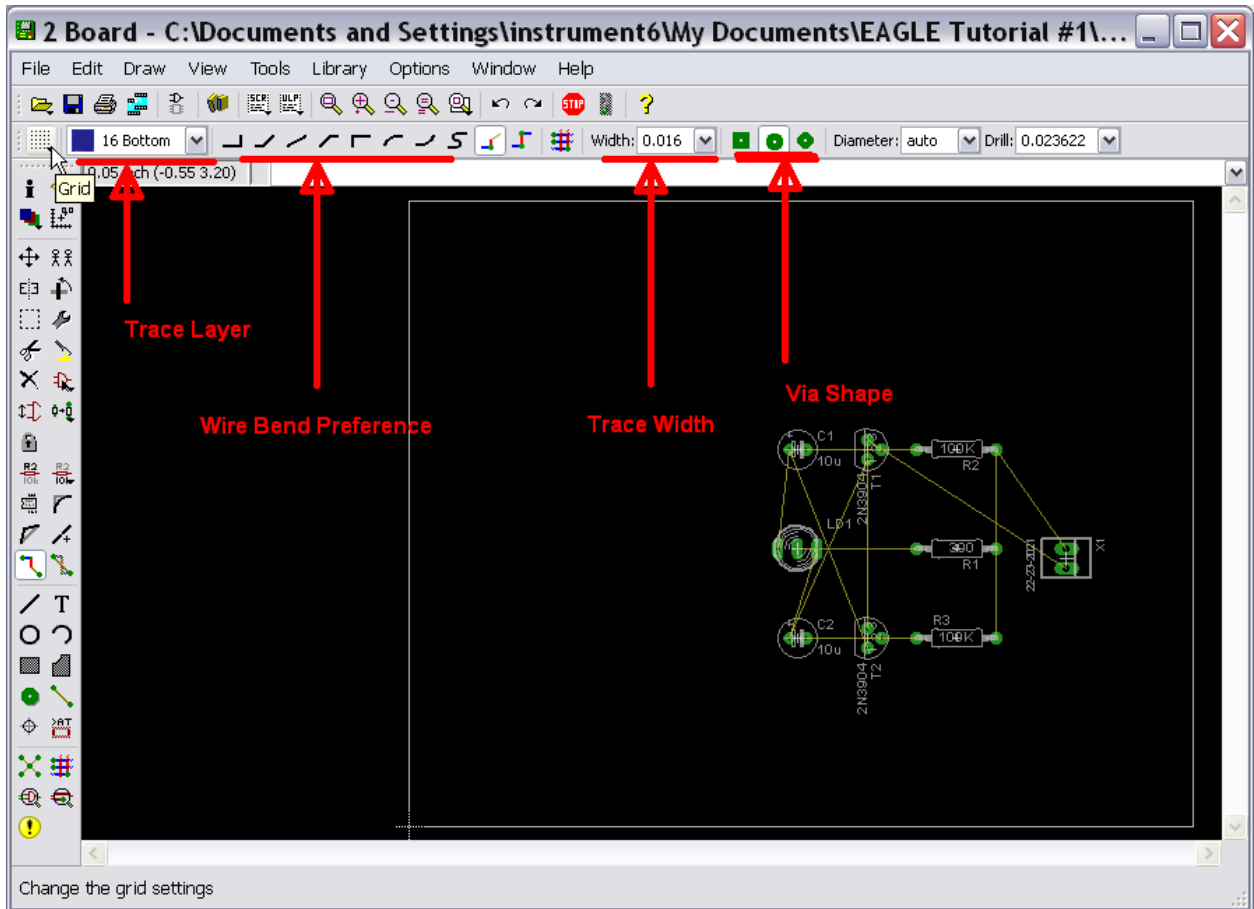
This results in some additional changes to the layout placement, this is a different possibility:



Now the traces need to be routed. Routing can be accessed by either using the Edit Menu (Edit—Route) or by using the Route button as shown in the following screenshots:



Here is a screenshot that explains a few of the other options to be aware of when routing your board:



A few board layout suggestions:

- Use a minimum trace width of 0.030 (30 mils) for boards to be milled on campus
- Limit the number of right angle bends
- Be certain to use vias if the top layer of the board won't be available to solder once the component is placed if the board is to be milled without plated thru holes (minimize top layer traces)
- Resize the board outline if the full default area is not needed
- Make sure that mounting holes are included
- Feel free to add text to the board layout area (text on top or bottom layer will appear as conductive material on final product)

Take your time!! Make sure that you are happy with the results!!

