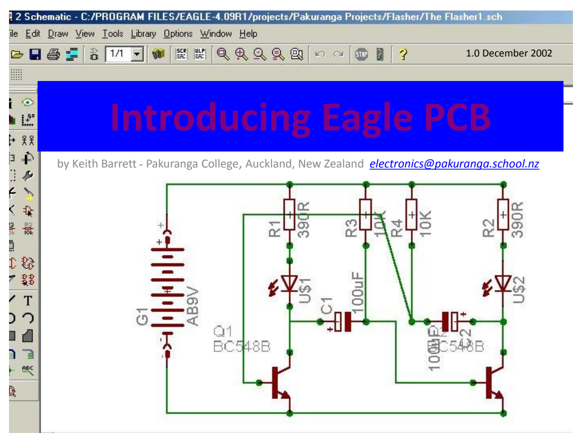
ETH Course 402-0248-00L: Electronics for Physicists II (Digital)

- **1: Setup uC tools, introduction**
- **2: Solder SMD AVR32 board**
- **3: Build application around AVR32**
- **4: Design your own PCB schematic**
- **5: Place and route your PCB**
- **6: Start logic design with FPGAs**

Printed Circuit Board (PCB) design tools

	Pros	Cons	Cost
Eagle (cadsoft)	Free (simple boards) Easy to learn Truly cross platform	Clunky interface Limited router	
Altium	Powerful	Windows only	3k CHF Or 300/yr ETH
Cadence	Really powerful	Arcane (l)unix only	$$$ except ETH has license seats

Kieth Barret's introduction to Eagle



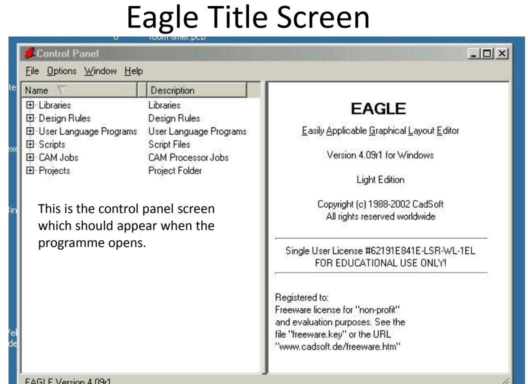
Why use Eagle PCB?

This is a CAD package which is available as a ***free*** version for small 2-sided boards (Eagle Light).

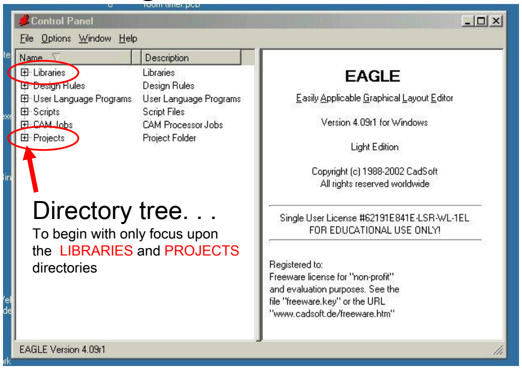
Although it may look intimidating at first glance, can be used to produce quality printed circuit boards from circuit schematic diagrams.

The website from which this software can be downloaded (Windows and Linux versions are available) is ***www.cadsoft.de***

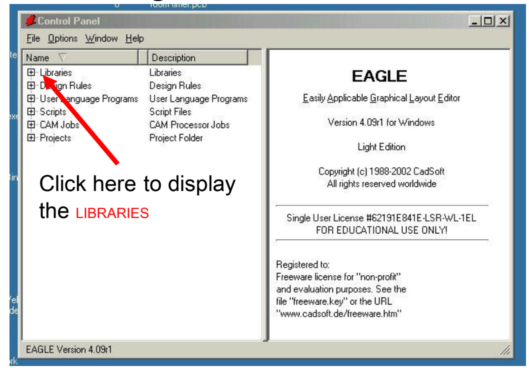
Eagle Title Screen



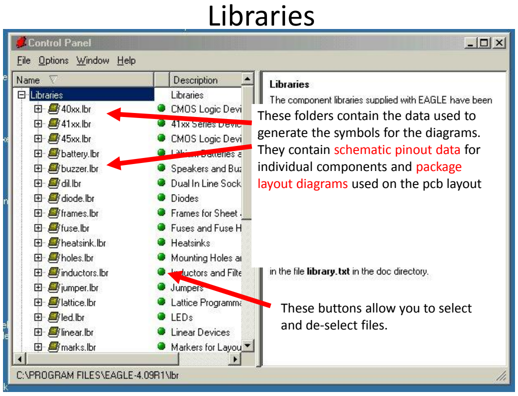
Eagle Control Panel



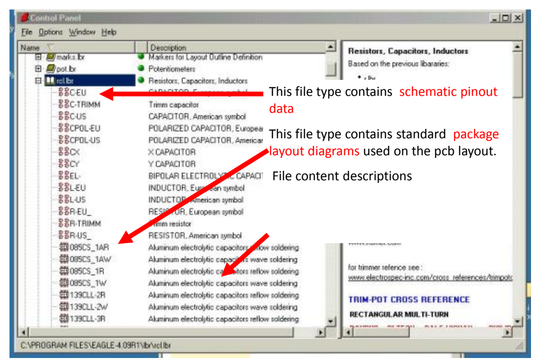
Eagle PCB Libraries



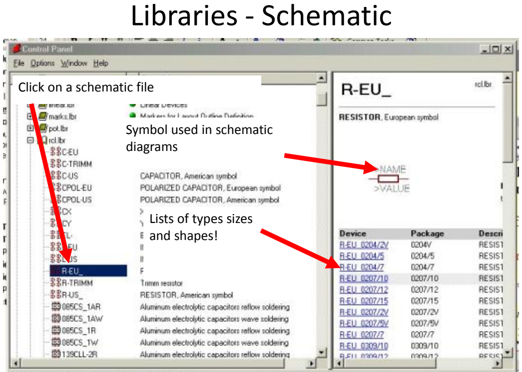
Libraries



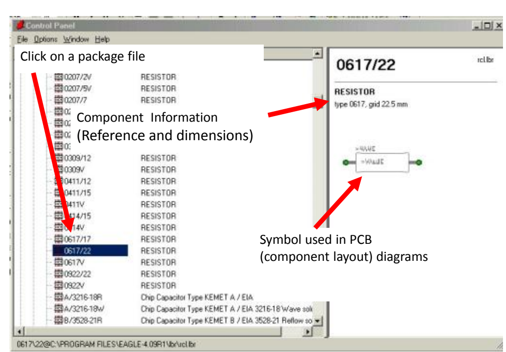
Libraries - example



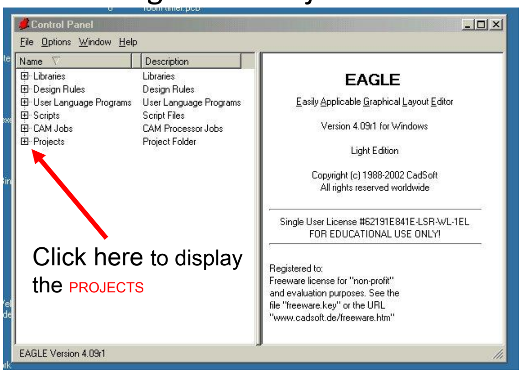
Libraries - Schematic



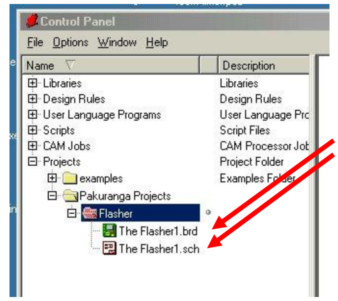
Libraries - Packages



Eagle PCB Projects



Projects

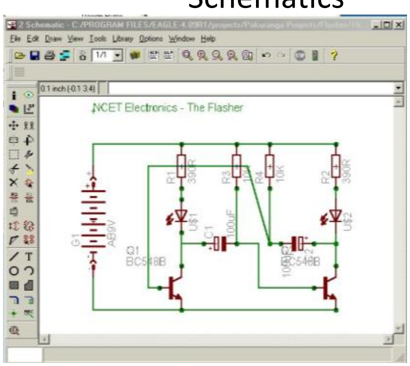


The Projects folders are where you place your work.

When creating a design there are two main types of file which are generated by the programme

.sch (Circuit schematic designs) and .brd (pcb layout boards)

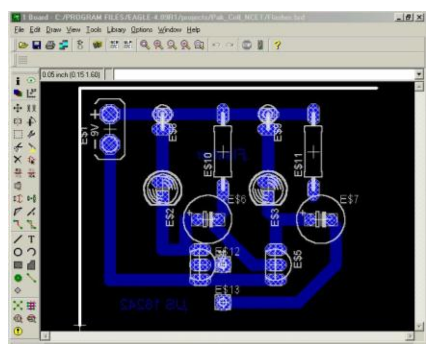
Schematics



The schematic window allows the design of circuits like this flip-flop using standard component symbols

From this point the pcb can be created on the "board" screen

Boards



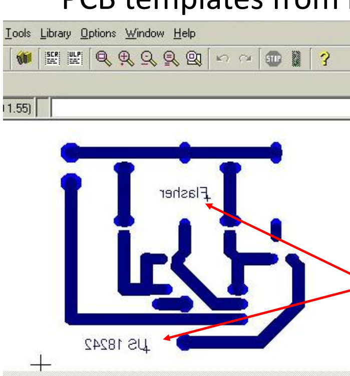
This is a board produced from the previous schematic. .

.

The software automatically generated the symbols and pathways from the schematic.

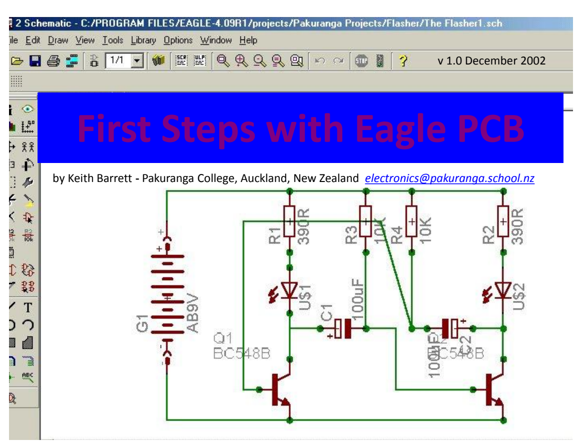
All positioning, track widths and pad sizes can be changed by the user.

PCB templates from boards



The colour scheme can be customised and layers removed so that the track layer can be printed directly onto a transparency to produce a master for photo-etching

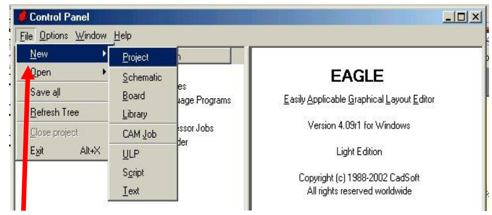
Text placed on the "bottom layer" is automatically reversed for printing



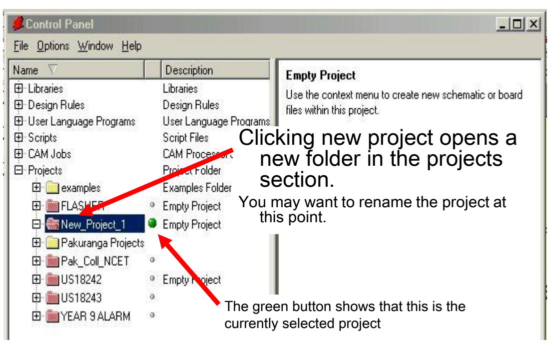
First Steps with Eagle PCB?

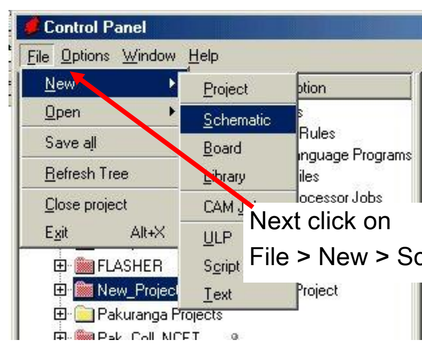
This is the second presentation and will show you how to produce a simple circuit schematic diagram using this software.

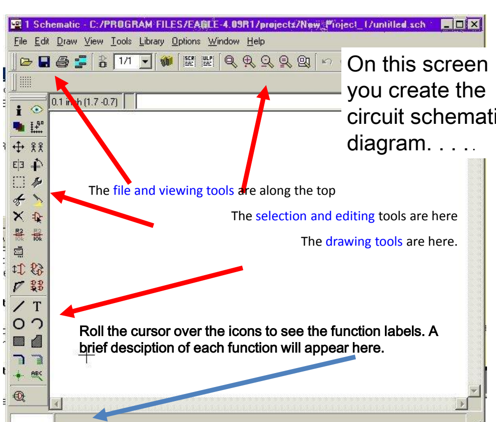
Creating a Project

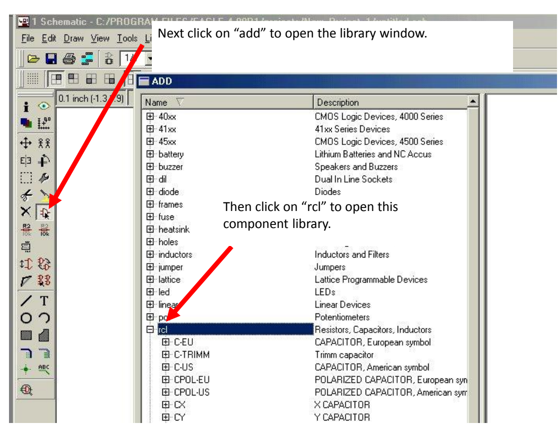


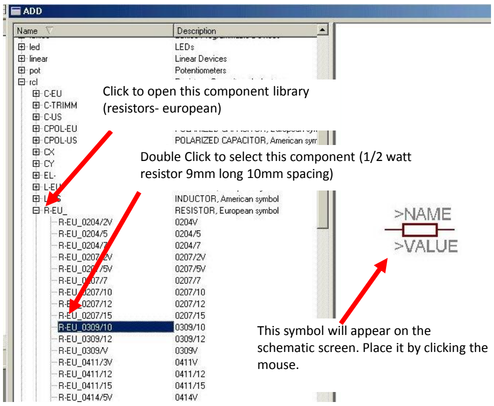
Run the Eagle program, when the control panel window appears click on File > New > Project







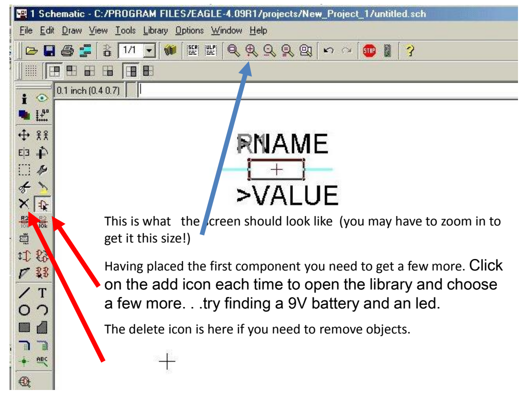




Click to open this component library (resistors- european)

Double Click to select this component (1/2 watt resistor 9mm long 10mm spacing)

This symbol will appear on the schematic screen. Place it by clicking the mouse.

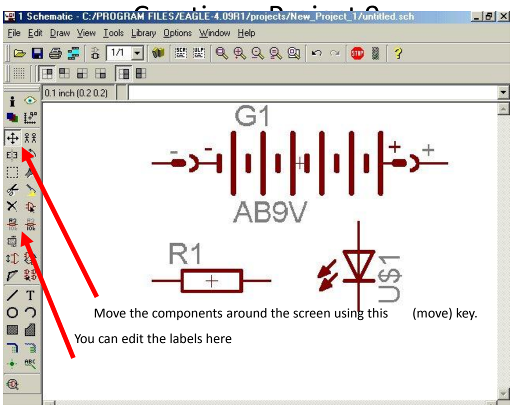


This is what the screen should look like (you may have to zoom in to get it this size!)

Having placed the first component you need to get a few more. Click on the add icon each time to open the library and choose a few more. . .try finding a 9V battery and an led.

The delete icon is here if you need to remove objects.

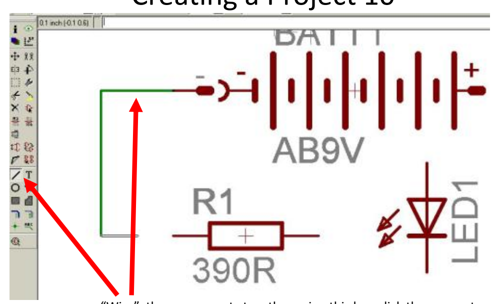
Creating a Project 9



Move the components around the screen using this (move) key.

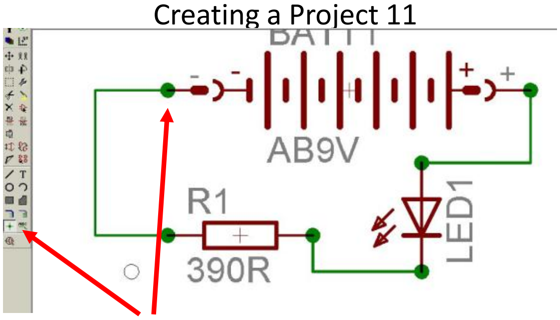
You can edit the labels here

Creating a Project 10



“Wire” the components together using this key. click the mouse to start, change direction and double click (or esc) to finish a connection.

Creating a Project 11



Place “junctions” where wires meet the components.

When you have a complete circuit SAVE it!

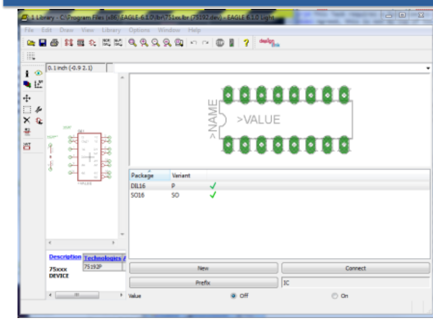
End of presentation 2

The next presentation shows you how to create and edit a pcb board from the circuit shown in this presentation.

The circuit schematic can be downloaded as “easy example 1.sch” from:

http://www.pakuranga.school.nz/depart/electronics/eaglepcb

Making library components

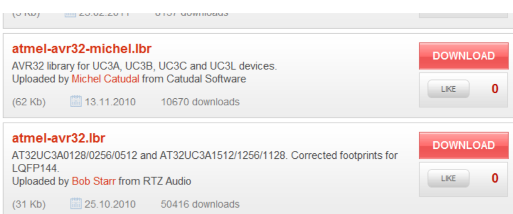


This is the 3rd presentation and will show you how to make a new library component

It follows from the excellent Eagle tutorial at http://myhome.spu.edu/bolding/EE4211/EagleTutorial4.htm

Atmel hosts a massive number of user-contributed libraries

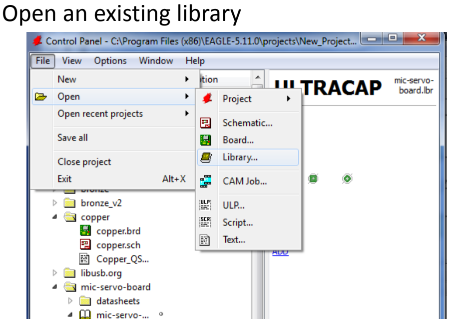
- For example, if you search for "atmel" the first items are interesting to us:



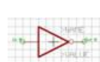
Caution: you get what you pay for….

What's in a library?

- Open an existing library

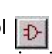
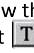


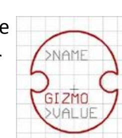
Each library contains three types of elements:



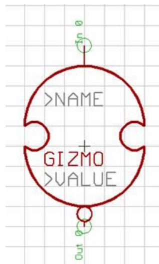
1. Symbols - These are the symbols that show up on a schematic. An inverter symbol is an example.
   – Click the Symbol tool  (from the top menu) and choose 7404 to see the inverter symbol.
2. Packages - These are the package outlines that will be used to make a PCB. A 14-pin DIL (Dual In-Line package) is an example. 
   – Click the Package tool  and select DIL14 to see the DIL14 package.
3. Devices - Devices are groups of symbols that exist in a package. For example a 7404 hex inverter is an example. It consists of six inverters, power and ground pins and contains a link to several packages including a 14-pin DIL package.
   – Click the Device tool  and select 7404 to see the 7404 hex inverter package.

Making symbol from scratch: Gizmogate

1. Make new library
2. Create new symbol  giving name
3. Draw the symbol, using Text tool for text 
   1. Put **>NAME** on *Names* layer using Change tool  selecting layer, and choosing *Names* and then clicking on >NAME.
   2. Do same for **>VALUE** but put on *Values* layer.
   3. They should turn gray.
   4. These will be filled in in your design



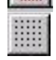
1. Now add input and output pin using Pin tool 
2. Use Change tool to select direction of pin
3. Name the pin with Name tool
4. Use the pin style to select the style, etc. 
5. Save the library

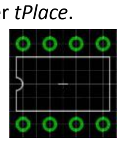


To copy a symbol from a library

1. In the existing library, use Edit Symbol tool to choose the part 
2. Make all layers visible with Layers button and selecting *All* 
3. Make a group with the Group tool by dragging around everything 
4. Select Cut to copy to clipboard 
5. Open your library and make your new part. Paste the copy using Paste tool. 
6. Change as you like using Change tool 

Making a package from scratch

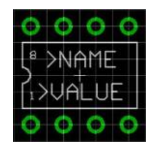
1. Select Package, enter name DIL-8 
2. Select Grid tool and select grid 0.025 with multiple of 2 
3. For through hole parts, you need holes in right place and large enough! (**Common error, holes too small!**) Use the Hole tool, place pads CCW from bottom left. 
4. For SMD, use the SMD Pad tool 
5. Draw the outline on layer *tPlace*. 



Place >NAME and >VALUE on part

1. Put >NAME on tName layer
2. Put >VALUE on tValue layer
3. Add pin 1&8 labels

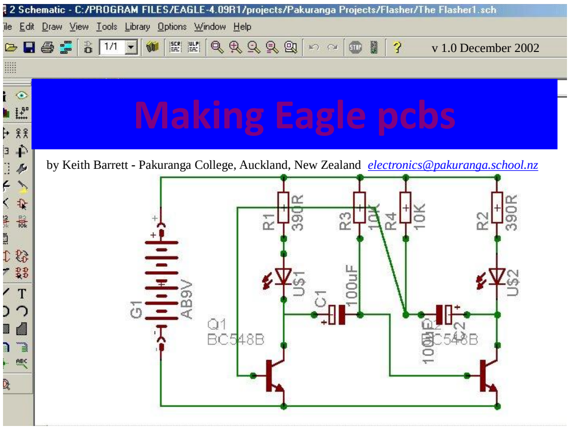
You're done with DIL-8 part. Save your library



Putting symbols and packages together

The final task in making a part is to create a *device* that has information on how symbols are placed inside of a package. For many devices, there will be only one symbol. However, many devices contain multiple symbols, as well as hidden power pins.

1. Make a new device with the Device button. Enter the name. 
2. Add N copies of symbol, depending on number of gates in device.
3. Change names using Names tool. 

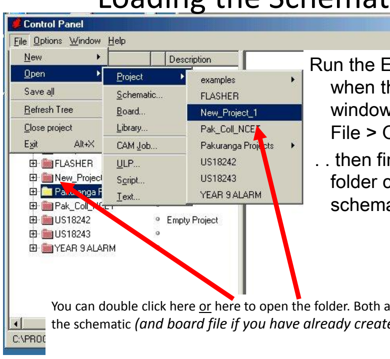


Making Eagle PCBs?

This is the 4th presentation and will show you how to produce a simple, single layer printed circuit board from a circuit schematic diagram using this software.

The circuit schematic for this project can be downloaded as "easy example 1.sch" from: http://www.pakuranga.school.nz/depart/electronics/eaglepcb

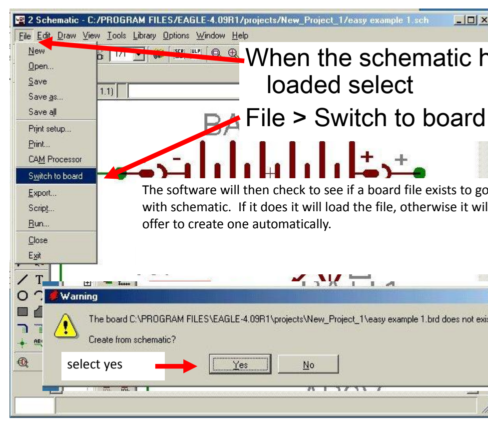
Loading the Schematic



Run the Eagle program, when the control panel window appears click on File > Open >Project

. . then find the project folder containing your schematic.

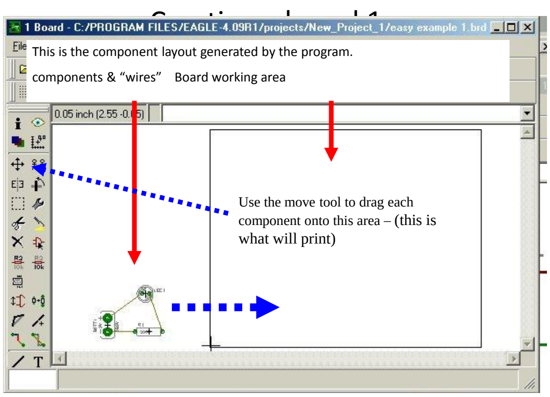
You can double click here or here to open the folder. Both actions will load the file for the schematic *(and board file if you have already created one)*

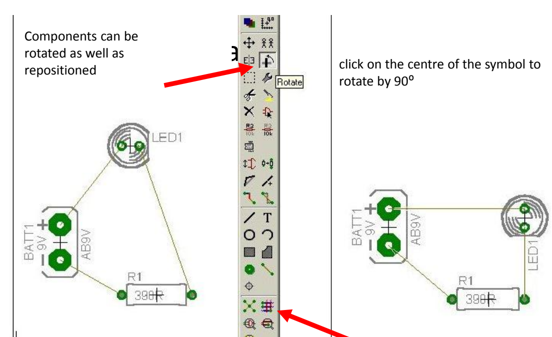


When the schematic has loaded select File > Switch to board

The software will then check to see if a board file exists to go with schematic. If it does it will load the file, otherwise it will offer to create one automatically.

select yes



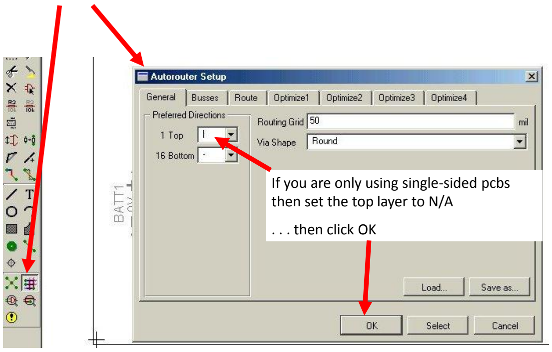


Components can be rotated as well as repositioned

click on the centre of the symbol to rotate by 90°

The next stage is to create the tracks using "autoroute"

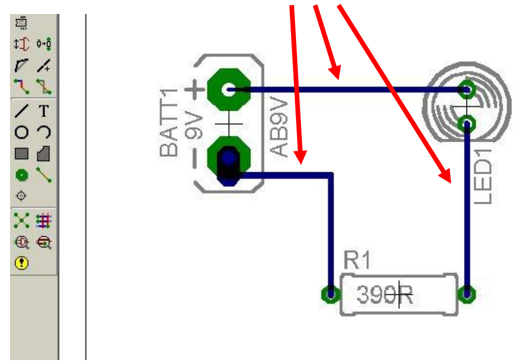
Clicking here generates an options menu. . .



If you are only using single-sided pcbs then set the top layer to N/A

. . . then click OK

Autoroute then turns the “wires” into track pathways. . .

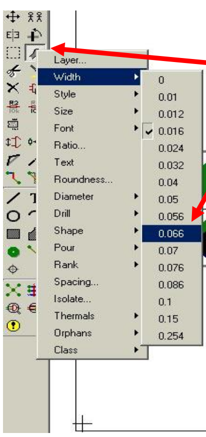


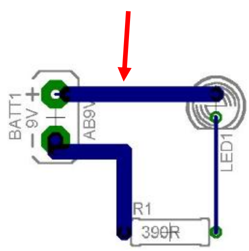
To change the track width,

select **change > width**

. . .then select the dimension (these are in inches – units can be set in mm)

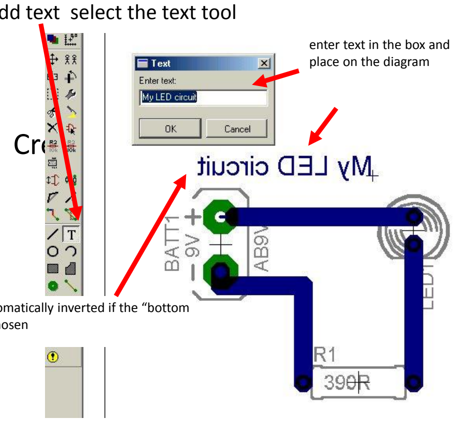
point at the track and right click.





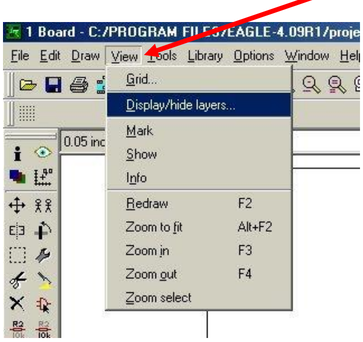
To add text select the text tool

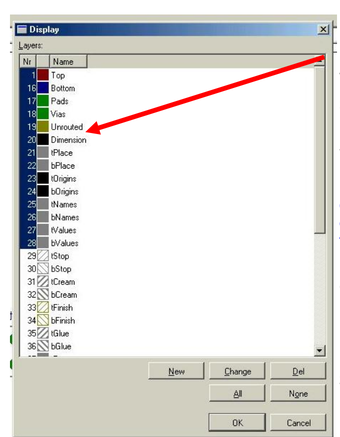
enter text in the box and place on the diagram



Text is automatically inverted if the “bottom layer” is chosen

Choose the layers to print select **View > Display/hide**





Here you can select what appears on screen and the printer. (blue = selected / white = deselected)

Click on the numbers to turn on/off. Click on the colour to edit the colour scheme

For most single sided pcbs everything should be off except layers 16,17 (tracks & pads) 45 (drill holes)

Save and print your board.

End of presentation 3

http://www.pakuranga.school.nz/depart/electronics/eaglepcb