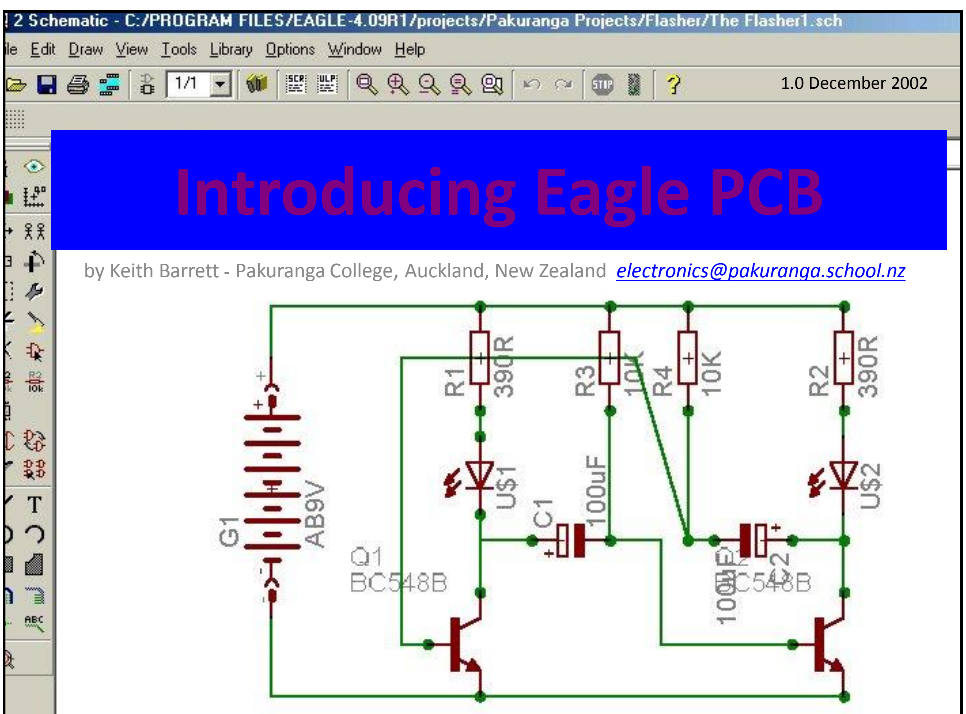
ETH Course 402-0248-00L: Electronics for Physicists II (Digital)

- **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. Fabricate your PCB at a production house**
- **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	Free for limited version. 345 EUR for full version if student.
Altium	Powerful	Windows only	3k CHF Or 300/yr ETH
Cadence	Really powerful	Arcane (l)unix only	$$$ except ETH has license seats

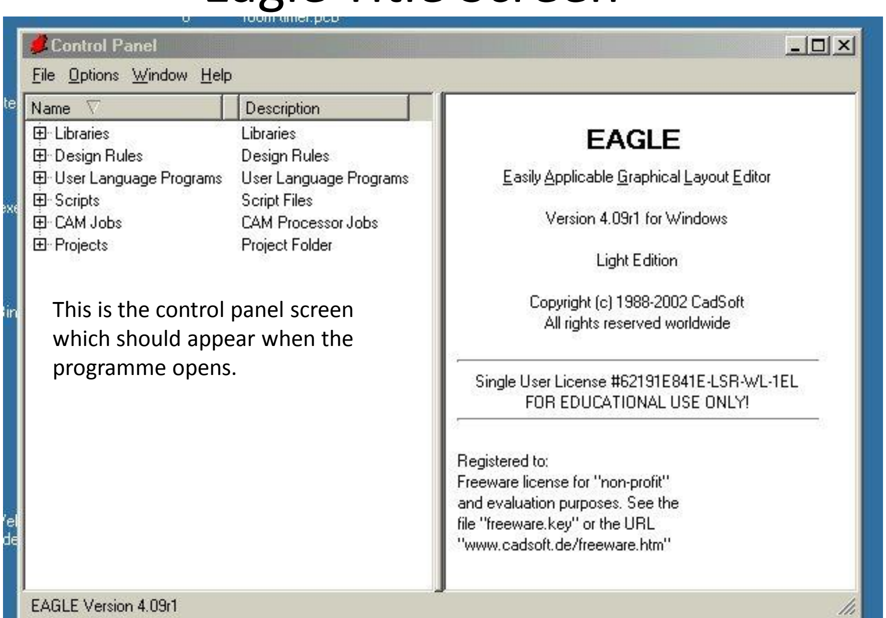
Keith Barret's introduction to Eagle



Eagle costs

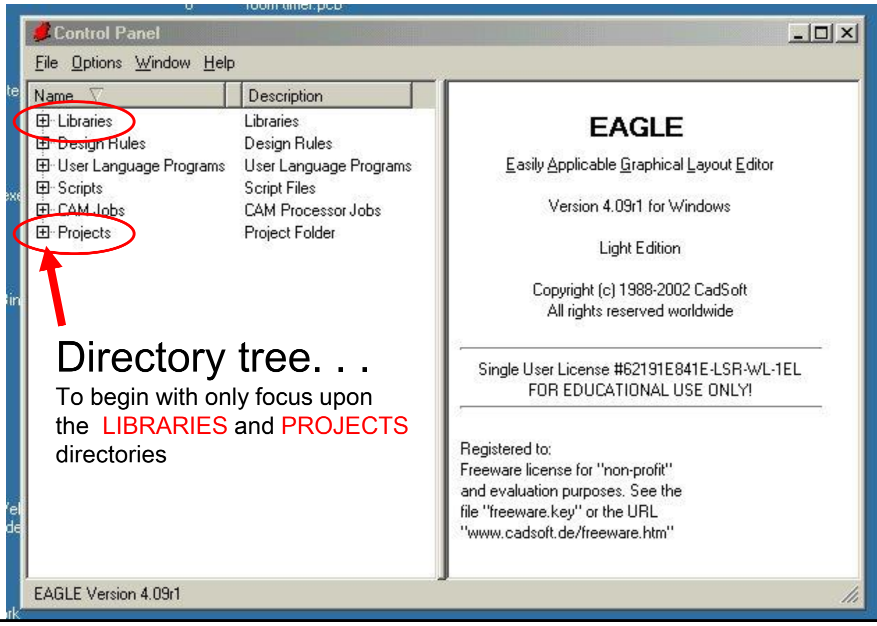
EAGLE Light	(1 schematic sheet, 2 signal layers, 100x80mm routing area)	Free for Educational purpose
EAGLE Standard	(99 schematic sheets, 6 signal layers, 160x100mm routing area)	http://www.cadsoftusa.com/shop/pricing/
EAGLE Hobbyist	(99 schematic sheets, 6 signal layers, 160x100mm routing area; **for individual, non commercial use only**)	http://www.cadsoftusa.com/shop/pricing/
EAGLE Professional	(999 schematic sheets, 16 signal layers, 4m x 4m routing area)	http://www.cadsoftusa.com/shop/pricing/

Eagle Title Screen



This is the control panel screen which should appear when the programme opens.

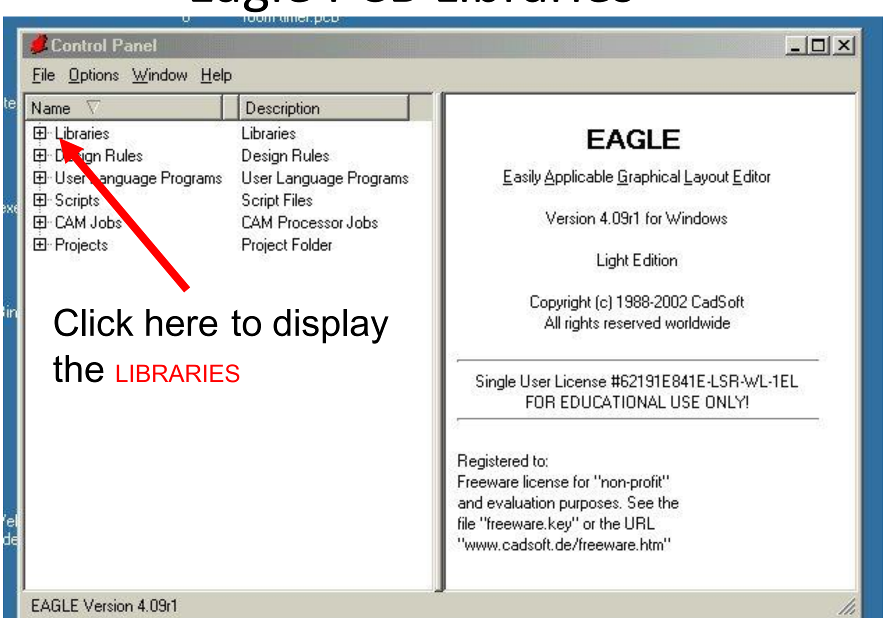
Eagle Control Panel



Directory tree. . .

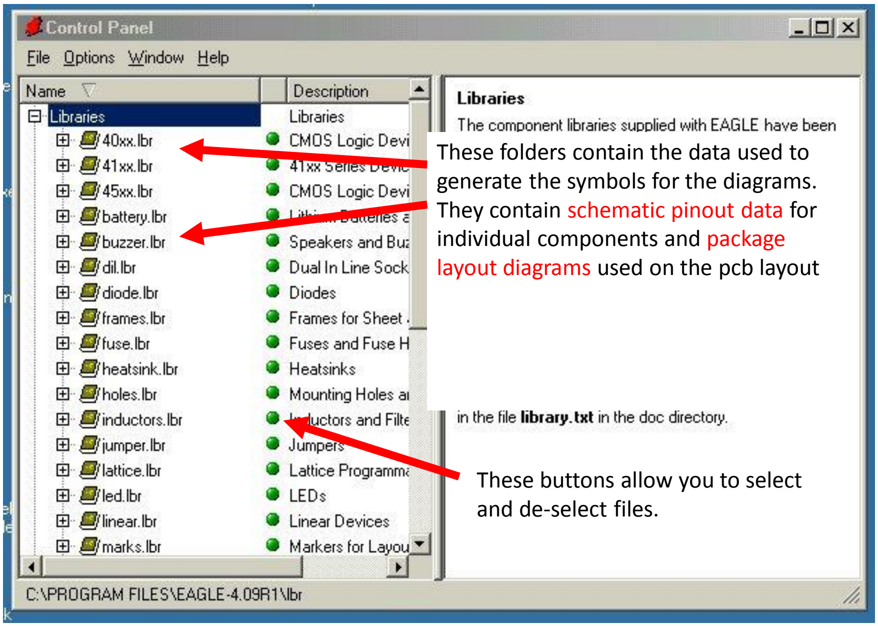
To begin with only focus upon the LIBRARIES and PROJECTS directories

Eagle PCB Libraries

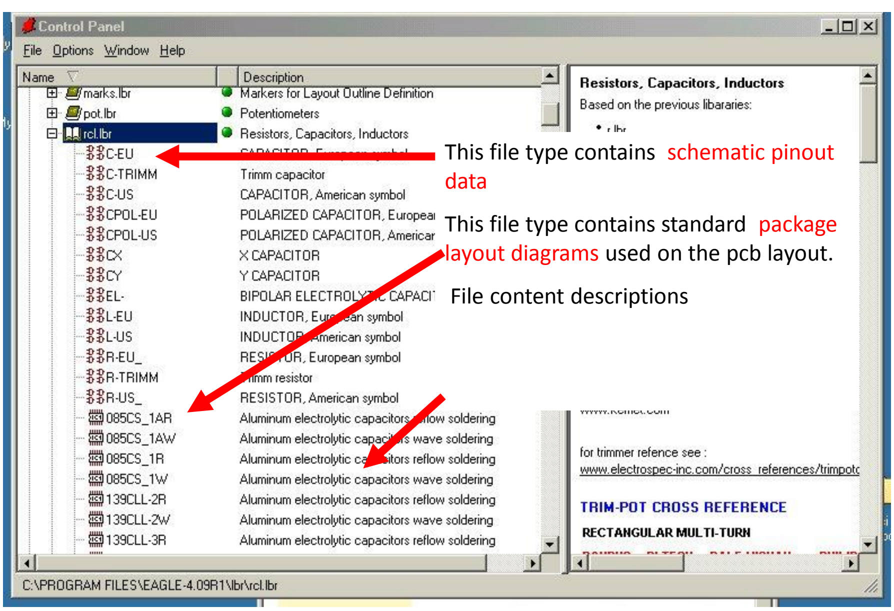


Click here to display the LIBRARIES

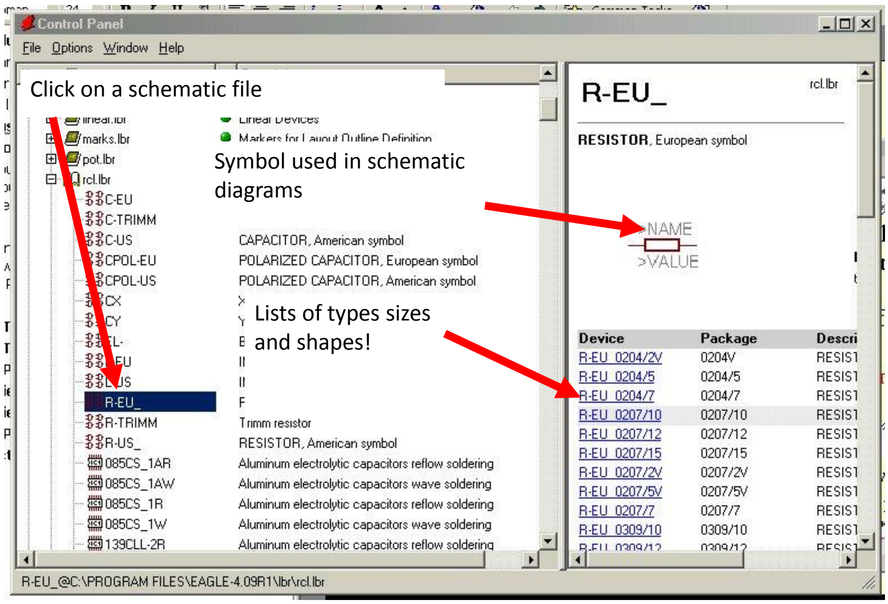
Libraries



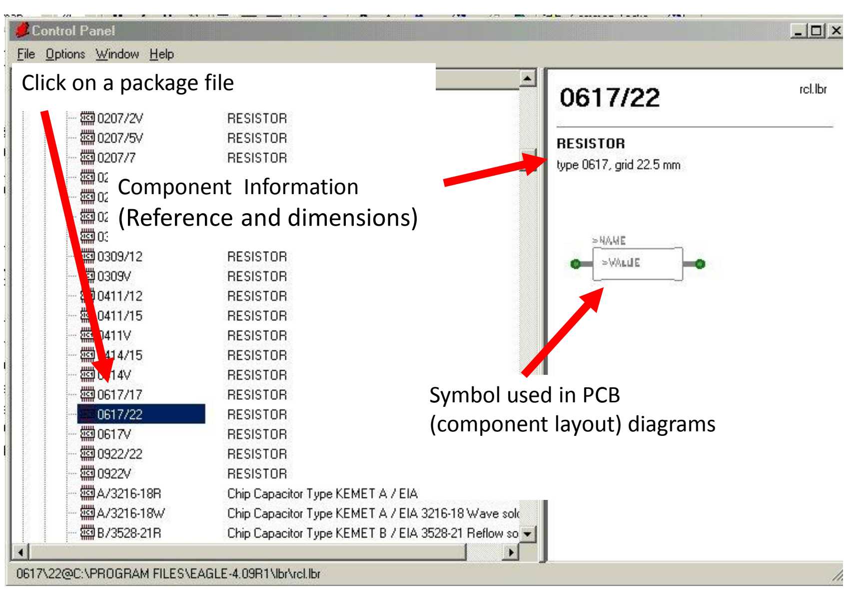
Libraries - example



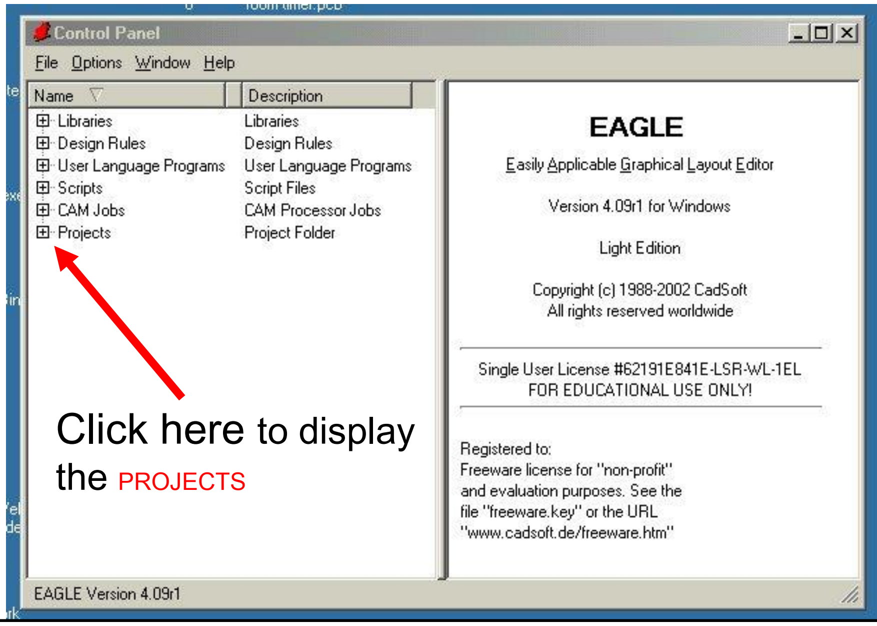
Libraries - Schematic



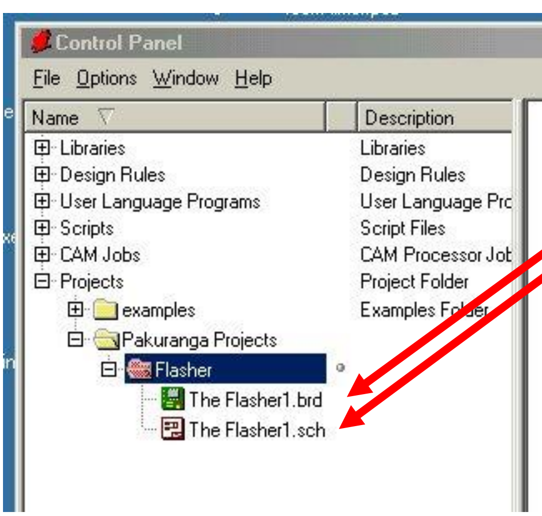
Libraries - Packages



Eagle PCB Projects



Projects

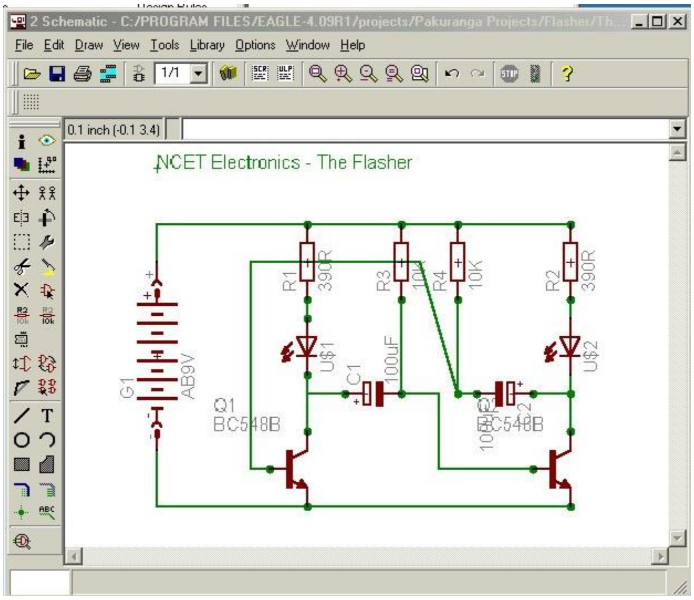


The Projects folders are where you place your work.

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)

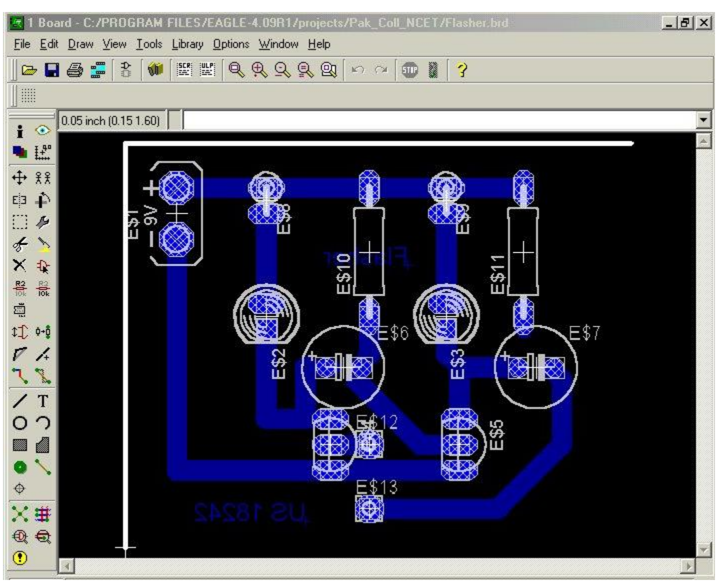
Schematics



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

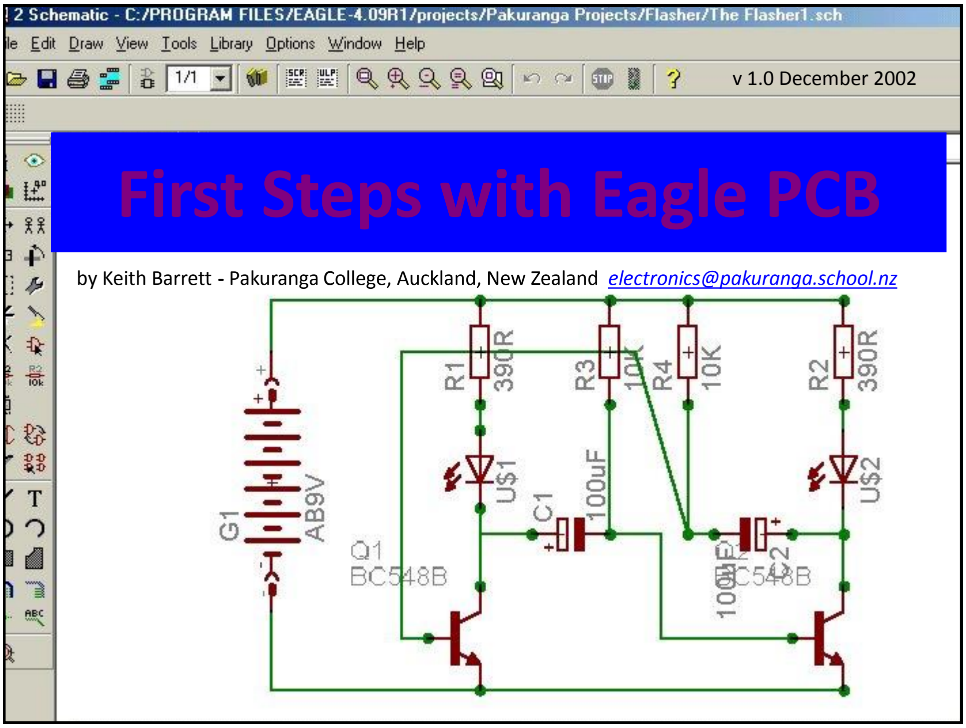
Boards



This is a board produced from the previous schematic. . .

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

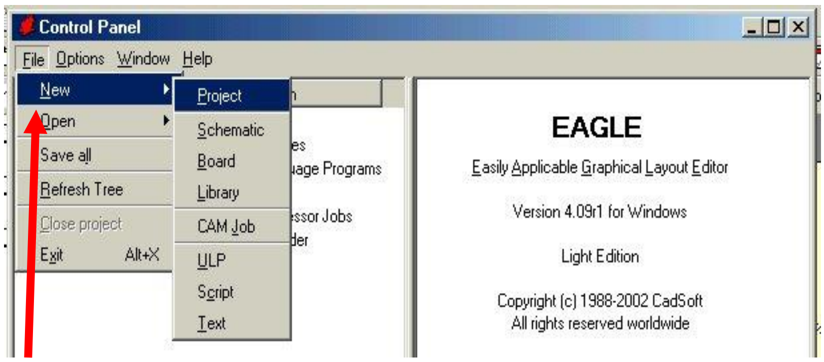
All positioning, track widths and pad sizes can be changed by the user.



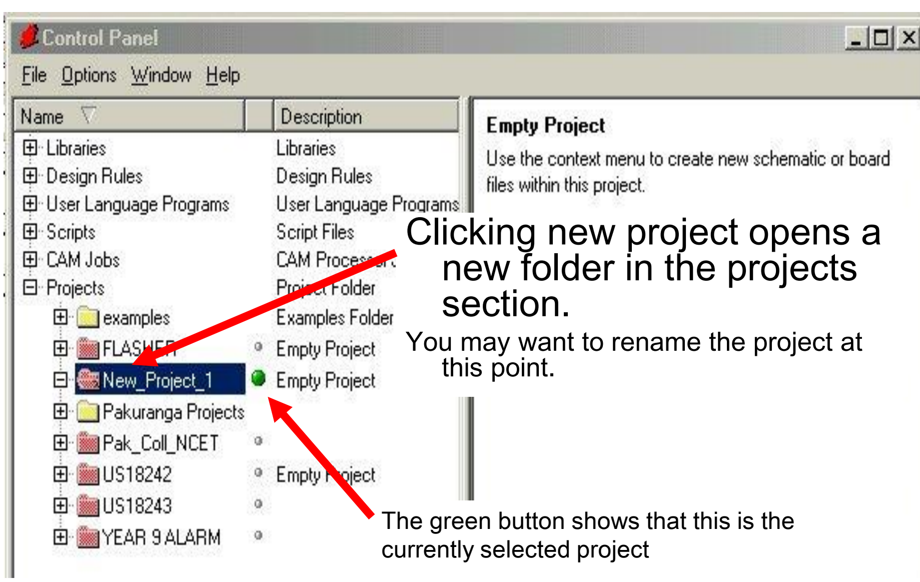
First Steps with Eagle PCB?

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

Creating a Project



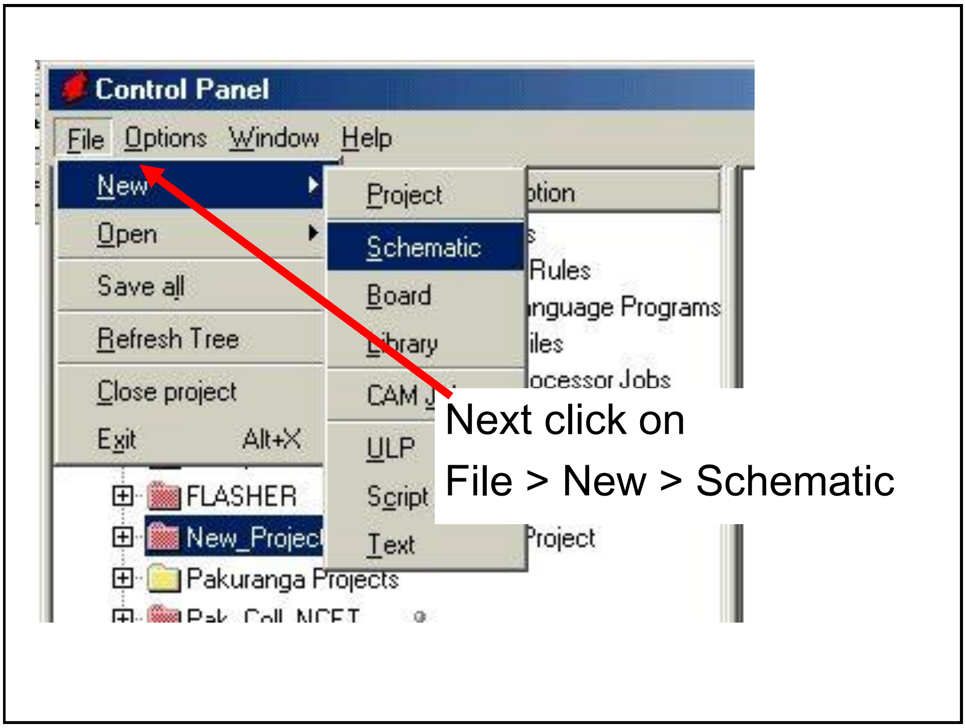
Run the Eagle program, when the control panel window appears click on File > New > Project

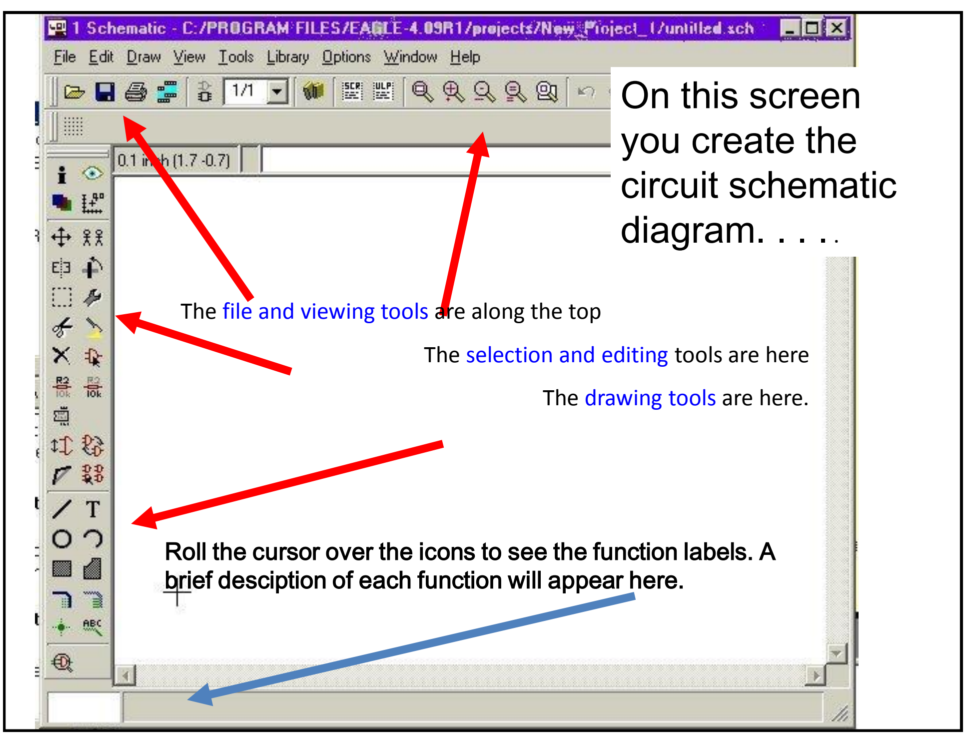


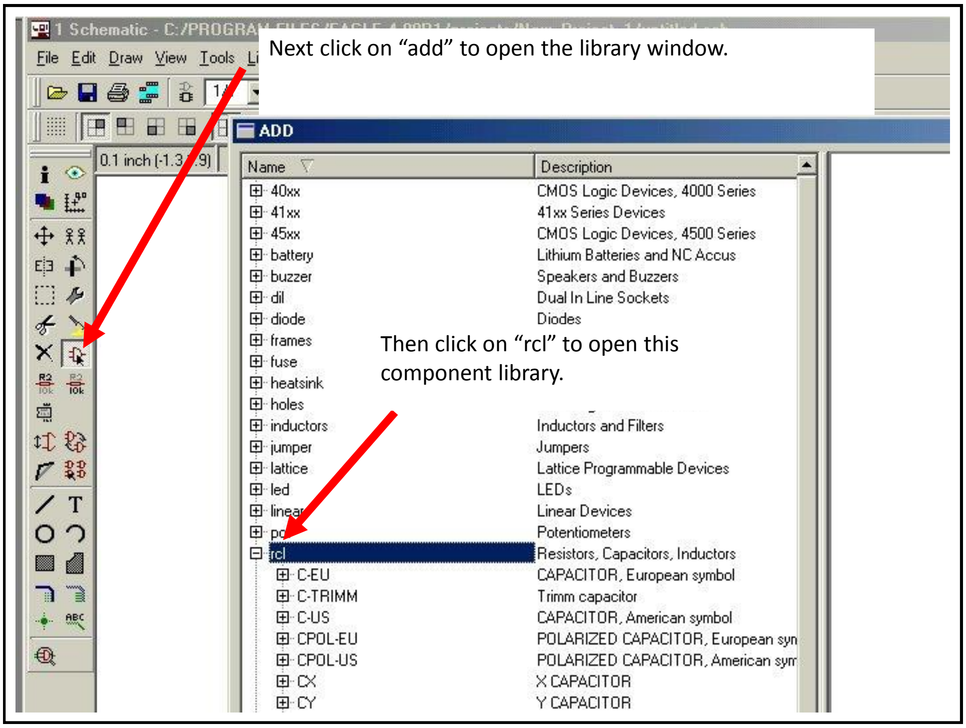
Clicking new project opens a new folder in the projects section.

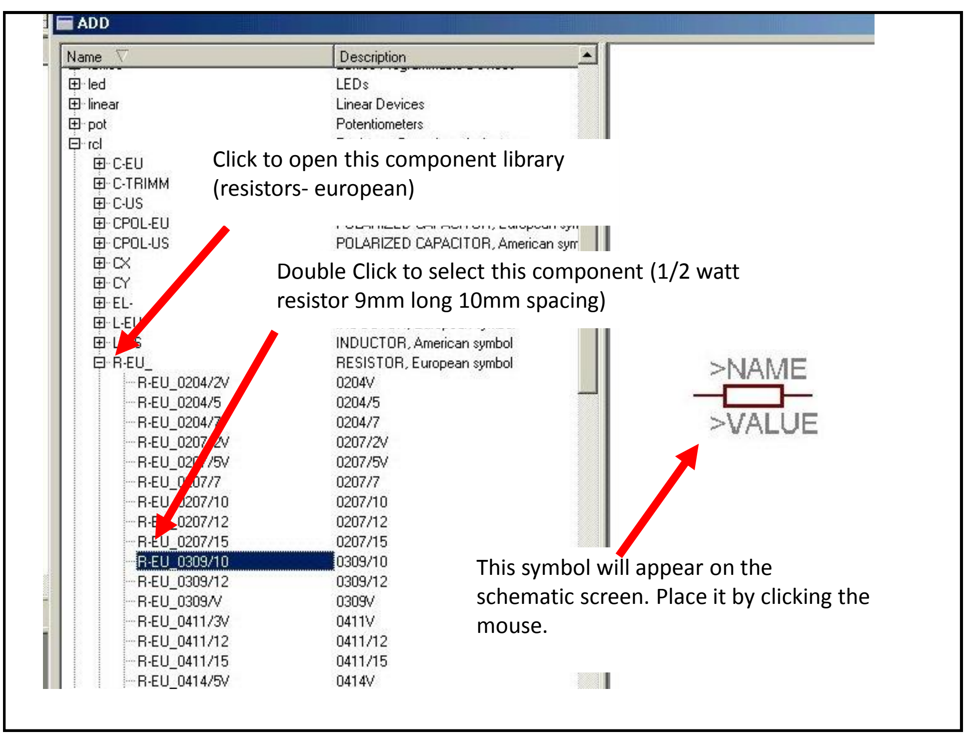
You may want to rename the project at this point.

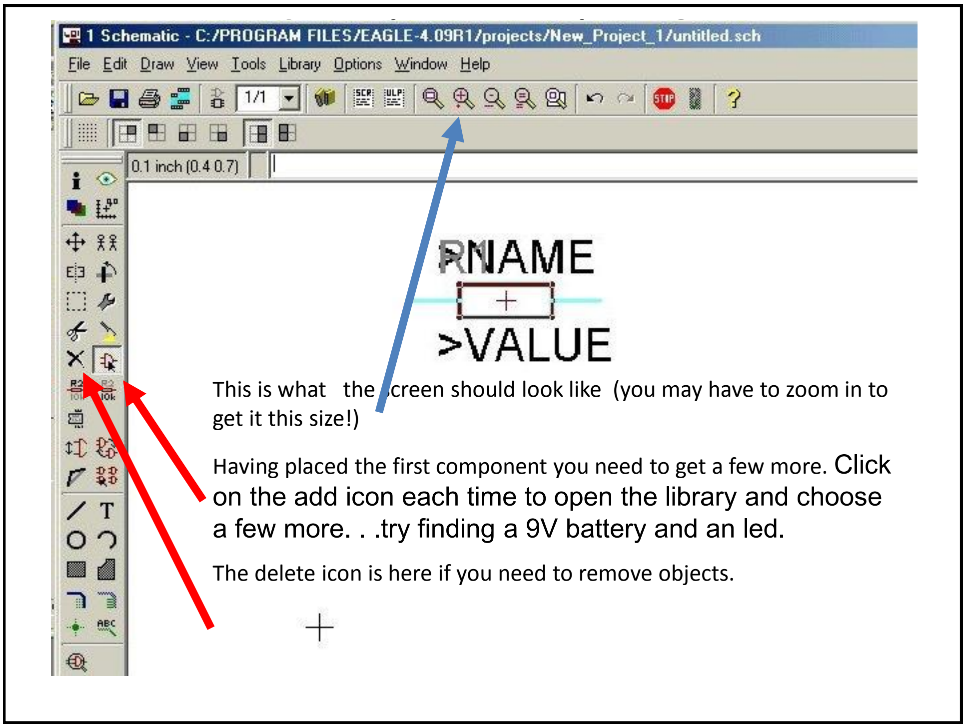
The green button shows that this is the currently selected project

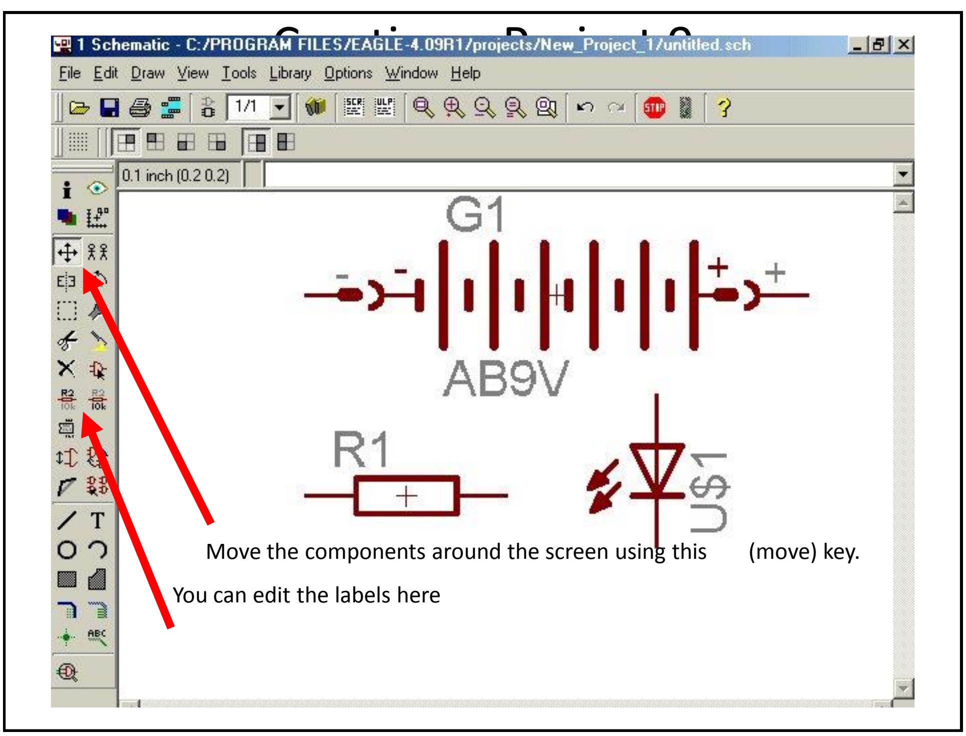

Control Panel
File Options Window Help
New
Open
Save all
Refresh Tree
Close project
Exit Alt+X
Project
Schematic
Board
Library
CAM
ULP
Script
Text
FLASHER
New_Project
Pakuranga Projects
Next click on
File > New > Schematic


1 Schematic - C:/PROGRAM FILES/EAGLE-4.09R1/projects/New_Project_1/untitled.sch
File Edit Draw View Tools Library Options Window Help
On this screen you create the circuit schematic diagram. . . . .
The file and viewing tools are along the top
The selection and editing tools are here
The drawing tools are here.
Roll the cursor over the icons to see the function labels. A brief desciption of each function will appear here.

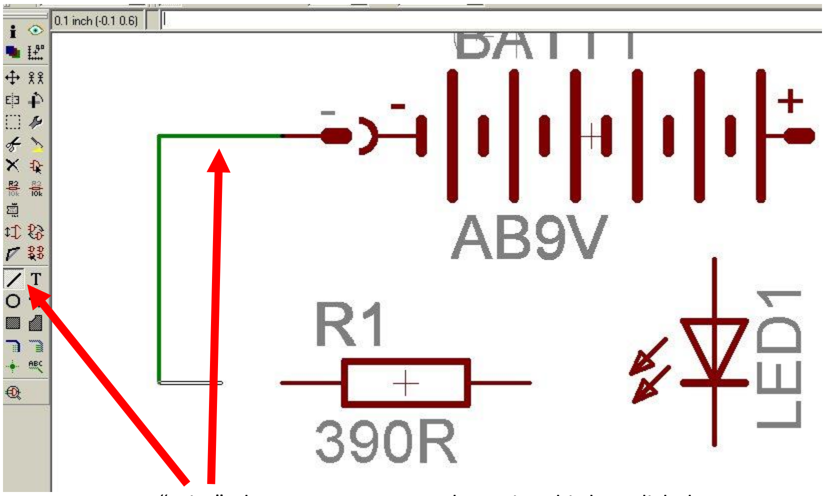

1 Schematic - C:/PROGRAM
File Edit Draw View Tools
Next click on “add” to open the library window.
ADD
0.1 inch (-1.3
Name
Description
40xx
CMOS Logic Devices, 4000 Series
41xx
41xx Series Devices
45xx
CMOS Logic Devices, 4500 Series
battery
Lithium Batteries and NC Accus
buzzer
Speakers and Buzzers
dil
Dual In Line Sockets
diode
Diodes
frames
fuse
heatsink
holes
Then click on “rcl” to open this component library.
inductors
Inductors and Filters
jumper
Jumpers
lattice
Lattice Programmable Devices
led
LEDs
Linear Devices
Potentiometers
rcl
Resistors, Capacitors, Inductors
C-EU
CAPACITOR, European symbol
C-TRIMM
Trimm capacitor
C-US
CAPACITOR, American symbol
CPOL-EU
CPOL-US
CX
X CAPACITOR
CY
Y CAPACITOR


ADD
Name
Description
led
LEDs
linear
Linear Devices
pot
Potentiometers
rcl
C-EU
C-TRIMM
C-US
CPOL-EU
CPOL-US
CX
CY
EL-
Click to open this component library (resistors- european)
POLARIZED CAPACITOR, American sym
Double Click to select this component (1/2 watt resistor 9mm long 10mm spacing)
INDUCTOR, American symbol
R-EU_
RESISTOR, European symbol
R-EU_0204/2V
0204V
R-EU_0204/5
0204/5
0204/7
0207/2V
0207/5V
0207/7
0207/10
0207/12
R-EU_0207/15
0207/15
R-EU_0309/10
0309/10
R-EU_0309/12
0309/12
R-EU_0309/V
0309V
R-EU_0411/3V
0411V
R-EU_0411/12
0411/12
R-EU_0411/15
0411/15
R-EU_0414/5V
0414V
>NAME
>VALUE
This symbol will appear on the schematic screen. Place it by clicking the mouse.


1 Schematic - C:/PROGRAM FILES/EAGLE-4.09R1/projects/New_Project_1/untitled.sch
File Edit Draw View Tools Library Options Window Help
1/1
0.1 inch (0.4 0.7)
>NAME
>VALUE
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.

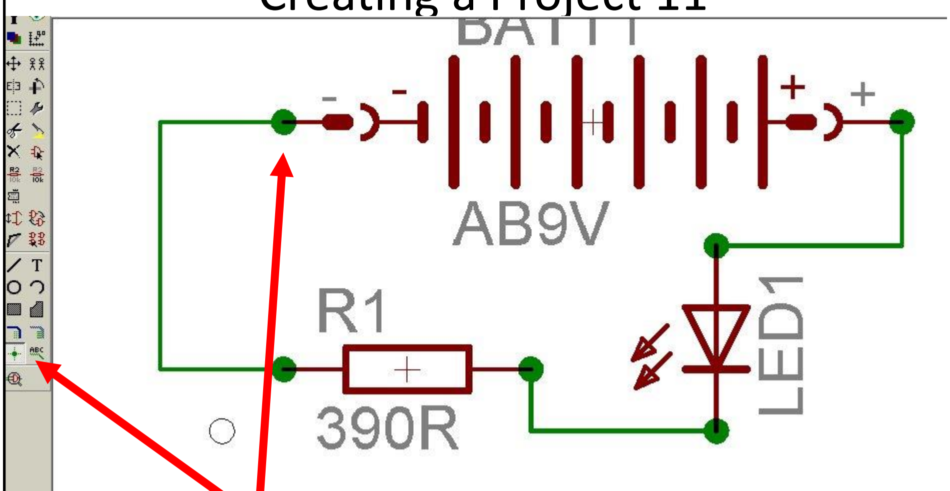

1 Schematic - C:/PROGRAM FILES/EAGLE-4.09R1/projects/New_Project_1/untitled.sch
File Edit Draw View Tools Library Options Window Help
1/1
0.1 inch (0.2 0.2)
G1
AB9V
R1
US1
Move the components around the screen using this (move) key.
You can edit the labels here

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.

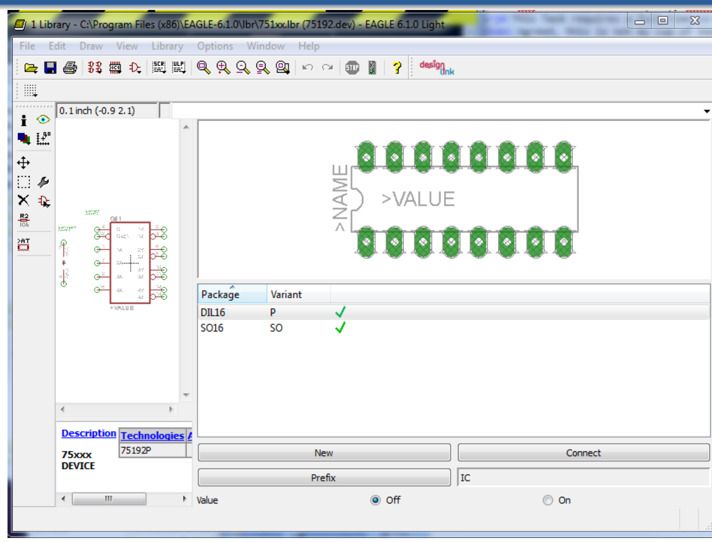
Creating a Project 11



Place "junctions" where wires meet the components.

When you have a complete circuit SAVE it!

Making library components

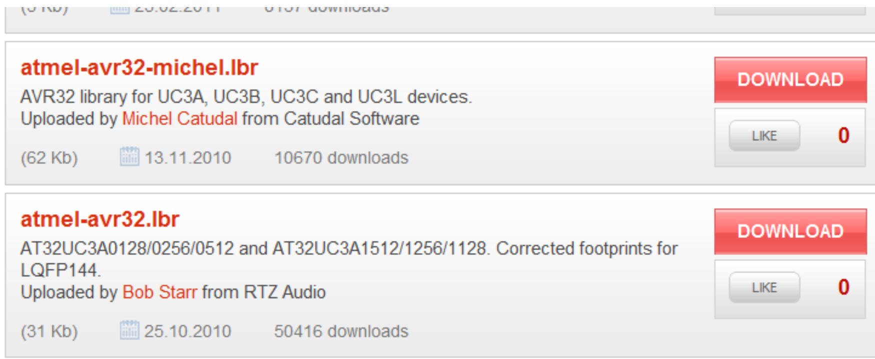


This is the 3rd presentation and will show you how to make a new library component

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

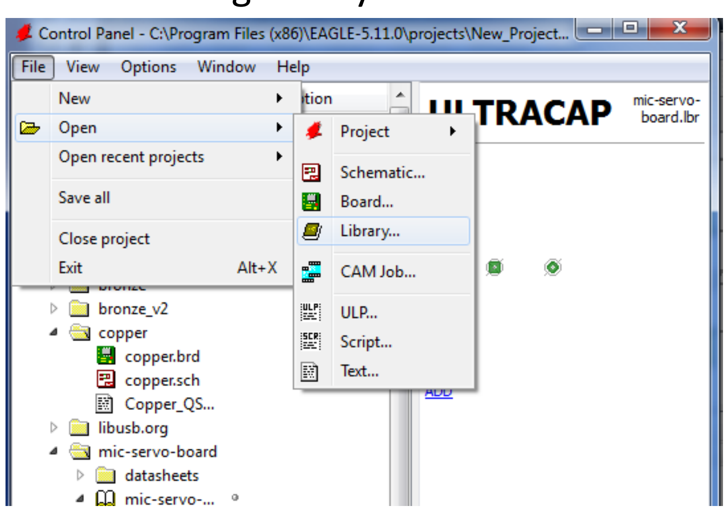
- For example, if you search at cadsoft for "atmel" the first items are interesting to us:



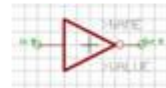
Caution: you get what you pay for….

What's in a library?

- Open an existing library



Each library contains three types of elements:



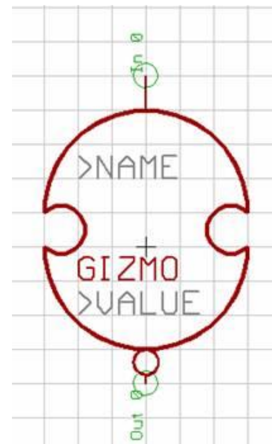
1. **Symbols** - These are the symbols that show up on a schematic. An inverter symbol is an example.
   – Click the Symbol tool (from the top menu) and choose 7404 to see the inverter symbol. 
2. **Packages** - These are the package outlines that will be used to make a PCB. A 14-pin DIL (Dual In-Line 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. 

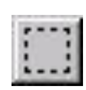
Making symbol from scratch: Gizmogate

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



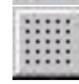
To copy a symbol from a library

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

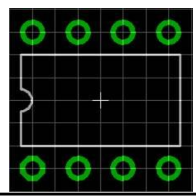
2. Select Grid tool and select grid 0.025 with multiple of 2



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

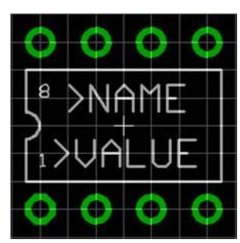
5. Draw the outline on layer *tPlace*.



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

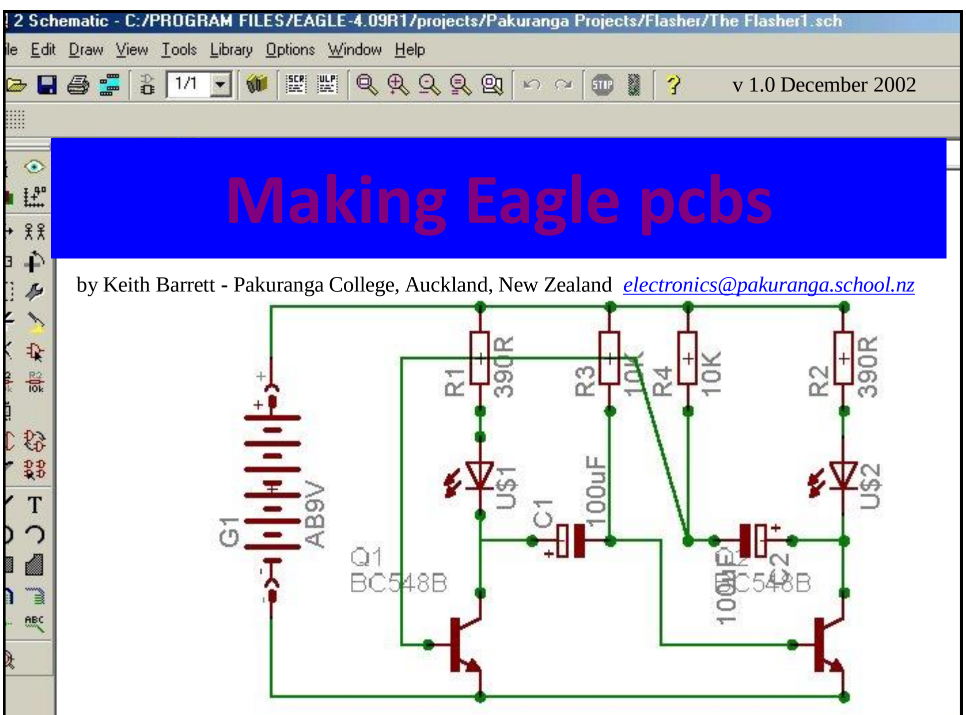
The final task in making a part is to create a *device* that has information on how symbols are placed inside of a package. For many devices, there will be only one symbol. However, many devices contain multiple symbols, as well as hidden power pins.

---

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.

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

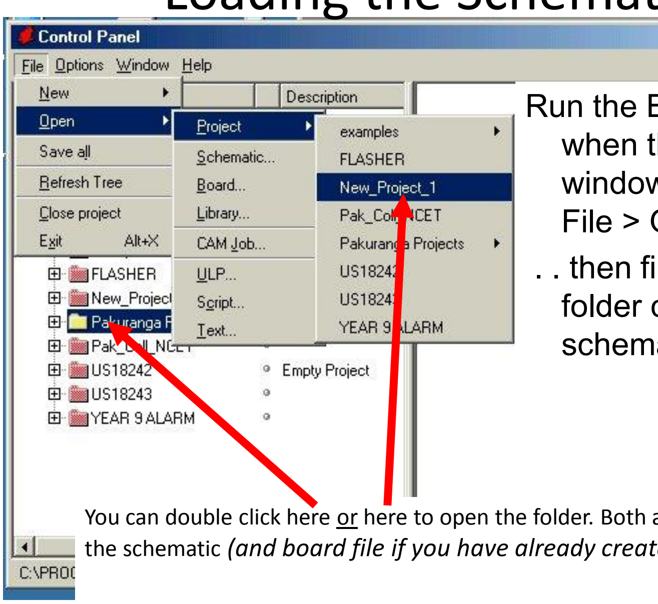
- 1: Setup uC tools, introduction
- 2: Solder SMD AVR32 board
- 3: Build application around AVR32
- 4: Design your own PCB schematic
- **5: Place and route your PCB. Fabricate your PCB at a production house**
- 6: Start logic design with FPGAs



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.

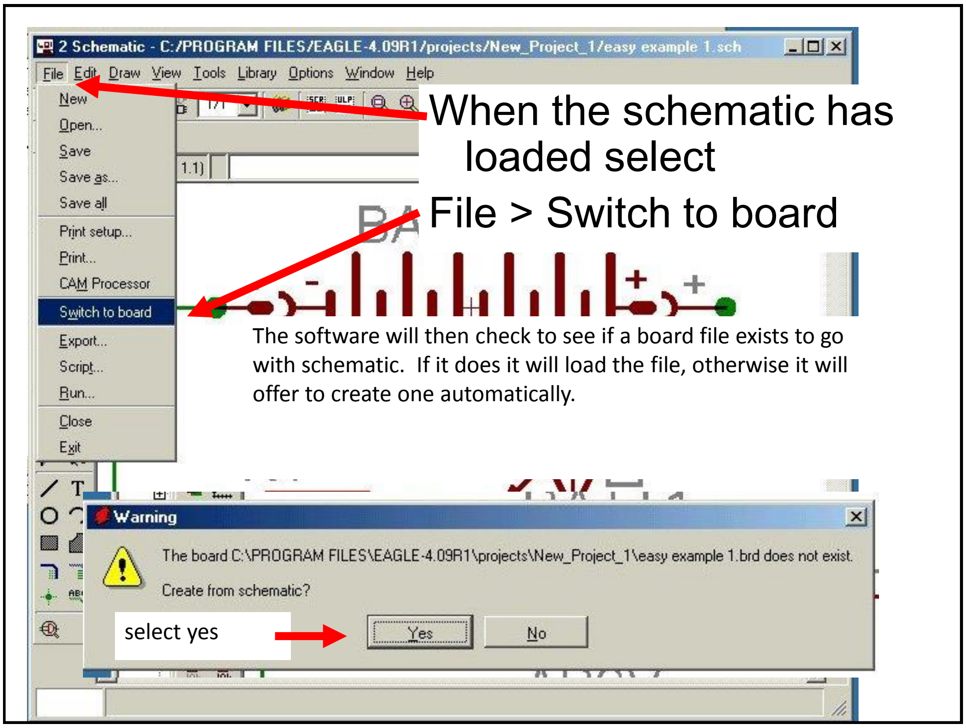
Loading the Schematic

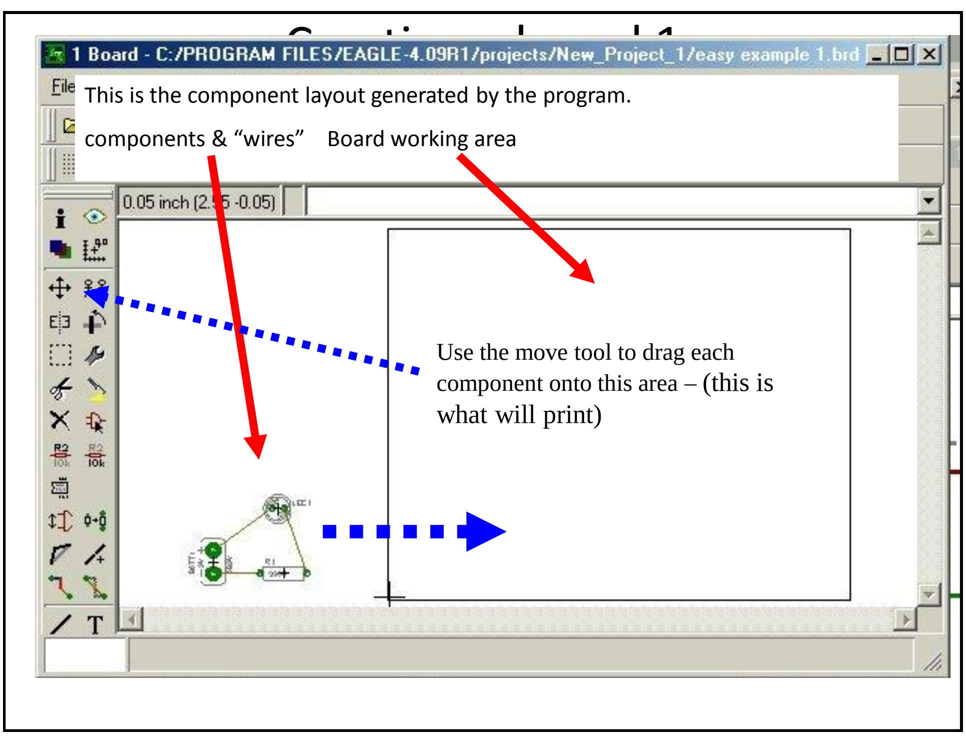


Run the Eagle program, when the control panel window appears click on File > Open >Project

. . then find the project folder containing your schematic.

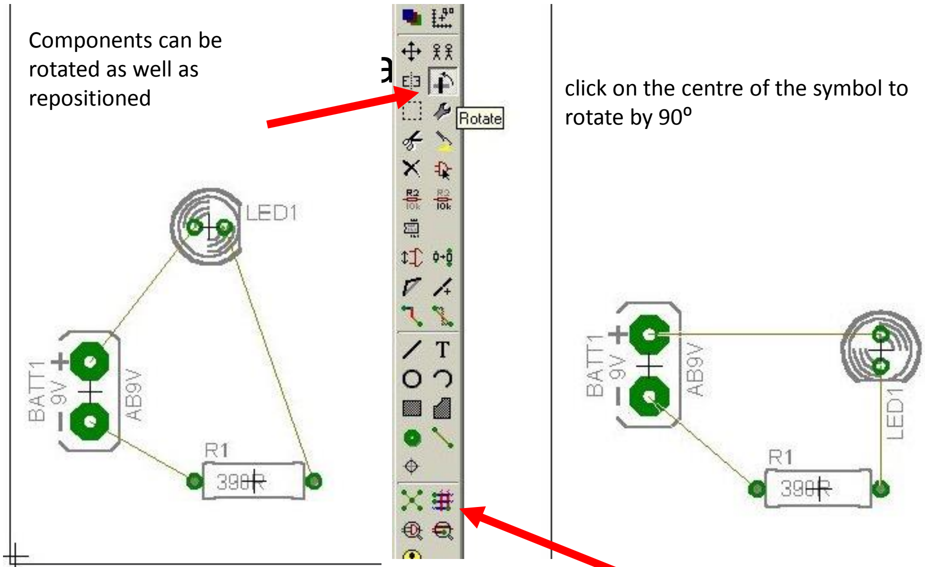
You can double click here <u>or</u> here to open the folder. Both actions will load the file for the schematic *(and board file if you have already created one)*


2 Schematic - C:/PROGRAM FILES/EAGLE-4.09R1/projects/New_Project_1/easy example 1.sch
File Edit Draw View Tools Library Options Window Help
New
Open...
Save
Save as...
Save all
Print setup...
Print...
CAM Processor
Switch to board
Export...
Script...
Run...
Close
Exit
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.
Warning
The board C:\PROGRAM FILES\EAGLE-4.09R1\projects\New_Project_1\easy example 1.brd does not exist.
Create from schematic?
select yes
Yes
No


1 Board - C:/PROGRAM FILES/EAGLE-4.09R1/projects/New_Project_1/easy example 1.brd
This is the component layout generated by the program.
components & “wires”
Board working area
Use the move tool to drag each component onto this area – (this is what will print)

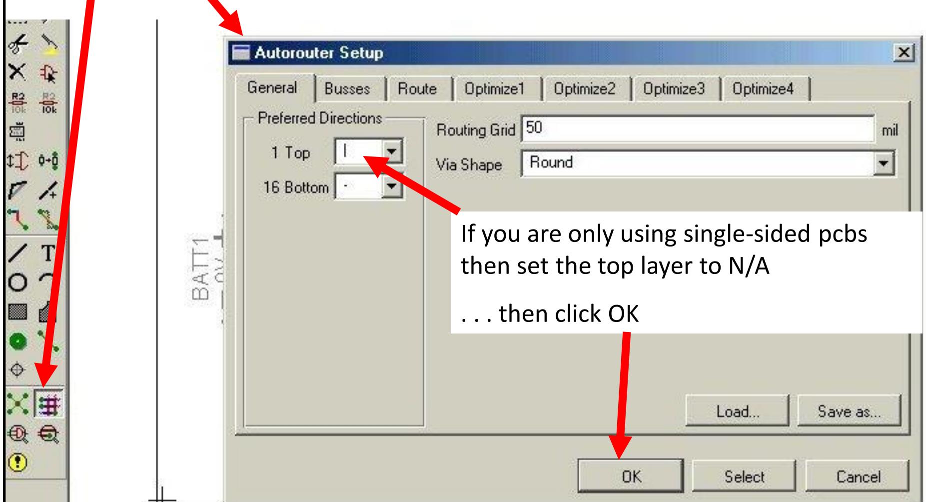
Components can be rotated as well as repositioned

click on the centre of the symbol to rotate by 90^o



The next stage is to create the tracks using “autoroute”

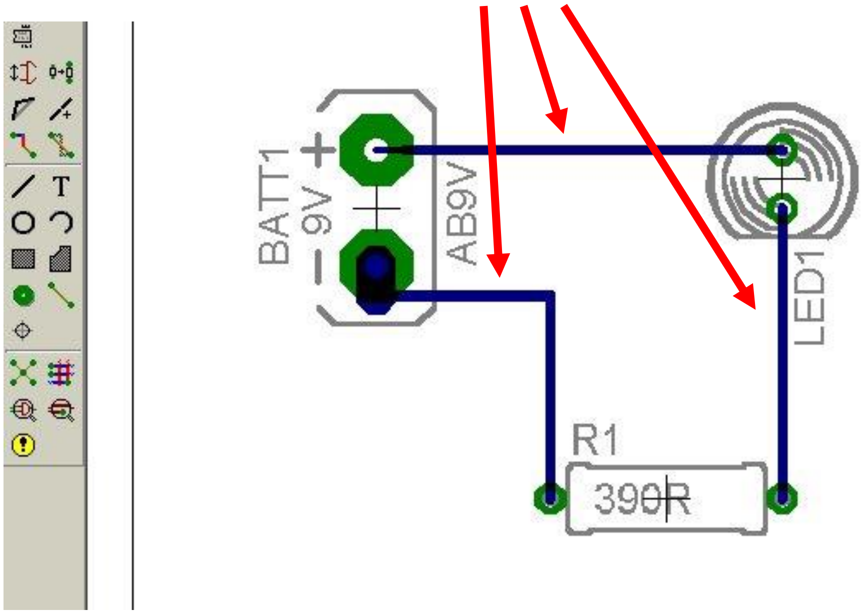
Clicking here generates an options menu. . .

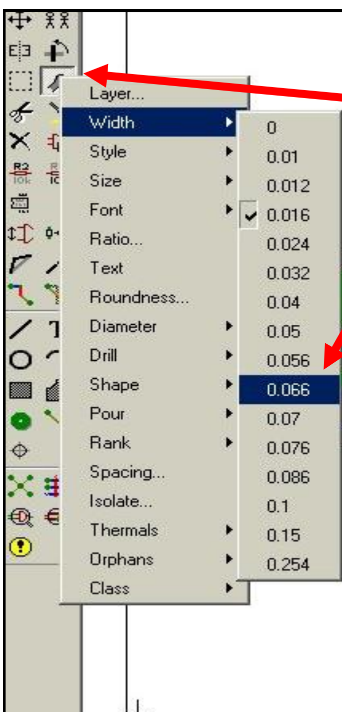


If you are only using single-sided pcbs then set the top layer to N/A

. . . then click OK

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



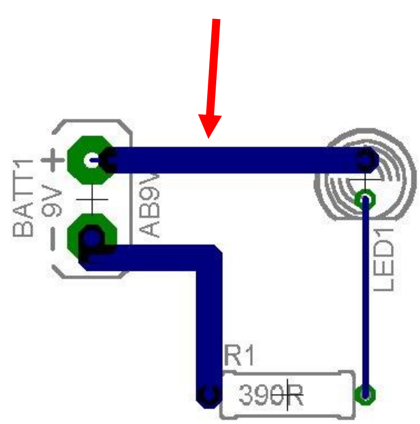


To change the track width,

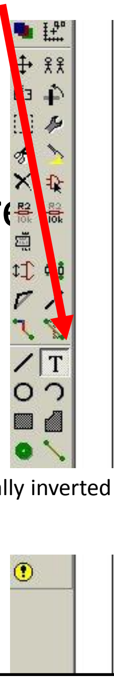
select **change > width**

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

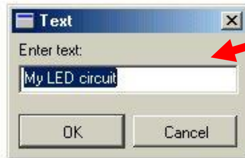
point at the track and right click.

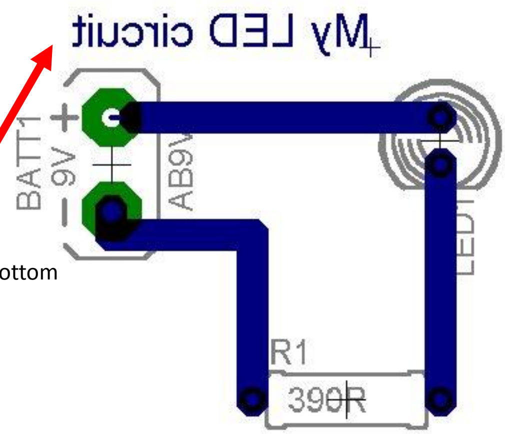


To add text select the text tool



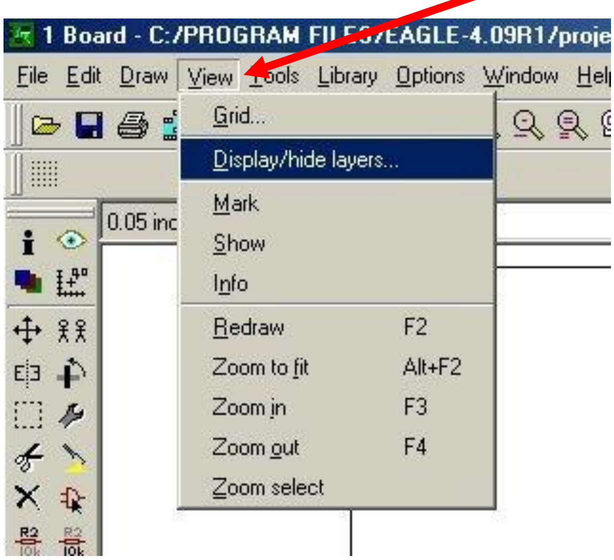
enter text in the box and place on the diagram

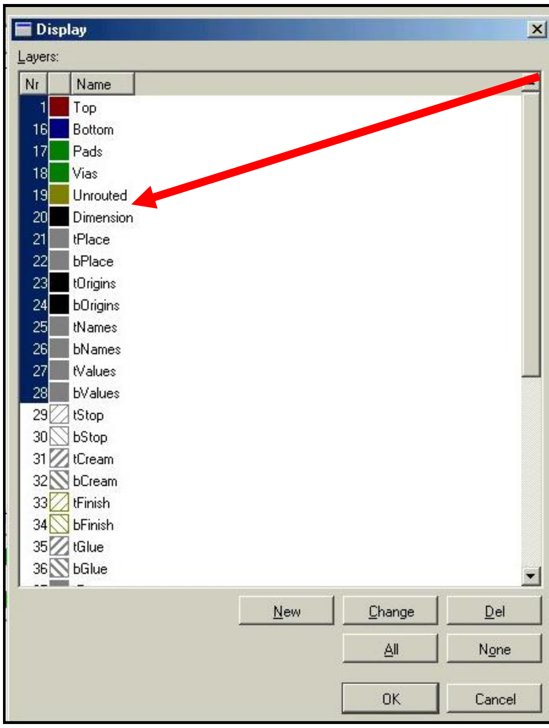




Text is automatically inverted if the "bottom layer" is chosen

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





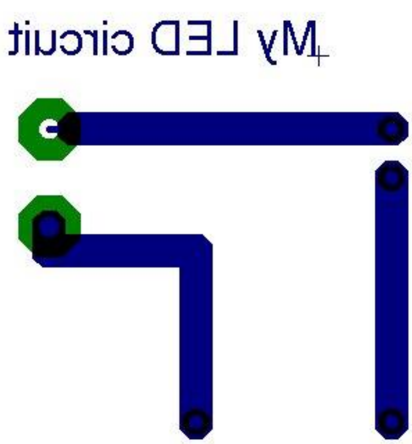
Here you can select what appears on screen and the printer. (blue = selected / white = deselected)

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

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

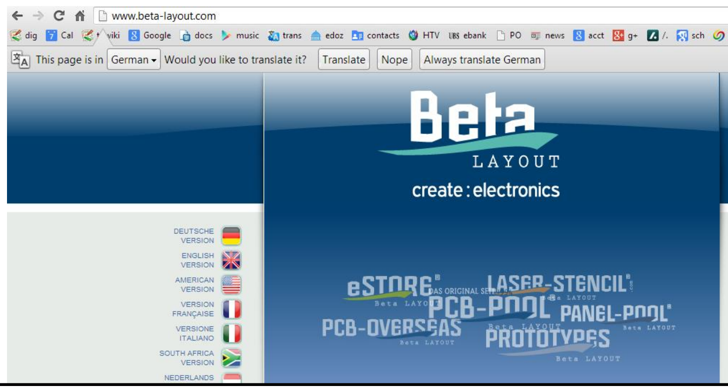
- **1: Setup uC tools, introduction**
- **2: Solder SMD AVR32 board**
- **3: Build application around AVR32**
- **4: Design your own PCB schematic**
- **5: Place and route your PCB. Fabricate your PCB at a production house**
- **6: Start logic design with FPGAs**

How do you get your PCB fabricated?

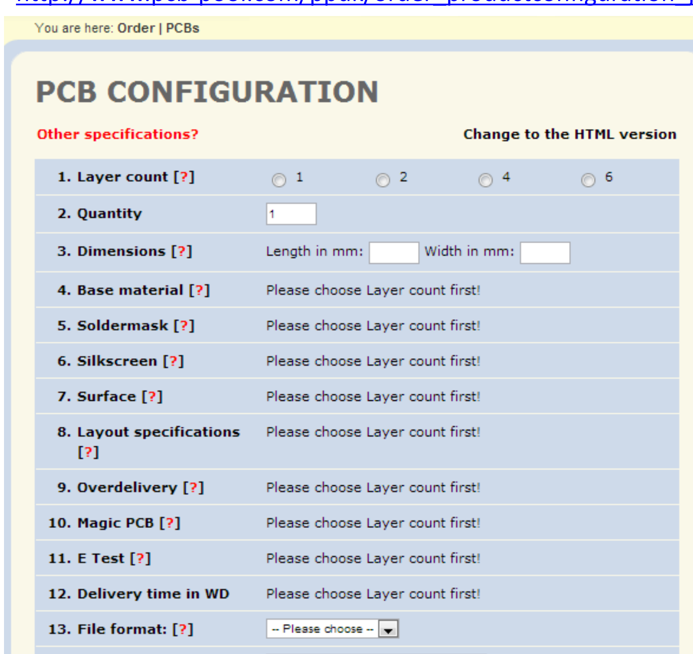
1. Have your own PCB router/driller machine.
   - Starts with blank PCB covered with metal, mills away all metal and places plugs for vias.
2. Or, send your board design away for fabrication

Example of having board fabricated

- You need to know
  1. Board dimensions in mm x mm
  2. Number of layers
  3. Basic rules such as min track width and spacing, minimum hole size



http://www.pcb-pool.com/ppuk/order_productconfiguration_js.html



You are here: Order | PCBs

PCB CONFIGURATION

Other specifications? Change to the HTML version

1. Layer count [?]	1 2 4 6
2. Quantity	1
3. Dimensions [?]	Length in mm: Width in mm:
4. Base material [?]	Please choose Layer count first!
5. Soldermask [?]	Please choose Layer count first!
6. Silkscreen [?]	Please choose Layer count first!
7. Surface [?]	Please choose Layer count first!
8. Layout specifications [?]	Please choose Layer count first!
9. Overdelivery [?]	Please choose Layer count first!
10. Magic PCB [?]	Please choose Layer count first!
11. E Test [?]	Please choose Layer count first!
12. Delivery time in WD	Please choose Layer count first!
13. File format: [?]	-- Please choose --
14. Project name:	

PCB CONFIGURATION

Other specifications? — Change to the HTML version

1. Layer count [?]	○ 1 ◉ 2 ○ 4 ○ 6
2. Quantity	2
3. Dimensions [?]	Length in mm: 100 Width in mm: 100
4. Base material [?]	◉ FR4, 35µmCu, 1,6mm ○ FR4, 35µmCu, 1,0mm
5. Soldermask [?]	◉ yes ○ no
6. Silkscreen [?]	◉ yes, only top ○ yes, only bottom ○ yes, top and bottom ○ no
7. Surface [?]	◉ ENIG (Electroless Nickel Gold) for ultra-flat pads ○ HAL (Hot Air Leveling - lead-free and RoHs compliant)
8. Layout specifications [?]	Min. track / gap size: ○ >= 0.125mm (5mil) ◉ >= 0.150mm (6mil) Min. drill-end diameter: ○ >= 0.2mm (8mil) ◉ >= 0.3mm (12mil)
9. Overdelivery [?]	○ yes, if available ◉ no ○ yes, if available as Magic-PCB® - with free embedded RFID chip and win a Reader/Writer-Kit!

16. Assembly [?]

○ yes ◉ no

Qty:

- 4 PCBs to be assembled
- SMT components per PCB [?]
- THT components per PCB [?]

Component assembly

- ◉ single sided
- ○ double sided

Components [?]

- ◉ will be provided by me
- ○ will be partial provided by me, the rest need to be ordered *
- ○ need to be ordered for me *

* The components will be ordered according to the supplier order numbers you have specified. We will bill you at the supplier's current list price (without markup).

PCB assembly delivery time

12 working days after completion of PCB production and receipt of the components ordered for you or provided by you

PCB assembly price

167.79 EUR excluding component costs

17. FITS-or-NOT number [?]

A small hint: Instead of using the mouse to change between fields, you can fill out the form faster using the tabulator and the arrow keys.

Your PCB

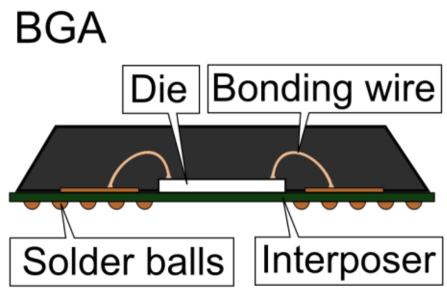
Required number: 4

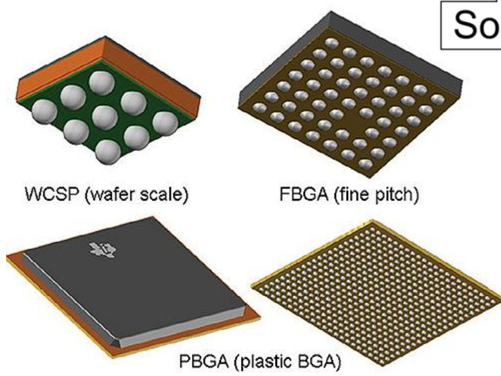
	Net EUR	Gross EUR
Price per piece	36.18	44.50
Total order value	**144.71**	**178.00**

... into the shopping basket

... price comparison — ... more for less!

Ball Grid Array (BGA)

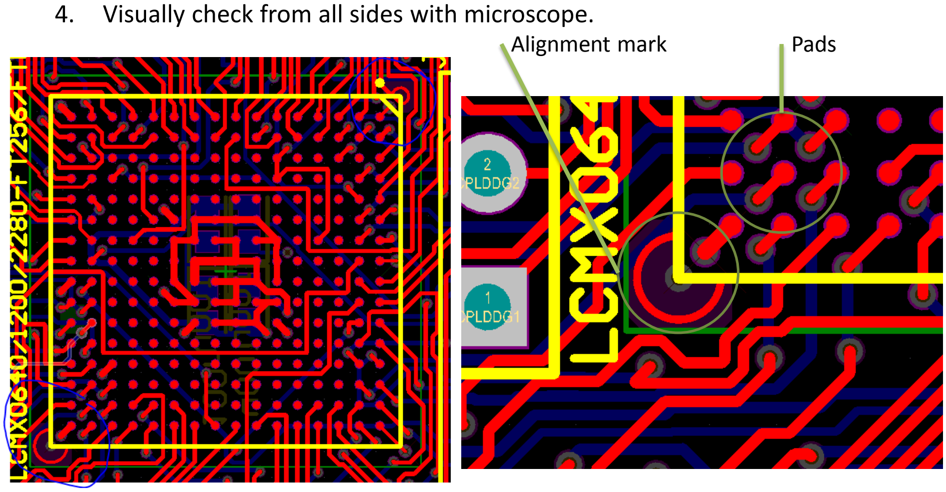




You **can** solder these parts using a reflow oven

Soldering BGA yourself in reflow oven

1. Apply thin layer of flux (**not** solder paste) to board.
2. Carefully place BGA part using your alignment marks (drawn on metal, not silkscreen!).
3. Reflow part in oven.
4. Visually check from all sides with microscope.



Reflow oven

- Applies temperature profile to slowly heat parts to avoid stress and allow volatile gases to escape, **activates flux**, then **reflows solder**
- Hint: Use **gold finish on pads** if you ever want to rework boards because gold does not oxidize

Beta Layout reflow oven plus controller
200 EUR – oven plus controller.