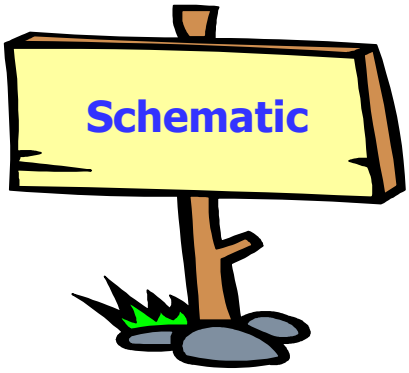
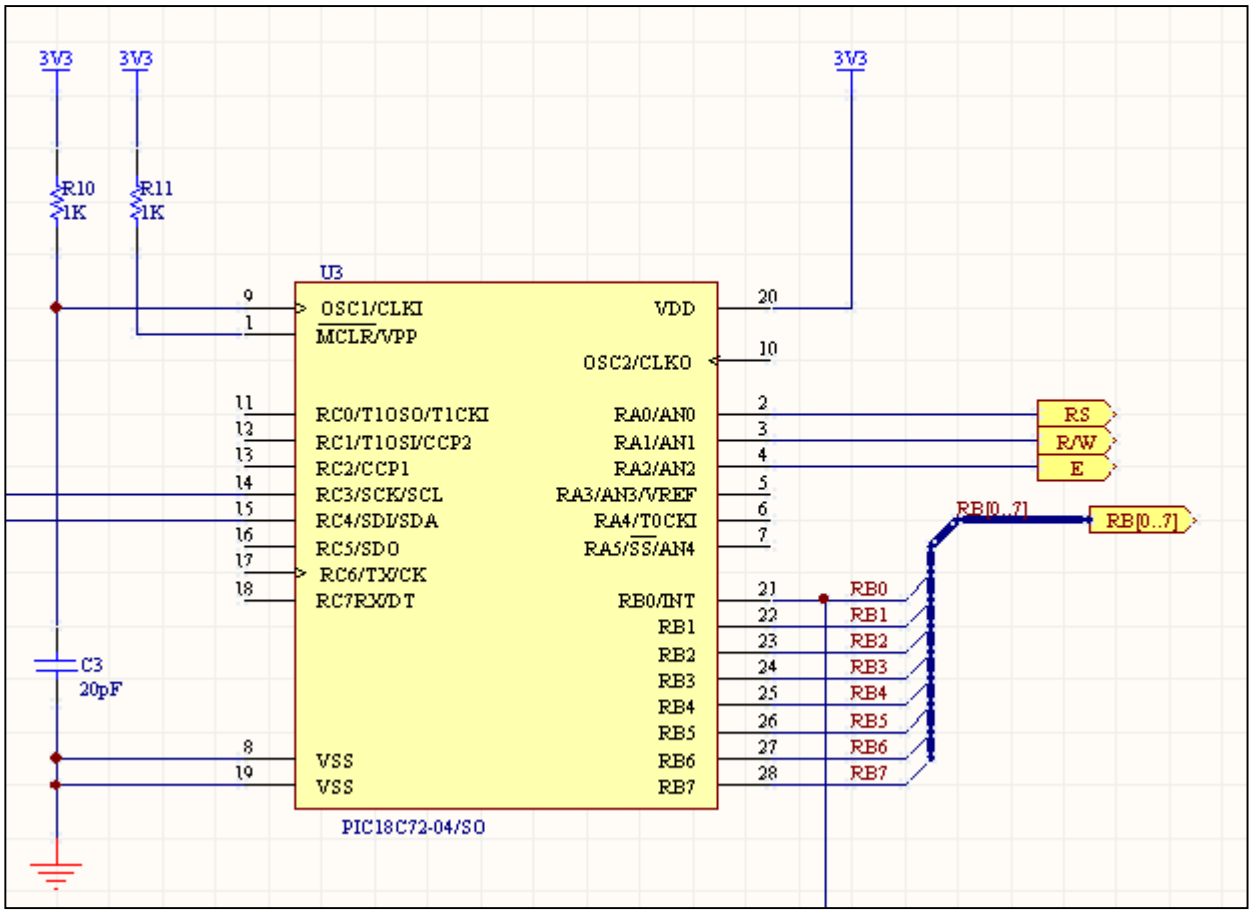


Prepare by : HK Sim simhkeng@gmail.com

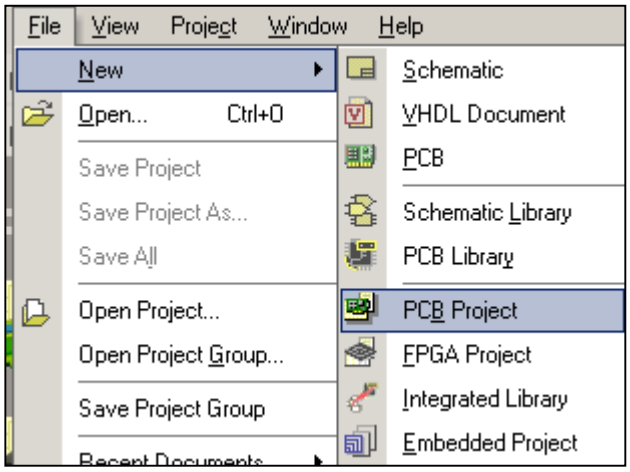
Date : 22 Dec 05



Prepare by : HK Sim simhkeng@gmail.com

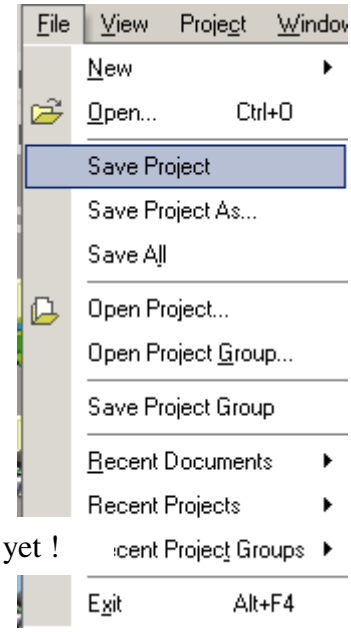
Start a new PCB project

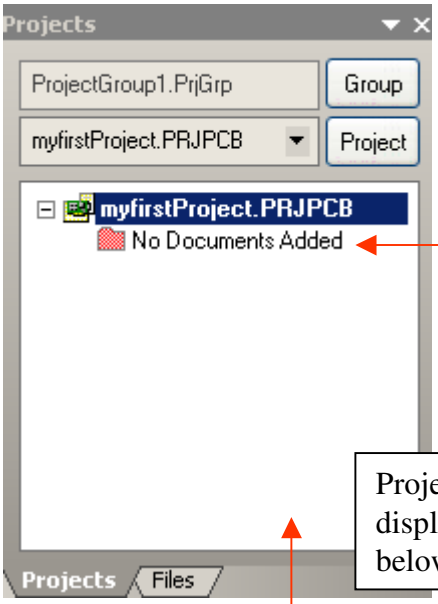
[File] / [New] / [PCB Project]



[File] / [Save Project]

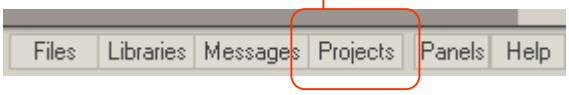
A pop-up menu will appear to enter the project name and the location of the project folder. Enter the project name accordingly and this will appear on the project pop-up menu





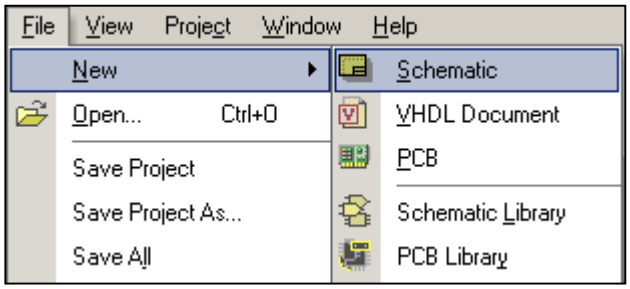
No design file yet !

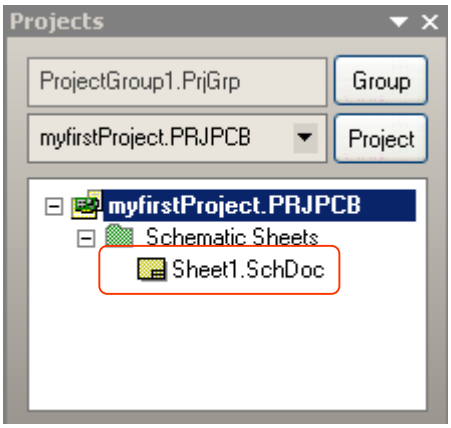
Project pop-up menu which can be display when click project menu below



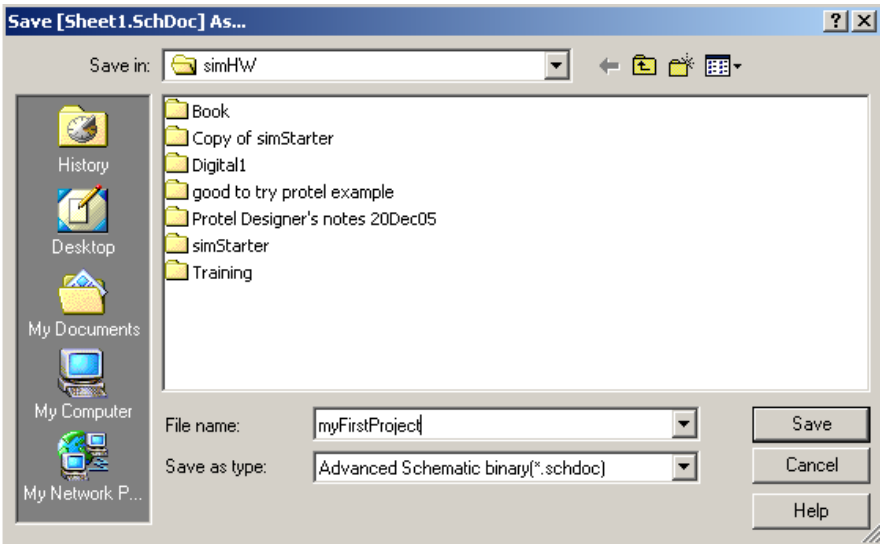
Prepare by : HK Sim simhkeng@gmail.com

Add new schematic page into the project





→ Save the schematic file

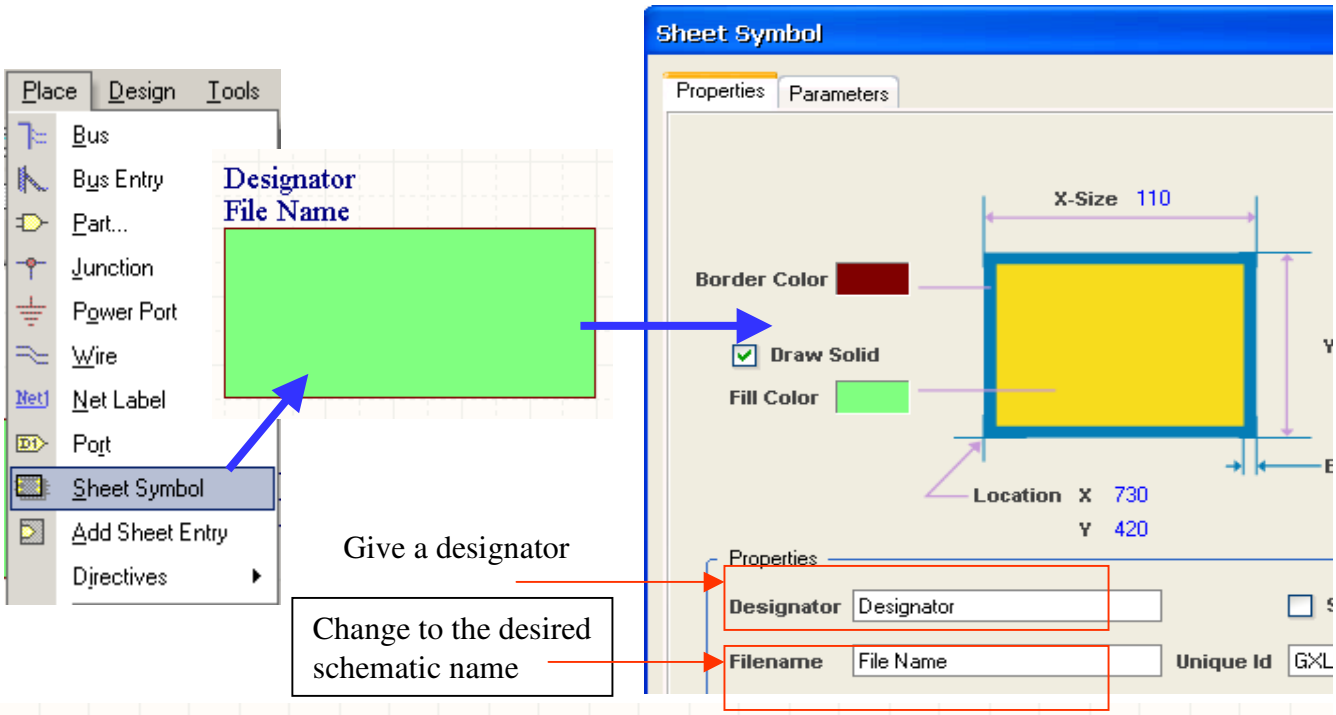


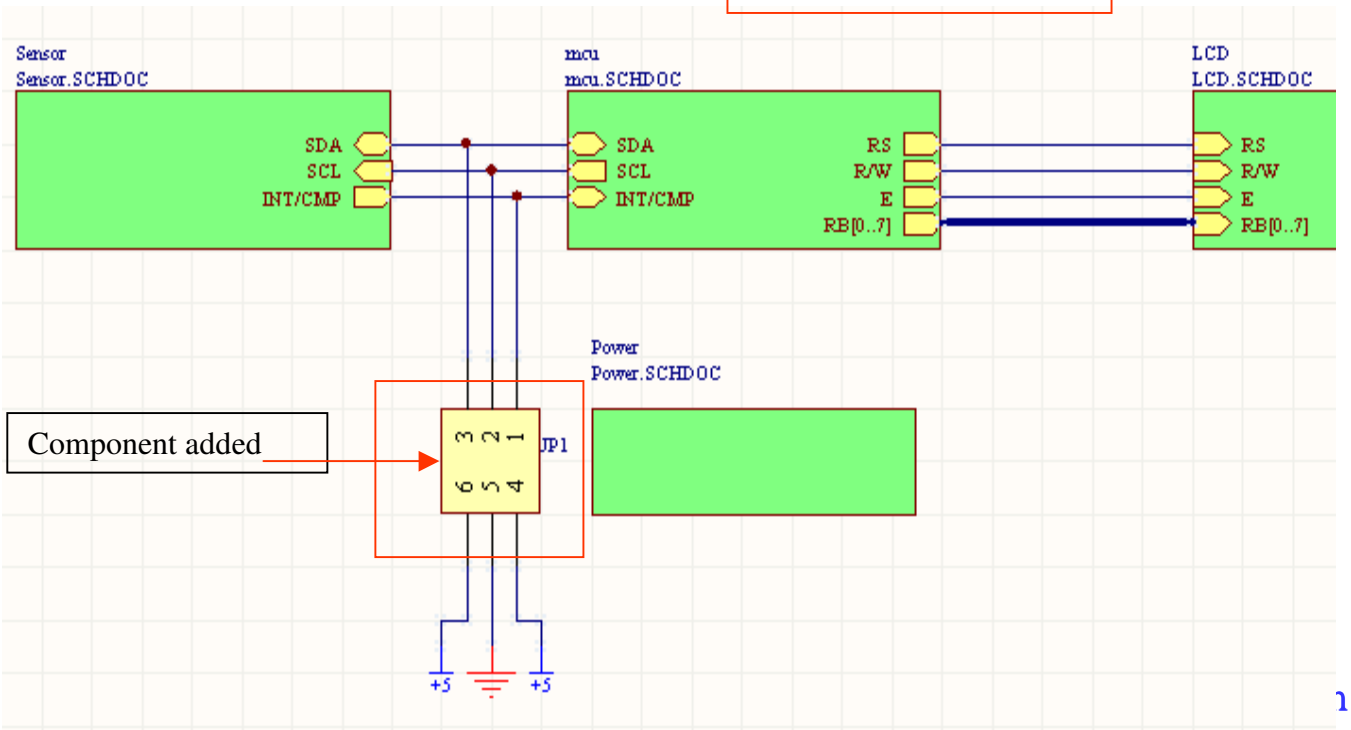
Prepare by : HK Sim simhkeng@gmail.com

Schematic Design concept – Multi-sheet

Under one project,

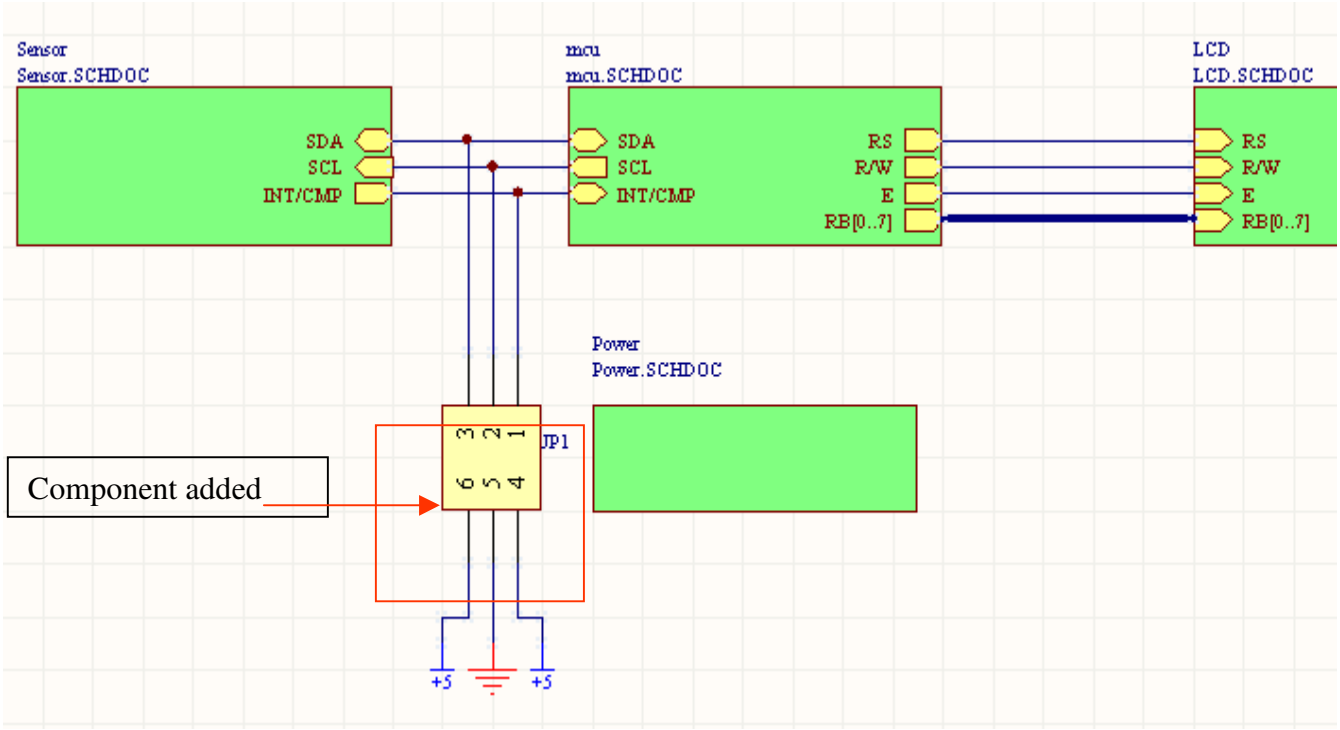
- Create each individual block of circuit on individual schematic sheet.
- For each schematic sheet, use **NetLabels** or **Ports** for input and outport
- Create a schematic named **main.schdoc** and using **Sheet Symbol** to link all the individual circuit's schematic together via the Netlabels or Ports.
- Components can be added to the main sheet too.





Link multi schematic sheet together

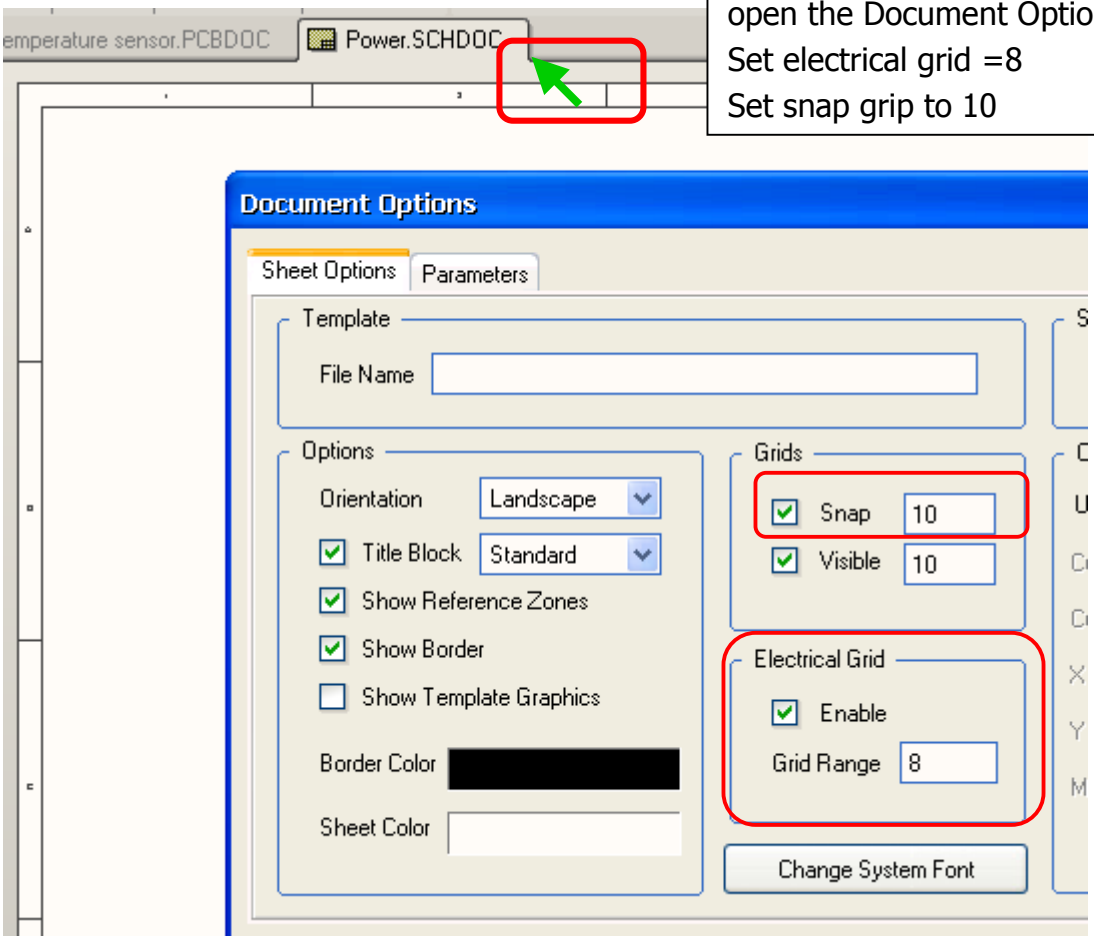
Sheet symbols are used to show which sub sheets link to the top sheet. Each symbol represents a sub-sheet and has a filename attribute that provides the link between the top sheet and the sub sub-sheet with that filename.



Prepare by : HK Sim simhkeng@gmail.com

Set Grid for design

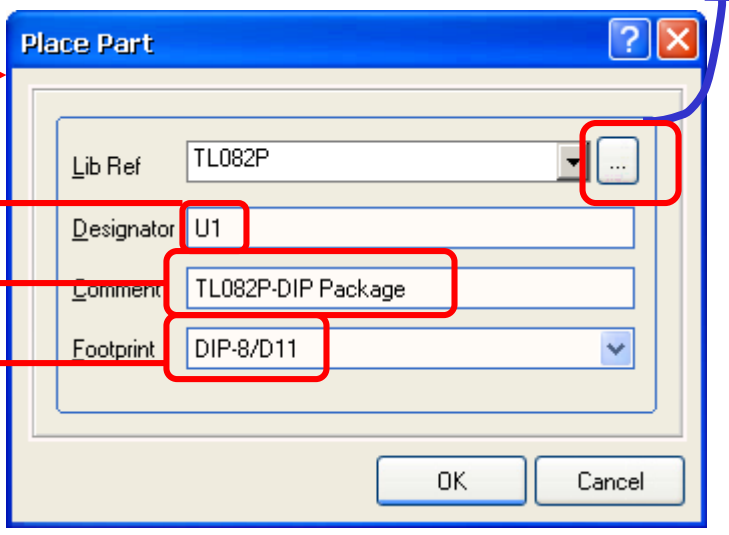
Double click the sheet border to open the Document Option Dialog.
Set electrical grid =8
Set snap grip to 10



Add component to schematic

To search more part , more libraries

[Pace] / [Part]



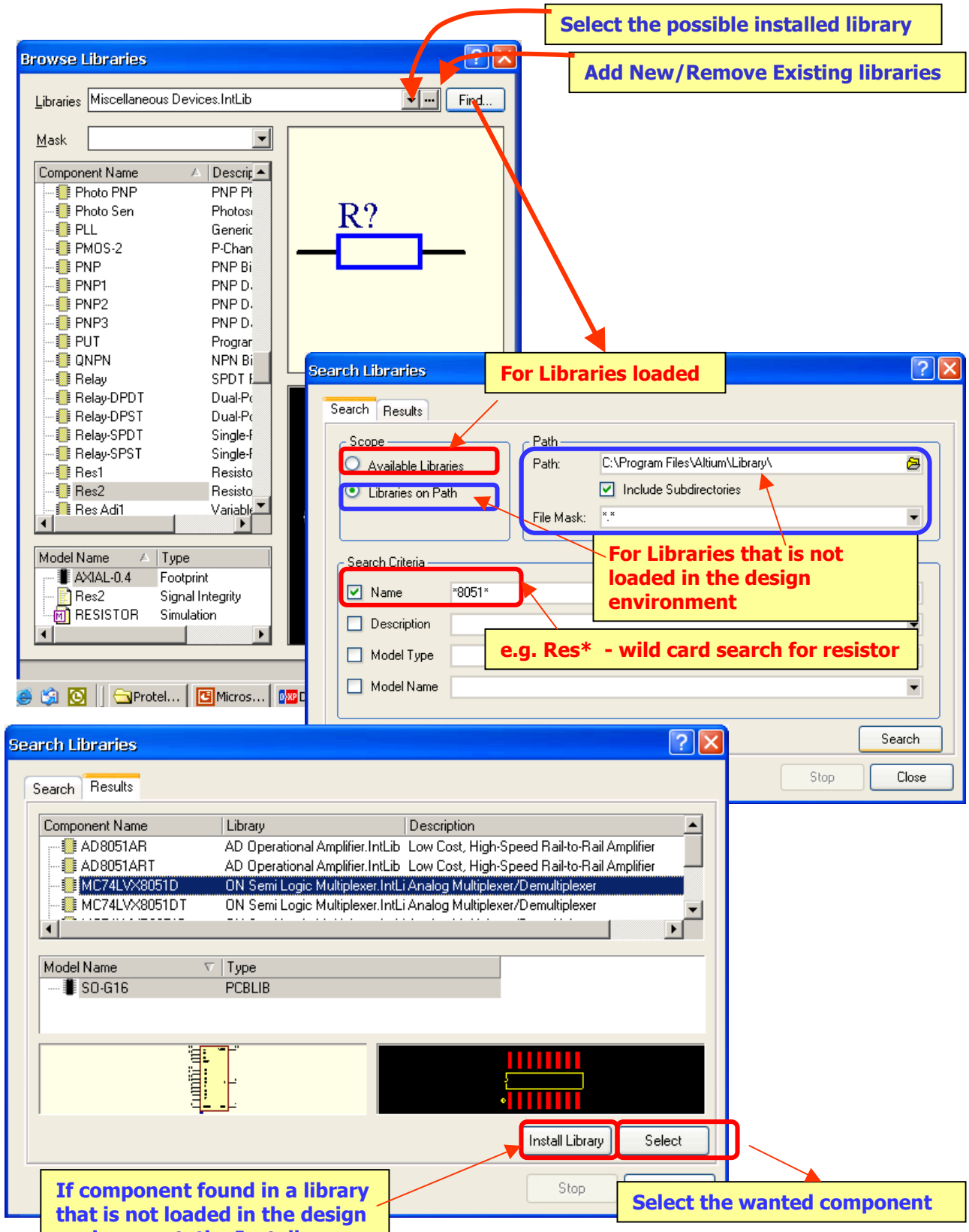
Enter component ID

Add comment

component foot print, Come with the component

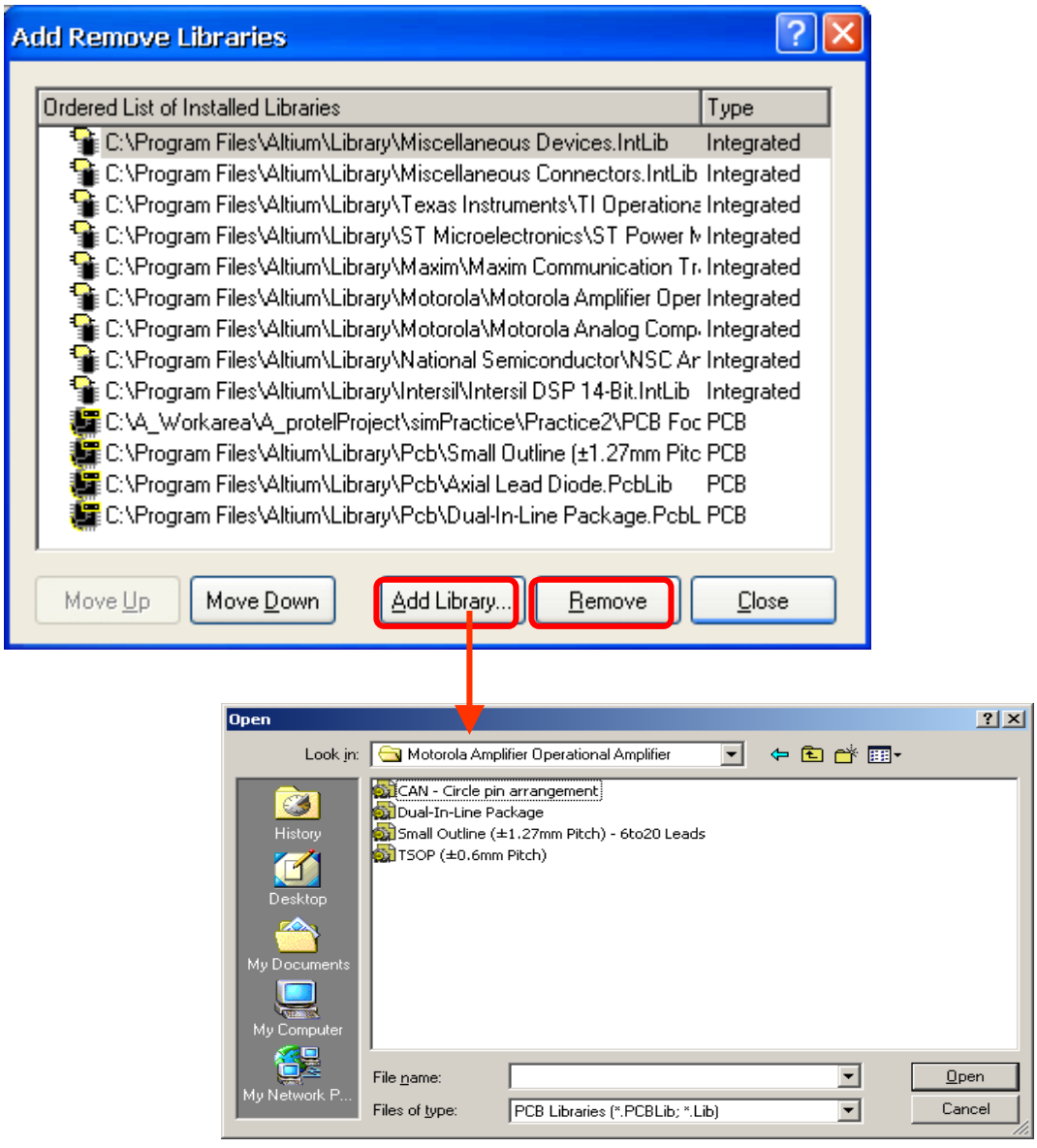
Prepare by : HK Sim simhkeng@gmail.com

Find Part from libraries



Prepare by : HK Sim simhkeng@gmail.com

Libraries that wanted to be use have to be loaded to the working environment.
DXP's library directory is c:\program Files\Altium\Library\<library folders>

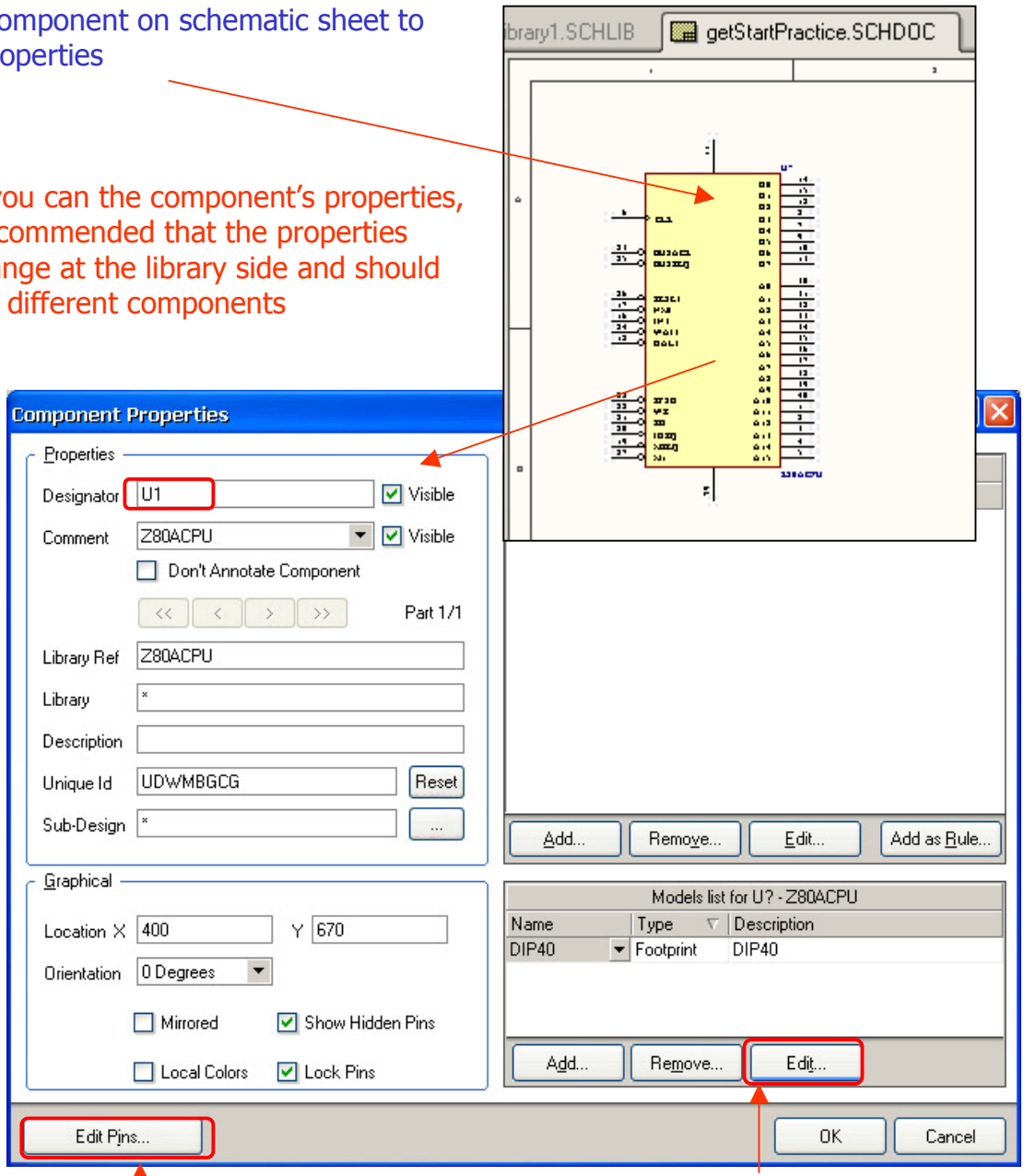


Prepare by : HK Sim simhkeng@gmail.com

Find Part from libraries

Double click component on schematic sheet to change the properties

Although you can the component's properties, but it is recommended that the properties should change at the library side and should name it as different components



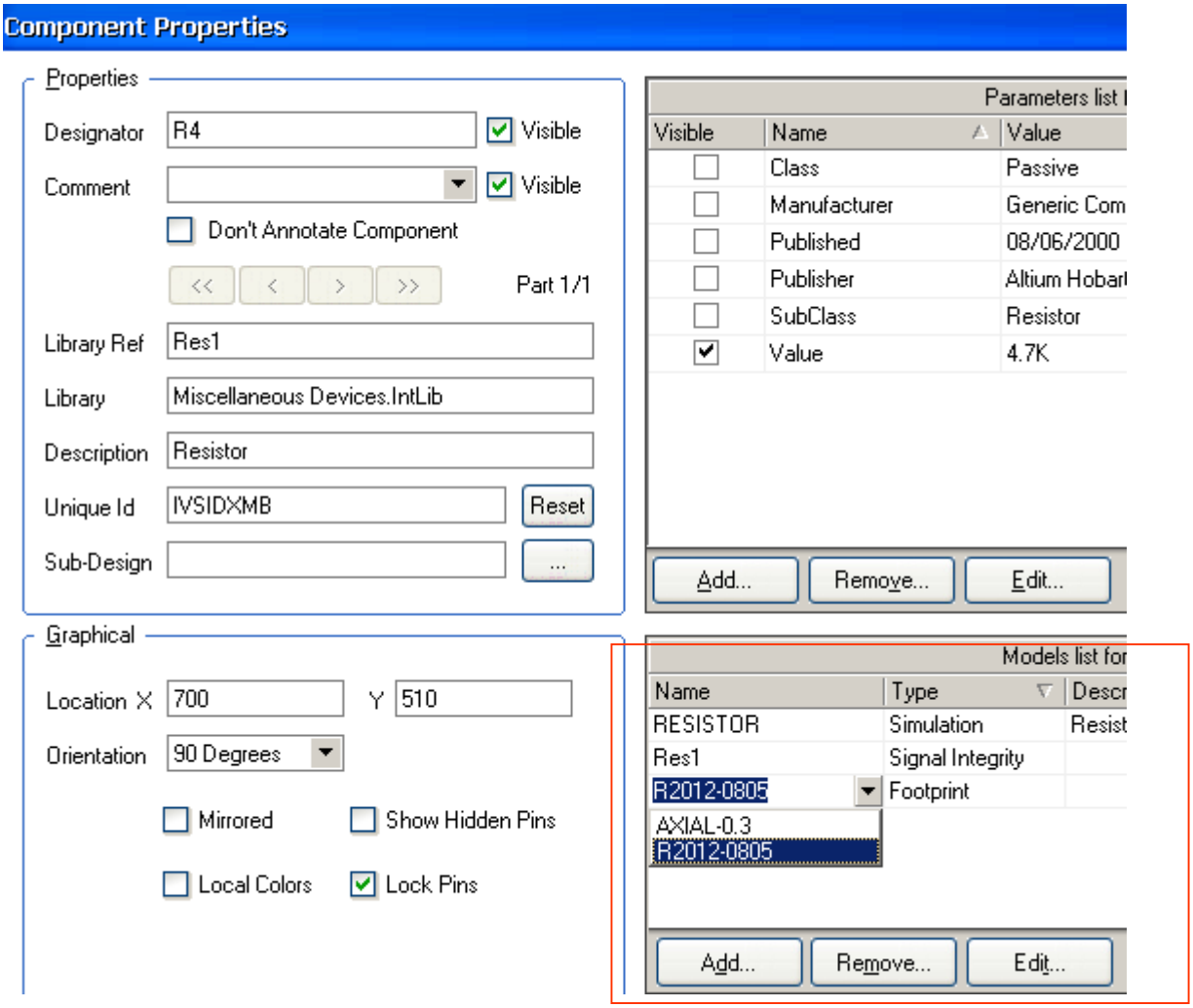
Check to make sure foot print is correct

Edit schematic's component

Prepare by : HK Sim simhkeng@gmail.com

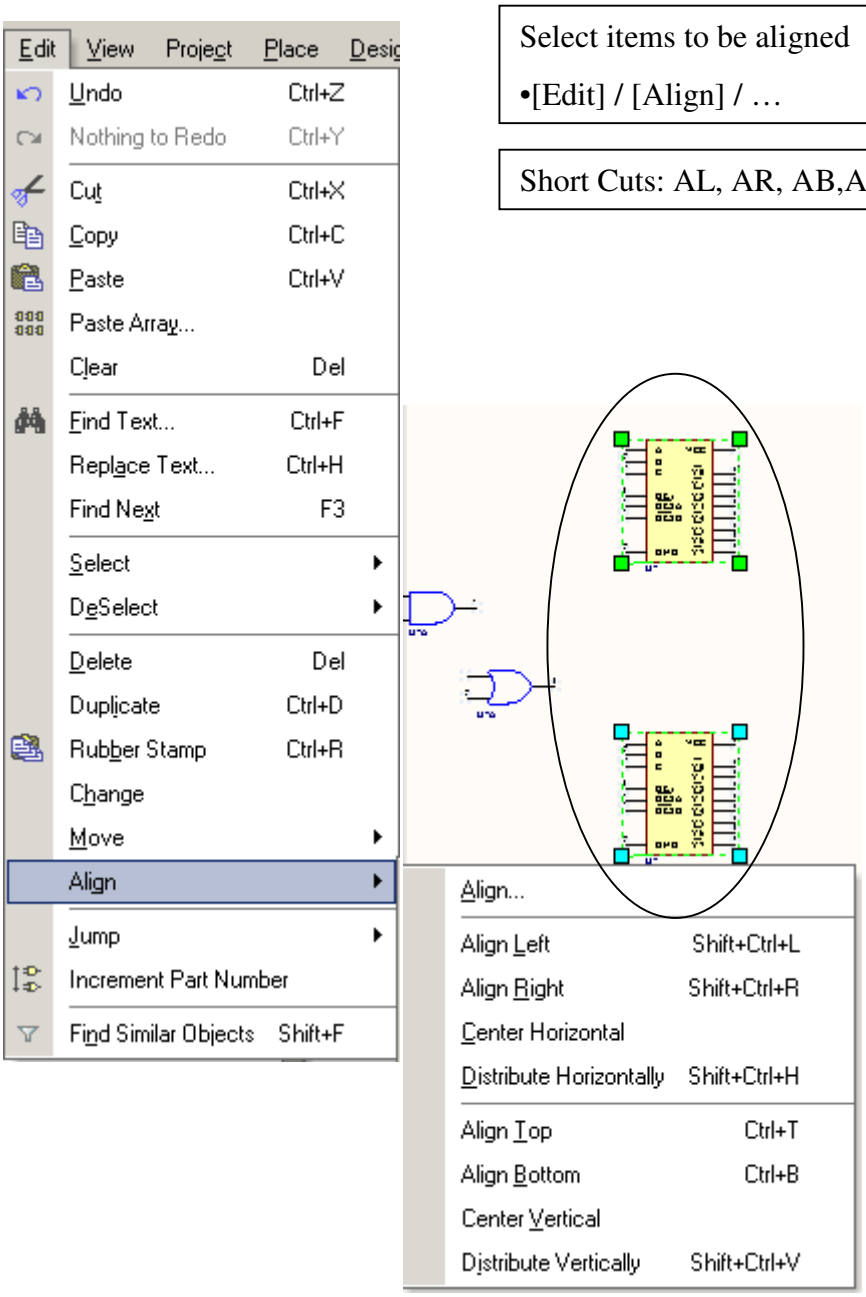
Add component foot prints

More than one Component foot can be assigned to one component. Double click component to pop-up the **component properties** and new foot print can be added or changed.



Prepare by : HK Sim simhkeng@gmail.com

Alignment of Parts



Select items to be aligned

•[Edit] / [Align] / …

Short Cuts: AL, AR, AB,AT

Prepare by : HK Sim simhkeng@gmail.com

Draw Connections

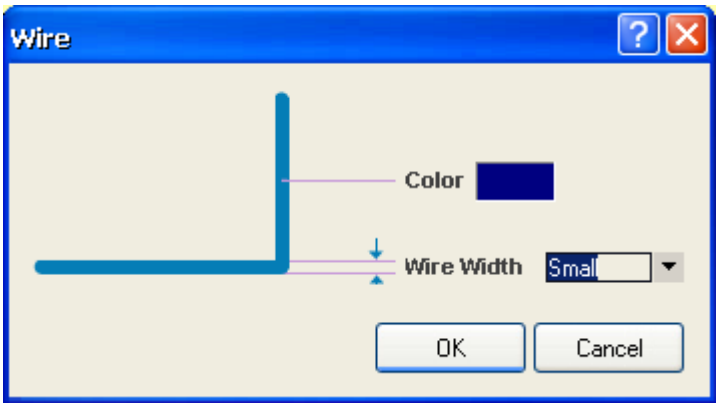
[Place] / [Wire]  PW

Change direction of wire : Shift + Space Bar [Shift] + [Space Bar]

Change … ? : Space Bar

Release the last portion of wire : Backspace BackSpace

Change property of wire : double click wire



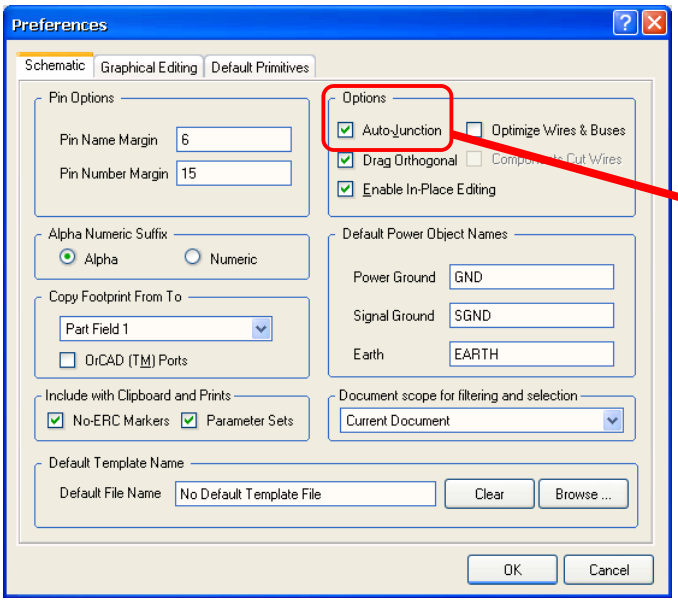
Auto Junction

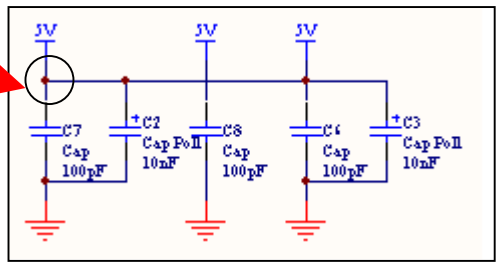
 PJ

During a line connection between components, you may need to cross over lines. Cross over line is not necessary connected to that line as in figure xx. When reaches the end and do a right click, then a junction is automatically form. This provided you set the Auto-Junction mode.

[Tool / Preference]

You can force to place a junction by using Short cut "PJ"





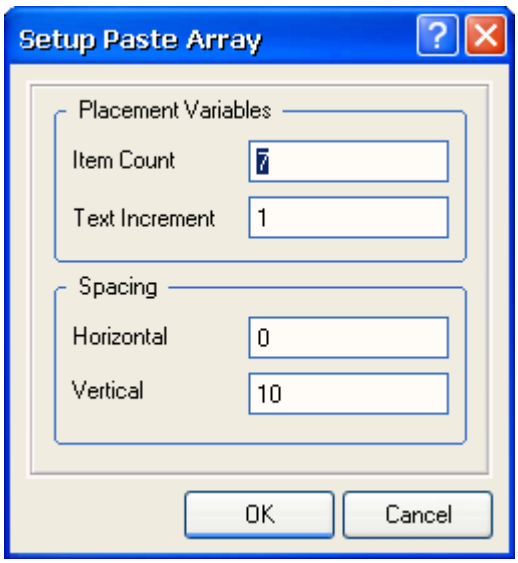
Prepare by : HK Sim simhkeng@gmail.com

Place Duplicate Items

EY

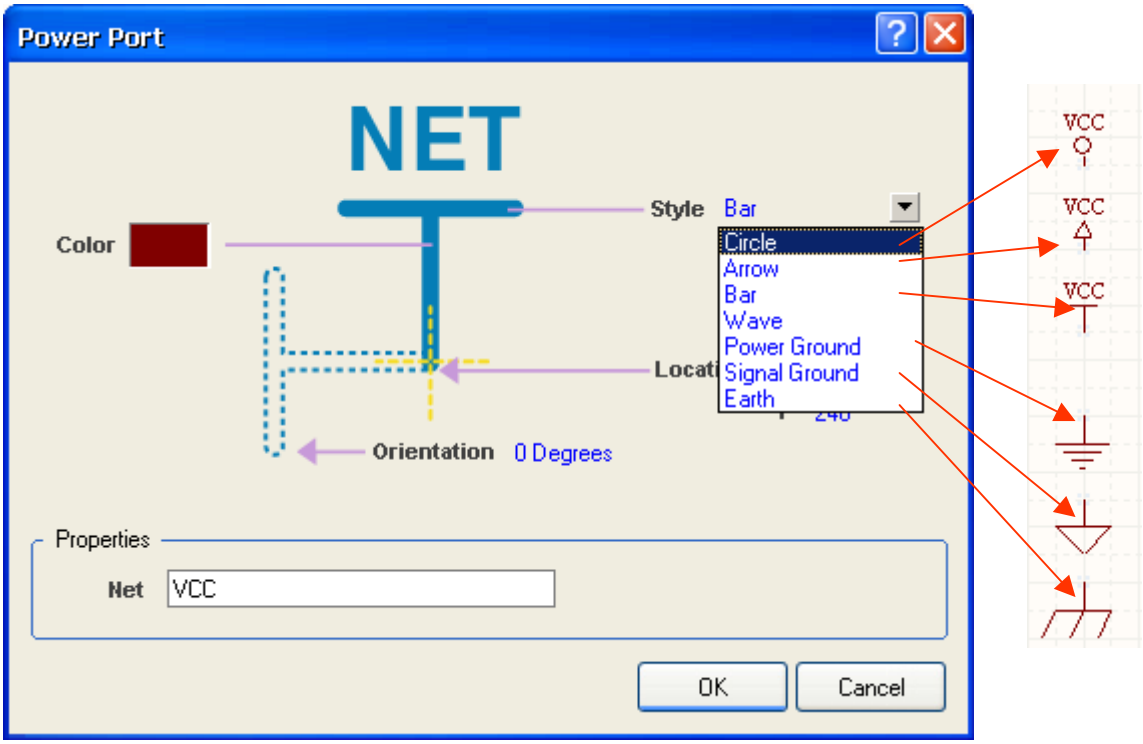
Copy the item to be duplicate , wire, NetLabel … etc

Use EDIT/PlaceArray , define the item count , OK and then move to the location and Click on it.



To select multiple components

Create Power Port

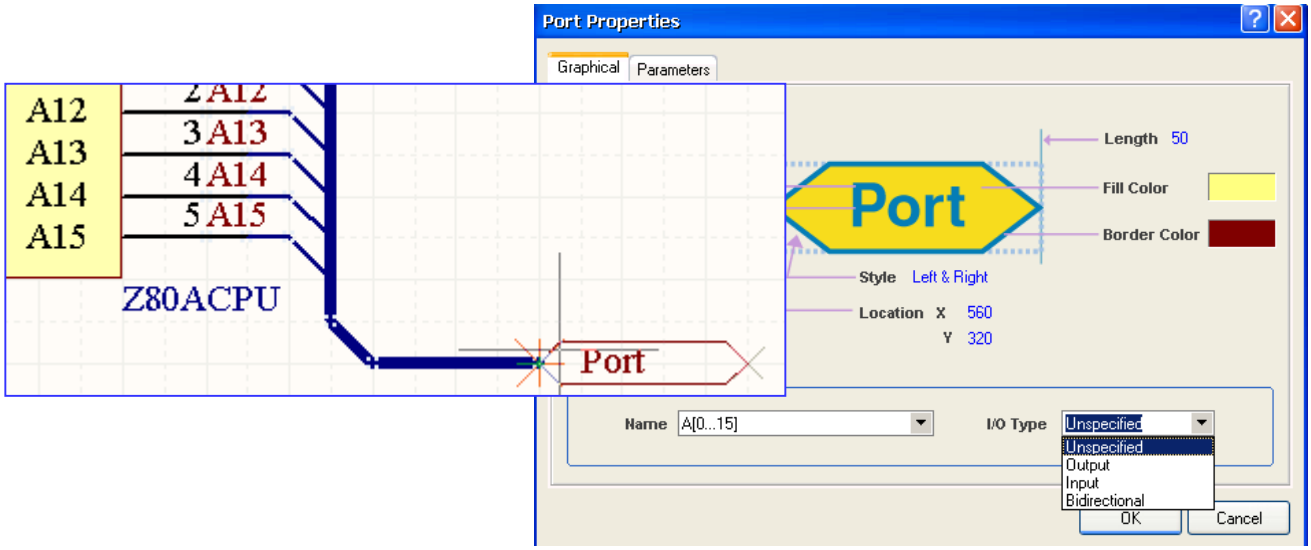


Prepare by : HK Sim simhkeng@gmail.com

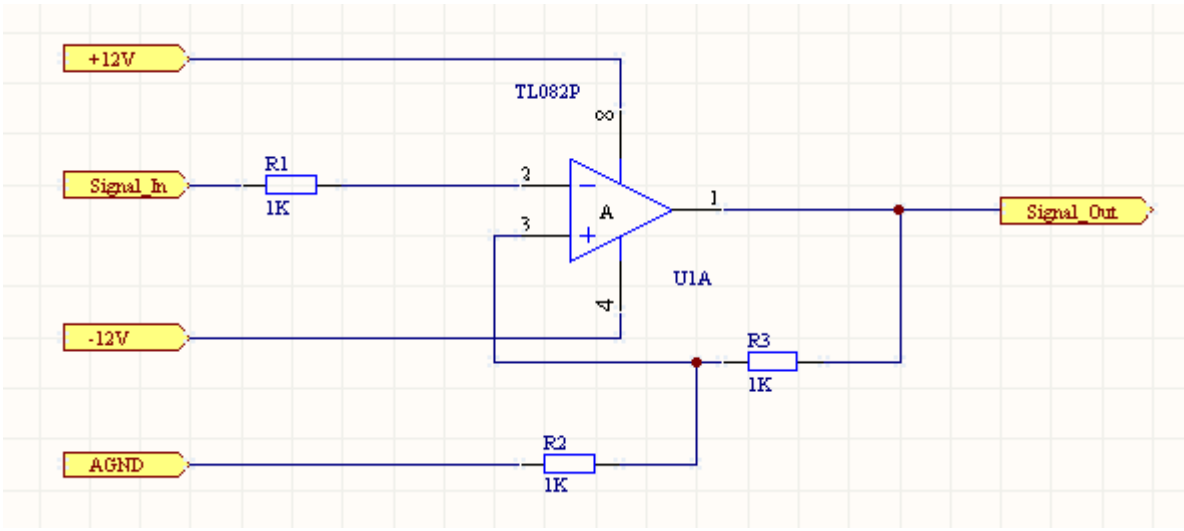
Place Port

Place / Port, move the port to the location and anchor the two points.

Double click to Popup the property . Enter Port Name and Select IO type.

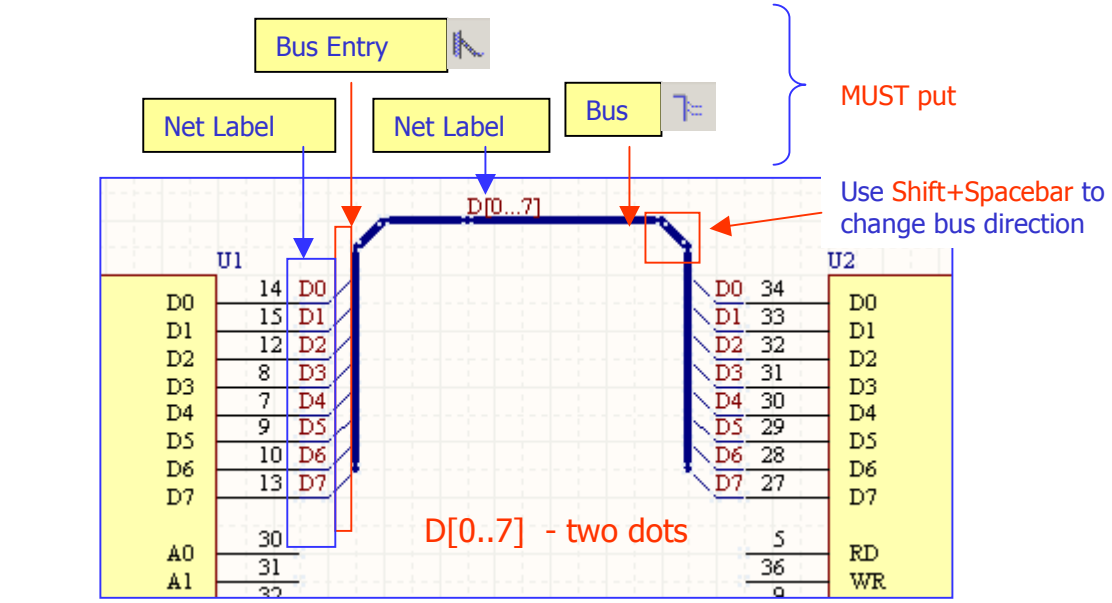


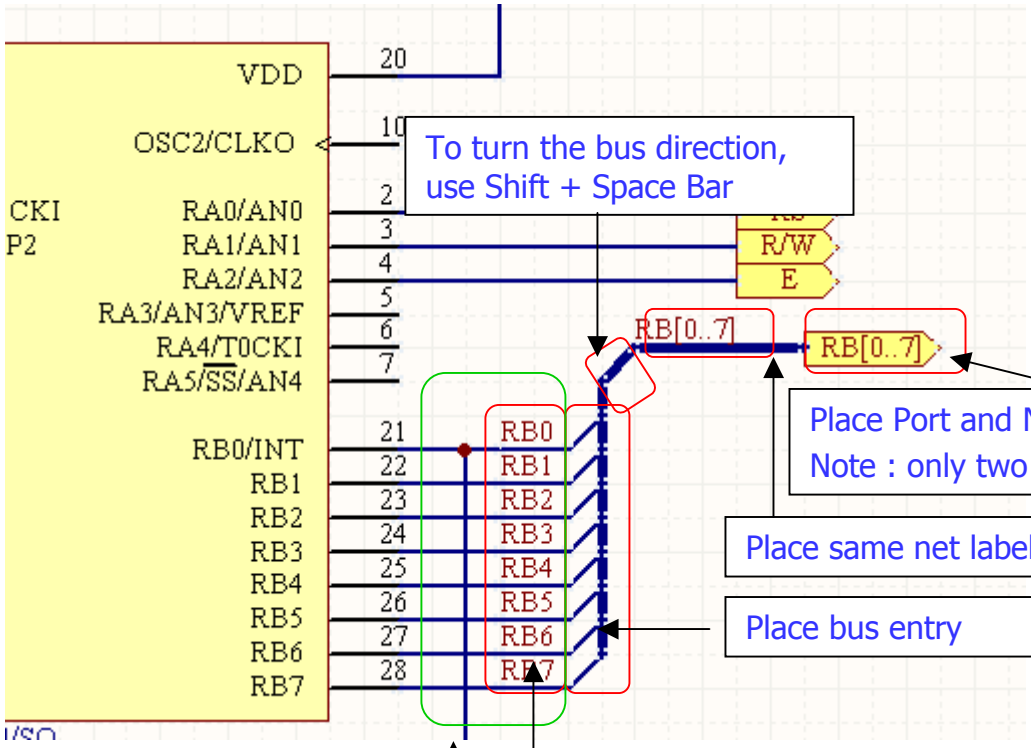
Data bus, Address bus declare as Bidirectional
Port type = Output will be connected to Input port of the component
Port type =Input will be connected to output port of the component



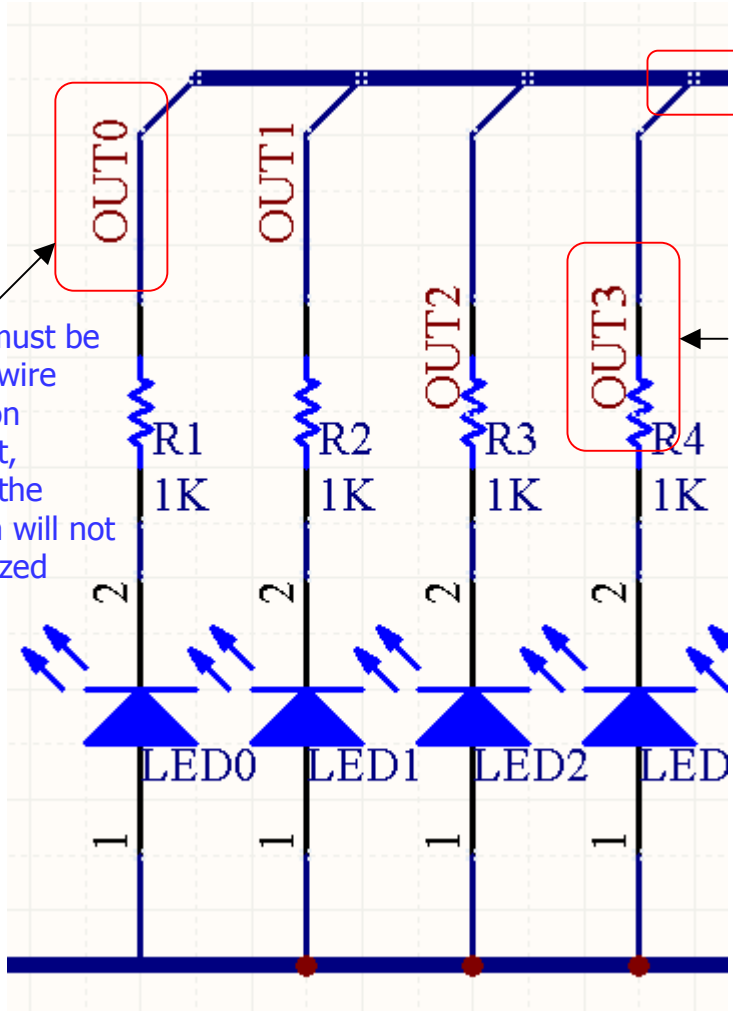
Prepare by : HK Sim simhkeng@gmail.com

Create Bus Connection





Prepare by : HK Sim simhkeng@gmail.com



When bus entry is connected to a bus, there will be 4 white dots appears

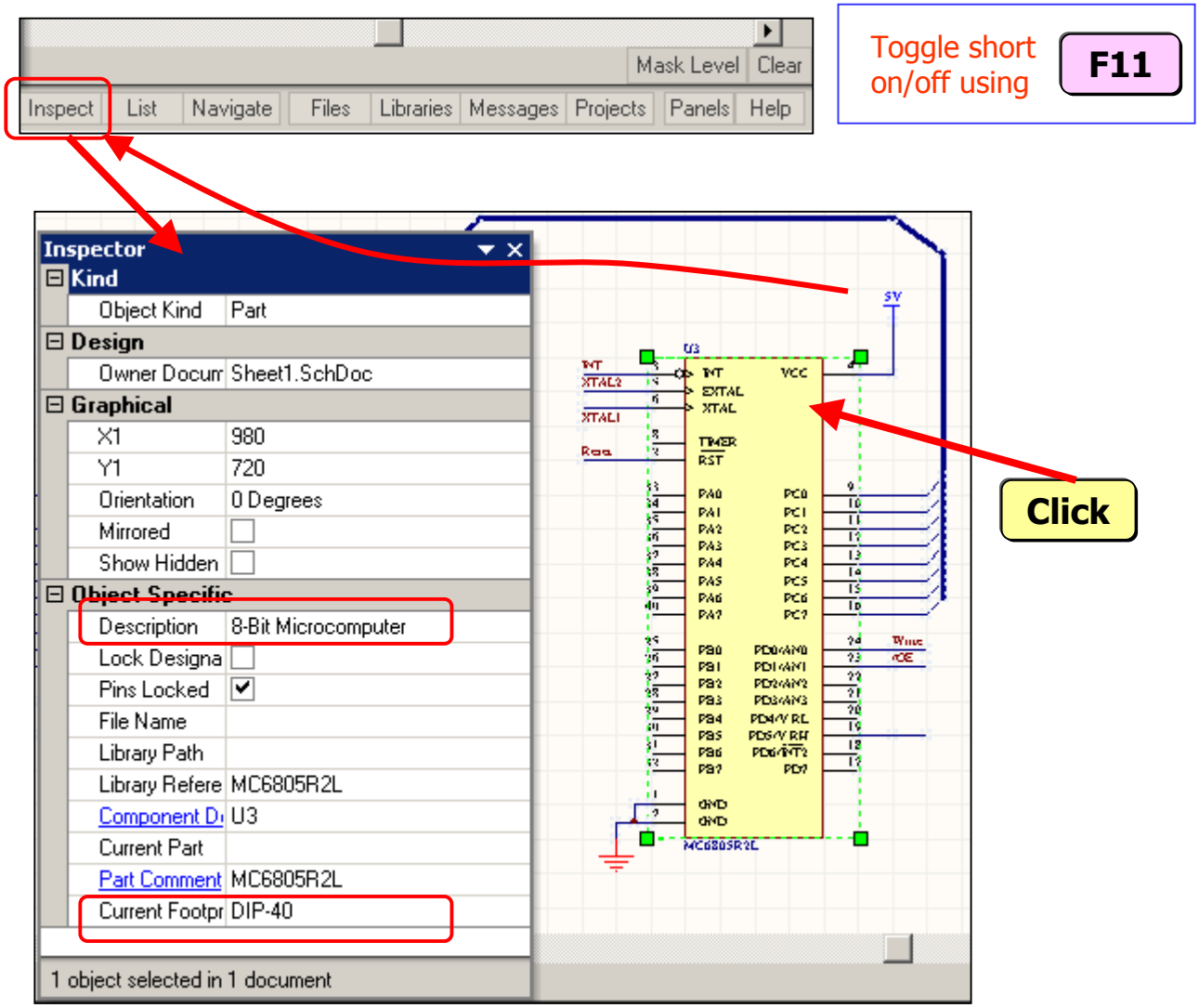
Net label OUT3 is not recognized and therefore one pin of R4 is not connected

Net label must be placed on wire and NOT on component, otherwise the connection will not be recognized

Prepare by : HK Sim simhkeng@gmail.com

Inspector component's properties

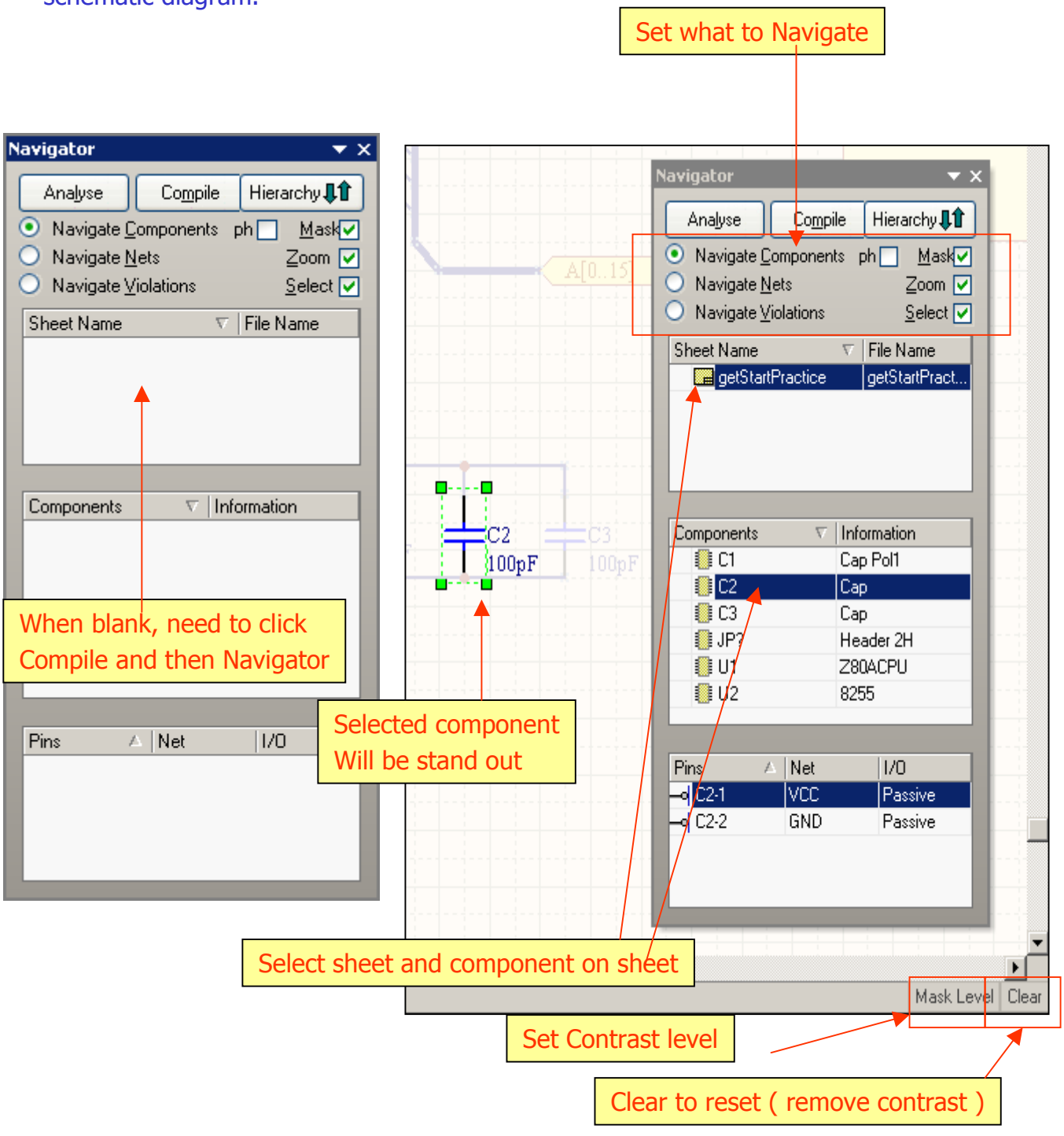
To inspect a particular component on the schematic, click to select the component and then click the Inspector item to pop-up the component information



Prepare by : HK Sim simhkeng@gmail.com

View connection of a component - **Navigator**

Navigator is use to look at the connection of a particular component . The selected component will be highlighted and the rest of component will be mask out. Upon click the Navigator interface, a pop-up menu showing all the component on the schematic diagram.



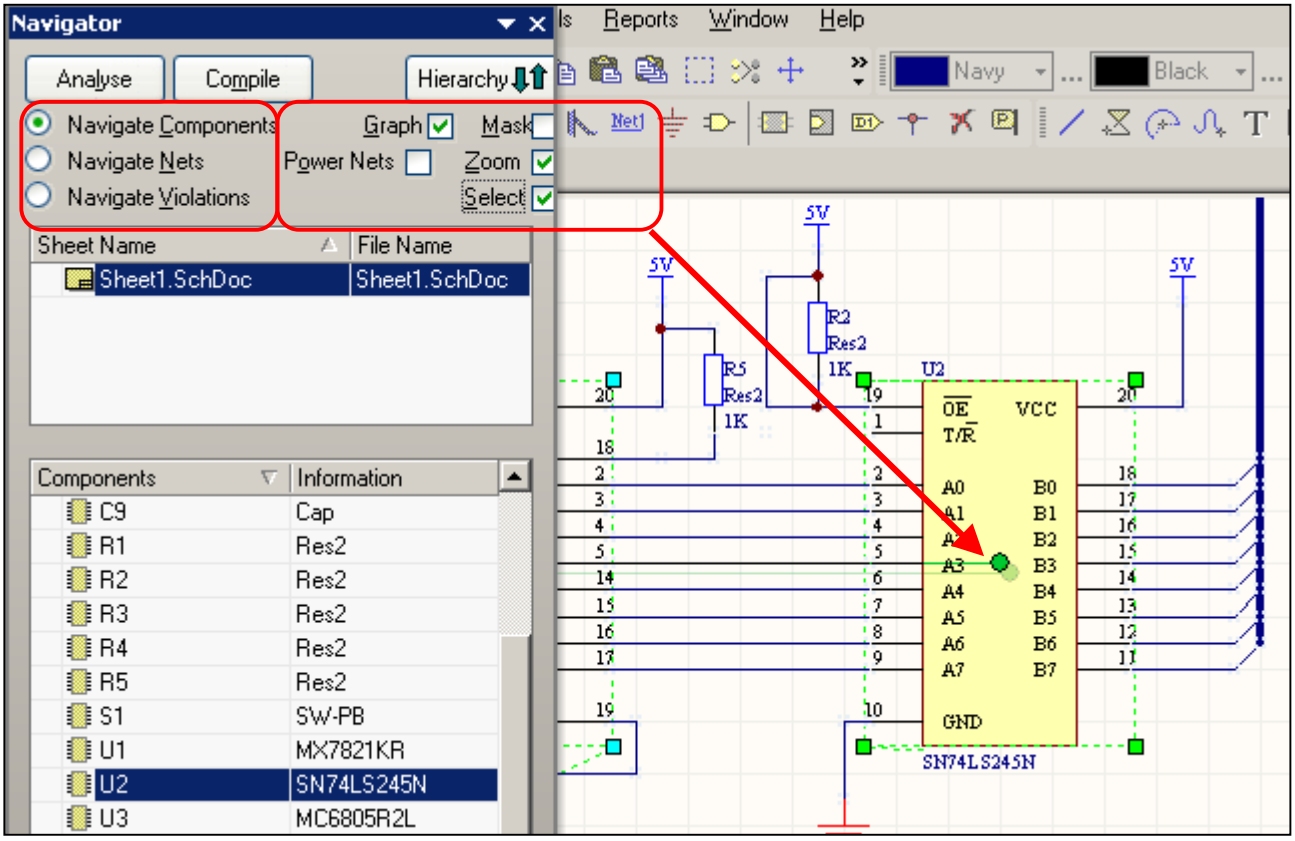
Prepare by : HK Sim simhkeng@gmail.com

View connection of a group of components – Navigator/Graph

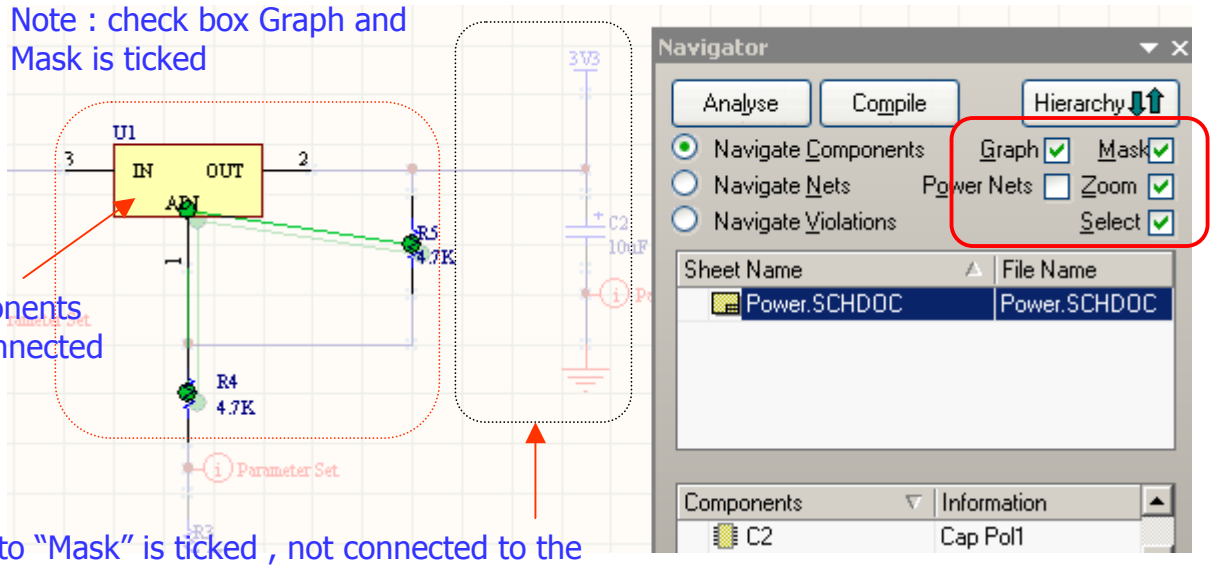
In the navigator, there is a few more function that you can use to see the connection information.

Graph function is very useful to see the inter connection components

You can navigator to see Net and Violations too



Note : check box Graph and Mask is ticked

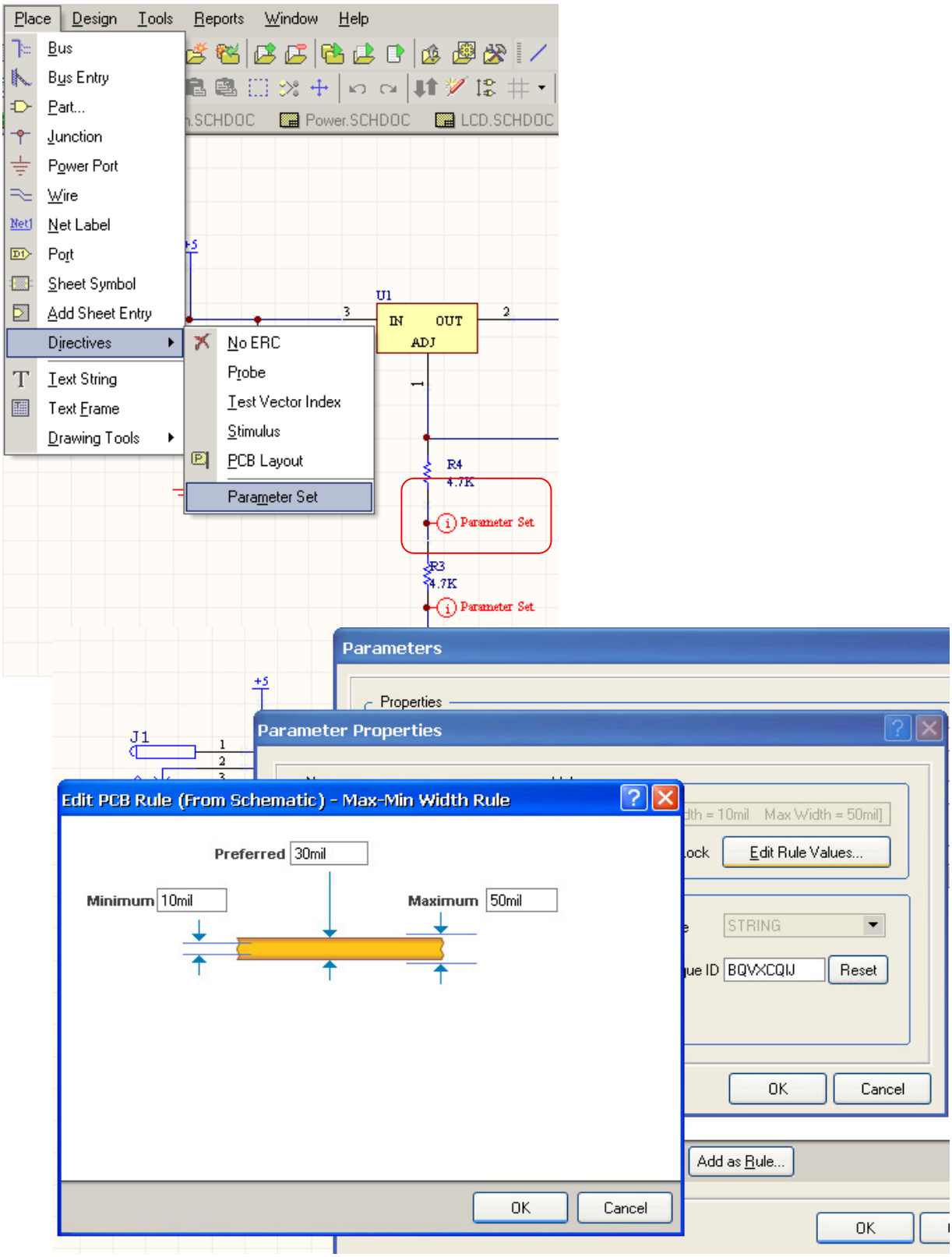


3 components inter-connected

Due to "Mask" is ticked , not connected to the component under investigation

Prepare by : HK Sim simhkeng@gmail.com

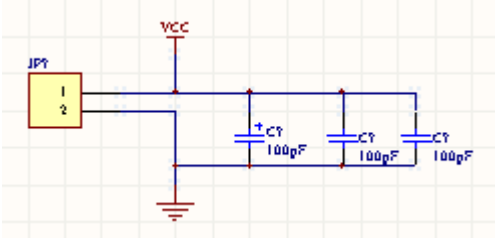
Specify a particular routing width on schematic – Parameter Set



Prepare by : HK Sim simhkeng@gmail.com

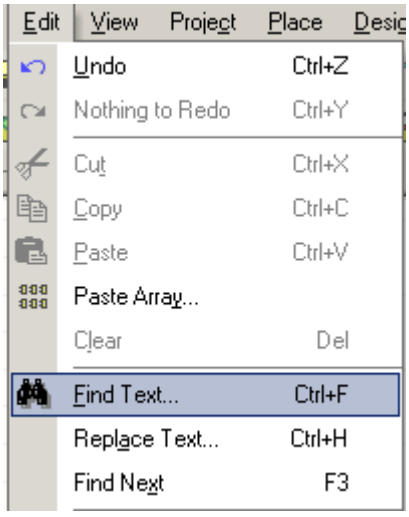
VCC and GND Connection

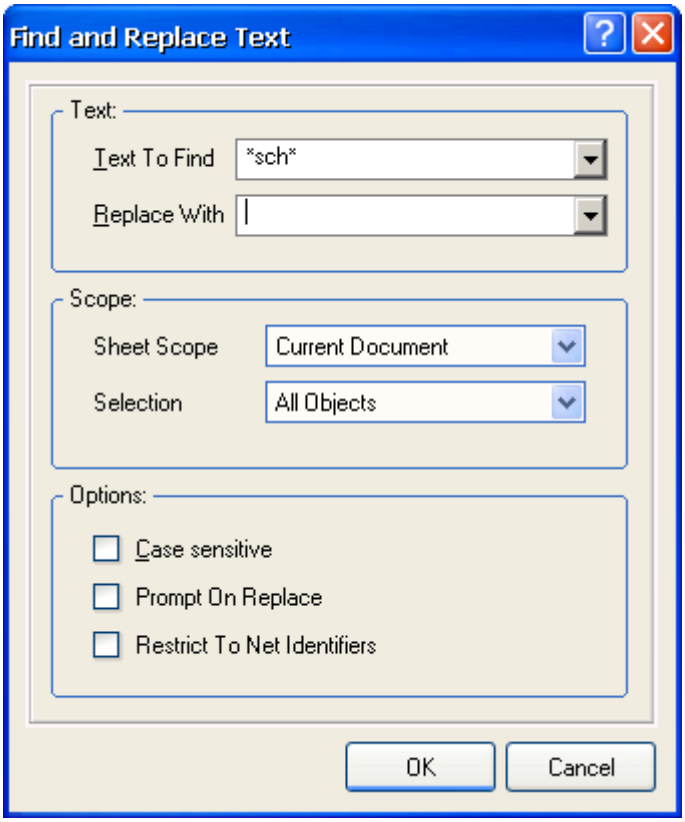
VCC and GND connection in a component is not seen (hidden) on schematic. They are normally label as VCC, GND. The connection of these two pins or more need to connected to Power source input by connector. Otherwise, these two pin will not be connected.



Find/Replace text

Good to locate a text string, e.g. designator, port name … etc on a schematic sheet

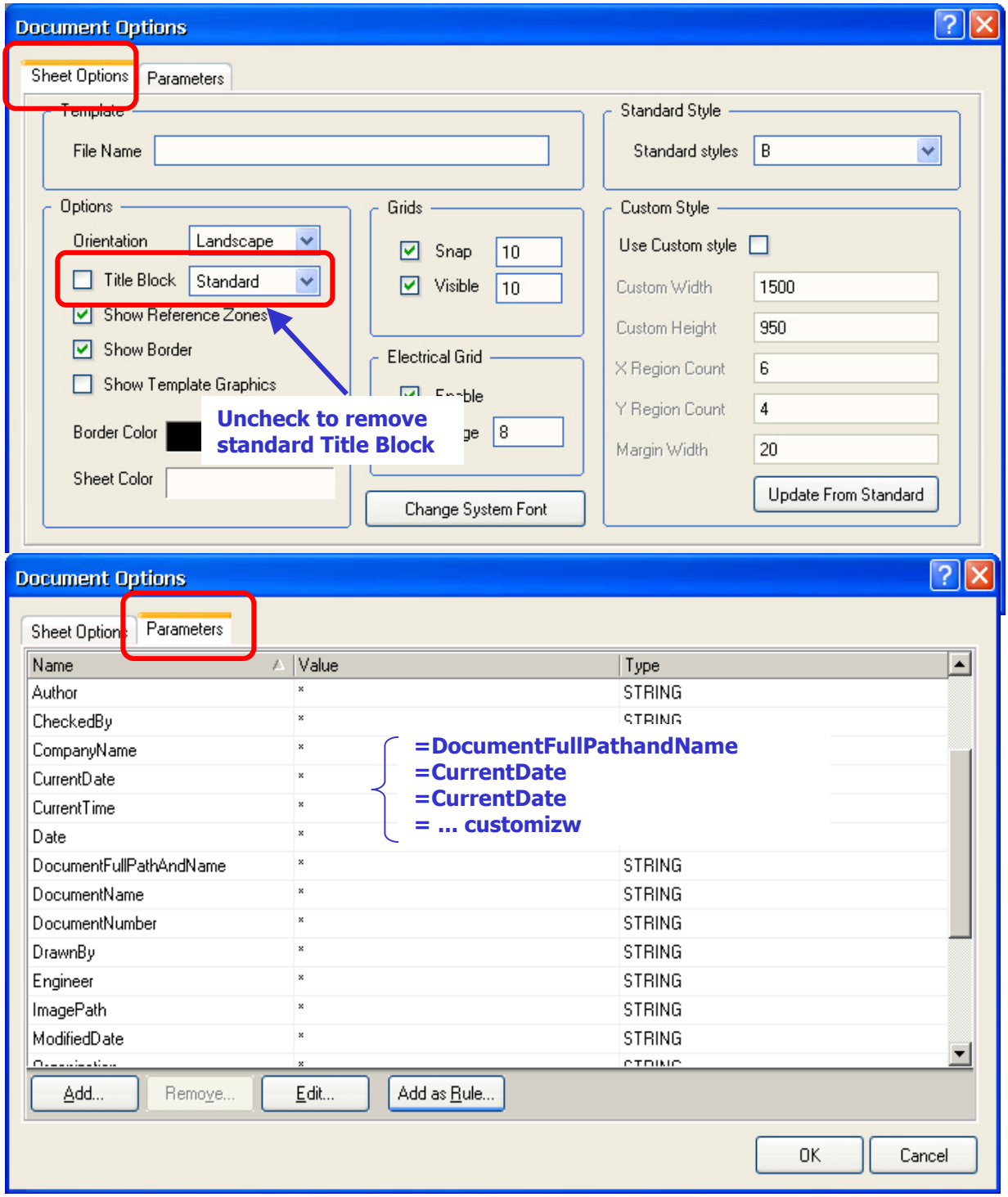




Prepare by : HK Sim simhkeng@gmail.com

Customize Document Title Block

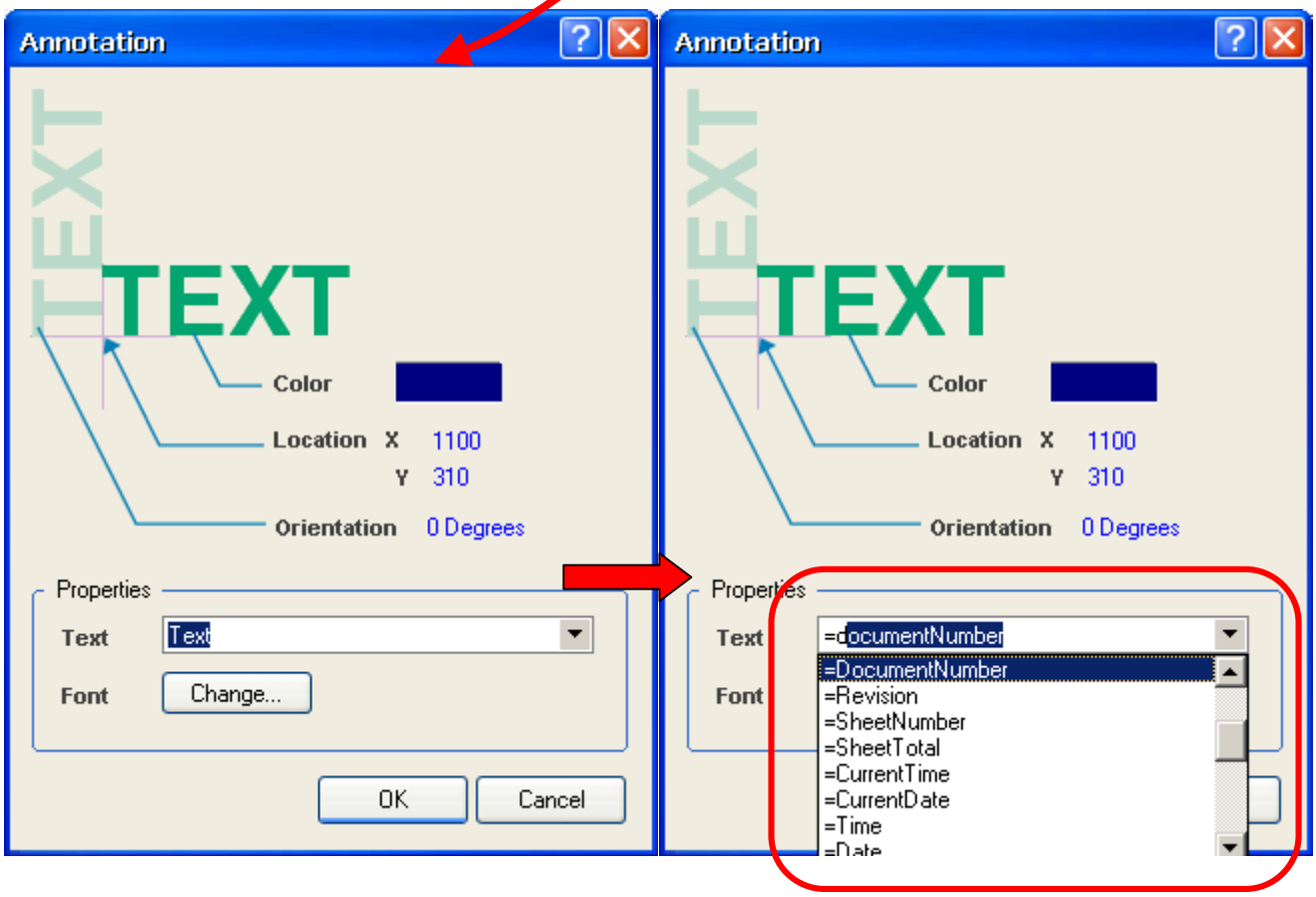
[Design]/[Option]



Prepare by : HK Sim simhkeng@gmail.com

[Tool] / [Place Text String] /[TAB button]

?????

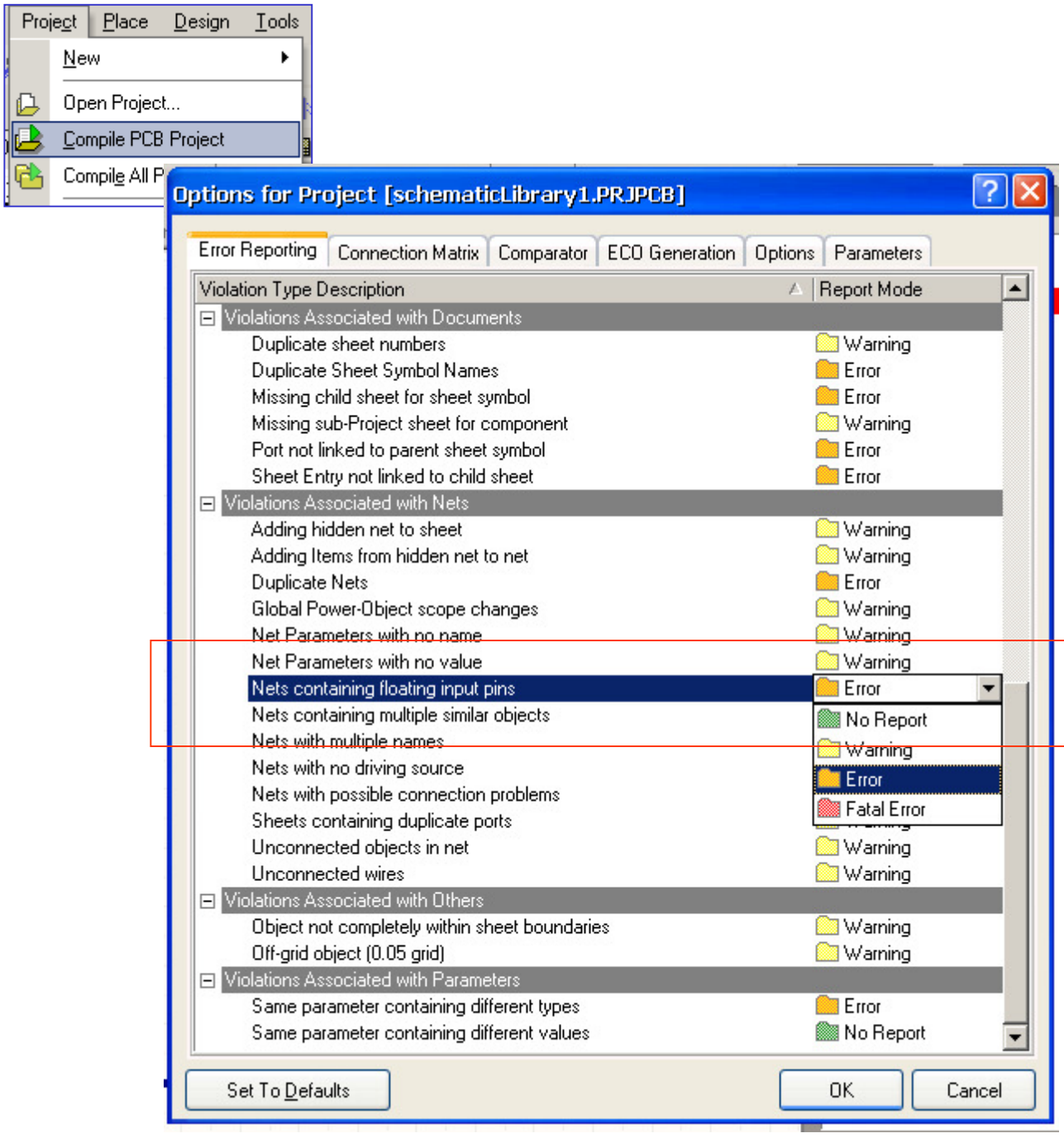


Pace/Port **Double click** **Start Changes**

Prepare by : HK Sim simhkeng@gmail.com

Compile a schematic to check design error

Use [Project / Compile PCB Project] to see if there is error.
Use [Project / Project Options] to see criteria. For error report, commonly change "Net containing floating Pins' set to warning instead of error
If after compilation and there is no error, the error report will be blank



Prepare by : HK Sim simhkeng@gmail.com

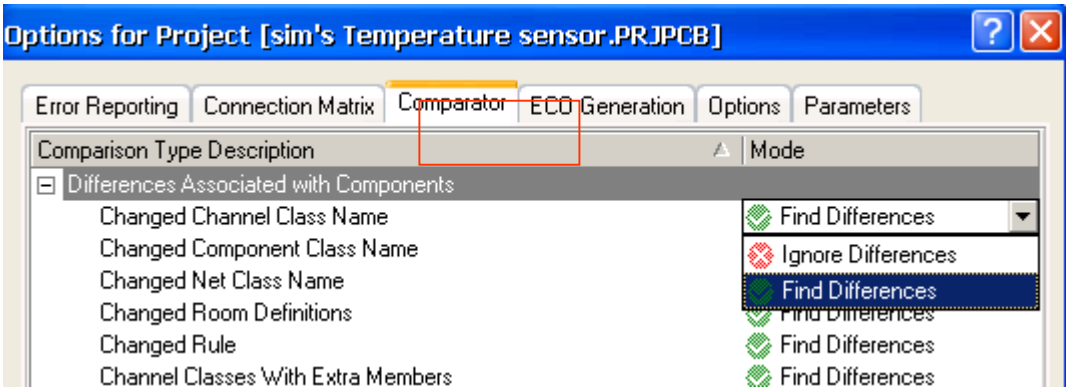
Set error message color

[Project]/[Project Option] – Connection Matrix

Prepare by : HK Sim simhkeng@gmail.com

Set comparison criteria between a new and old schematic ?

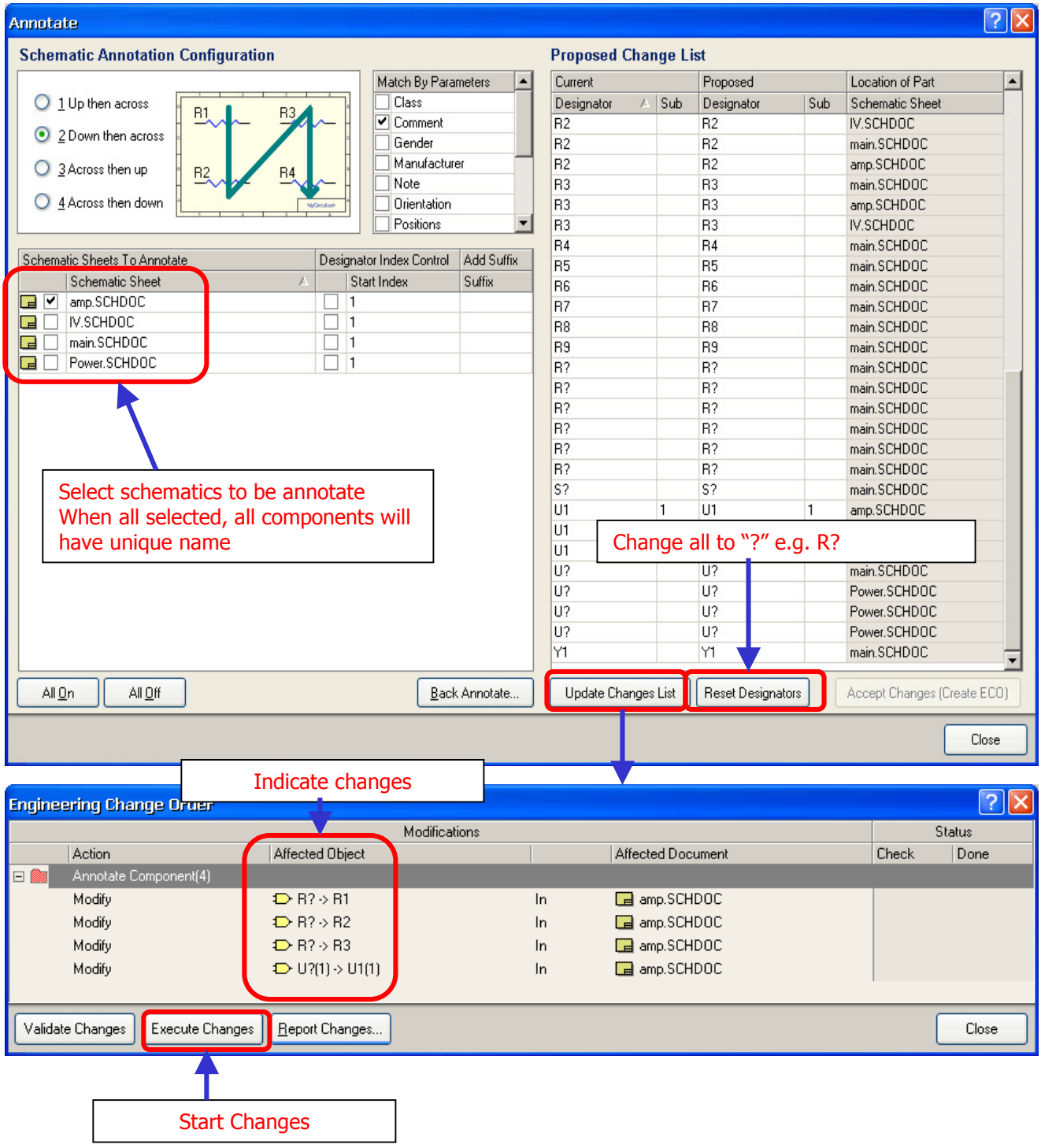
[Project]/[Project Option] – Comparator



Prepare by : HK Sim simhkeng@gmail.com

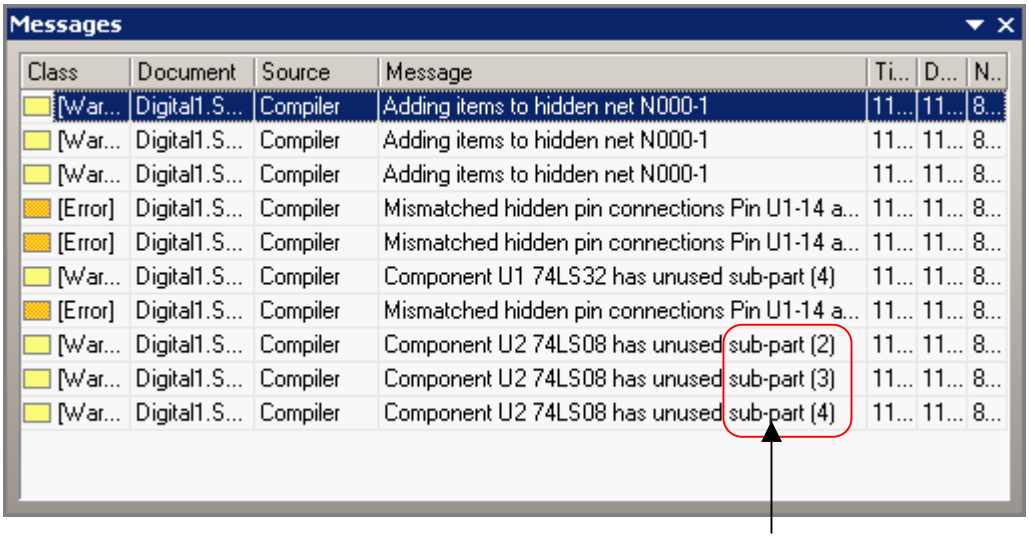
Avoid duplicate designator

System assign designator - [Tool]/[Annotate]



Prepare by : HK Sim simhkeng@gmail.com

Error example



Indicate un-use parts

Prepare by : HK Sim simhkeng@gmail.com

Reports

Bill Of Material (BOM)

Bill of Materials For Project [sim's Temperature sensor.PRJPCB]

Drag a column header here to group by that column

Designator	LibRef	Description	Footprint
C1	Cap Pol1	Polarized Capacitor (Radial)	MCCT-B
C2	Cap Pol1	Polarized Capacitor (Radial)	MCCT-B
C3	cap	Capacitor	R2012-0805
J1	PWR2.5	Low Voltage Power Supply Connect	PWR2.5
JP1	Header 3X2A	Header, 3-Pin, Dual row	HDR2X3_CEN
LCD1	DMC-50448N_modified	8 char x 2 line LCD character display	LCD-50448N
R1	Res1	Resistor	R2012-0805
R2	Res1	Resistor	R2012-0805
R3	Res1	Resistor	R2012-0805
R4	Res1		
R5	Res1		
R6	Res1		
R7	Res1		
R8	Res1		
R9	Res1		
R10	Res1		
R11	Res1		
U1	LM		
U2	TC		
U3	PIC		

S1	SW DIP-8	DIP Switch	DIP_SW_8WAY_SMD	
U1	TL082P	JFET-Input Operational Ampli	DIP-8/D11	TL082P
U2	L7805CP	Positive Voltage Regulator	SFM-T3/132V	L7805CP
U3	L7812CV	Positive Voltage Regulator	SFM-T3/132V	L7812CV
U4	L7912CV	Negative Voltage Regulator	SFM-T3/132V	L7912CV
U5	TL082P	JFET-Input Operational Amp	DIP-8/D11	TL082P-DIP Package
U6	AT89C2051			AT89C2052
U7	MAX232ACPE	+5V Powered, Multi-Channel		
Y1	XTAL	Crystal Oscillator	BCY-W2/D3.1	6MHZ

Export... Excel... Selected Only Close

Missing FootPrint

Cross Reference

Component Cross Reference Report For Project [sim's Temperature sensor.PRJPCB]

Document

Designator	LibRef	Description	Footprint	Comment
Document : LCD.SCHDOC				
LCD1	DMC-50448N_modified	8 char x 2 line LCD character displa	LCD-50448N	DMC-504 48N
R1	Res1	Resistor	R2012-0805	
R2	Res1	Resistor	R2012-0805	
Document : Power.SCHDOC				
C1	Cap Pol1	Polarized Capacitor (Radial)	MCCT-B	
C2	Cap Pol1	Polarized Capacitor (Radial)	MCCT-B	
J1	PWR2.5	Low Voltage Power Supply Connec	PWR2.5	PWR2.5
R3	Res1	Resistor	R2012-0805	
R4	Res1	Resistor	R2012-0805	
R5	Res1	Resistor	R2012-0805	
U1	LM317T	3-Terminal Adjustable Regulator	DSO-G3/C6.6	
Document : Sensor.SCHDOC				
U2	TCN75_SIM		SOIC8	Serial Temperature
Document : main.SCHDOC				
JP1	Header 3X2A	Header, 3-Pin, Dual row	HDR2X3_CEN	
Document : mcu.SCHDOC				
C3	cap	Capacitor	R2012-0805	
R6	Res1	Resistor	R2012-0805	
R7	Res1	Resistor	R2012-0805	
R8	Res1	Resistor	R2012-0805	
R9	Res1	Resistor	R2012-0805	
R10	Res1	Resistor	R2012-0805	
R11	Res1	Resistor	R2012-0805	
U3	PIC16C73-04/SO	EPROM-Based 8-Bit CMOS Microc	SO-G28/G7.45	PIC18C72-04/SO

Prepare by : HK Sim simhkeng@gmail.com

Schematic Short cut commands

Alignment

CTRL + B - Alignment all selected parts to bottom
CTRL + T - Alignment all selected parts to Top

CTRL + Shift+L - Alignment all selected parts to Left
CTRL + Shift+R - Alignment all selected parts to Right
CTRL + Shift+H - Evenly align all selected parts horizontally
CTRL + Shift+V - Evenly align all selected parts vertically

Duplicate Parts

CTRL + R

Find text string

CTRL + F

F11 – inspector
F12 – list (not much use)
CTRL + Shift +G – toggle grid display ON/OFF

Position Display

Home + MouseCursor – centralize display to cursor position

Display Size

PgUp – enlarge display size
PgDn – reduce display size
Shift + MouseRoller – enlarge/reduce display in smaller step

Move Parts

Shift + MouseLeft +Drag – move parts only, not connection
CTRL+ MouseLeft +Drag – move parts with connection

Toggle between Schematic and PCB

CTRL + TAB
(note : PCB and Schematic must be loaded inside the memory before execute this command)

Prepare by : HK Sim simhkeng@gmail.com