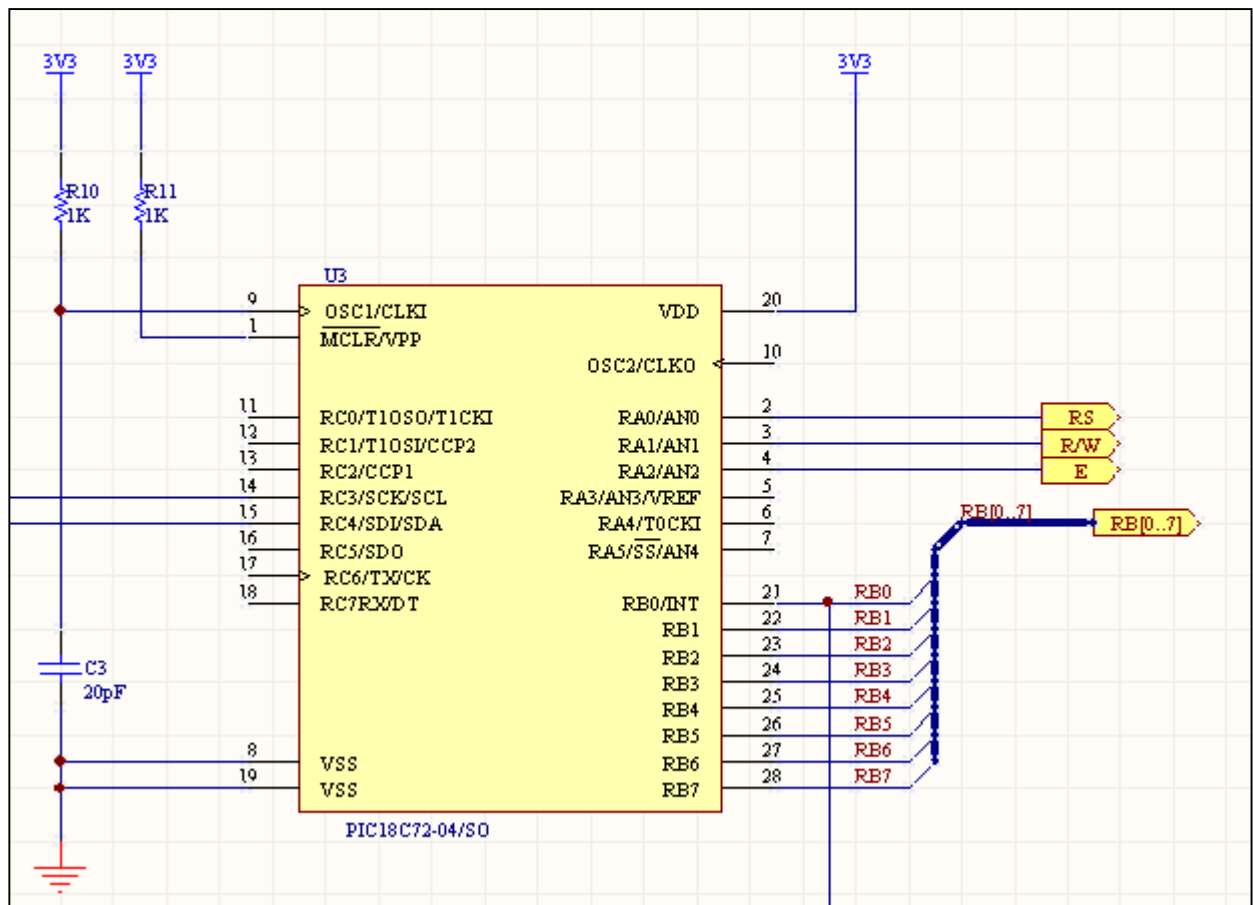


Prepare by : HK Sim simhkeng@gmail.com

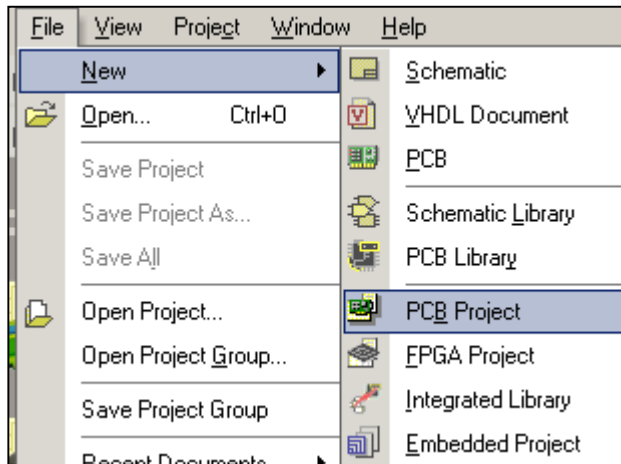
Date : 22 Dec 05



Prepare by : HK Sim simhkeng@gmail.com

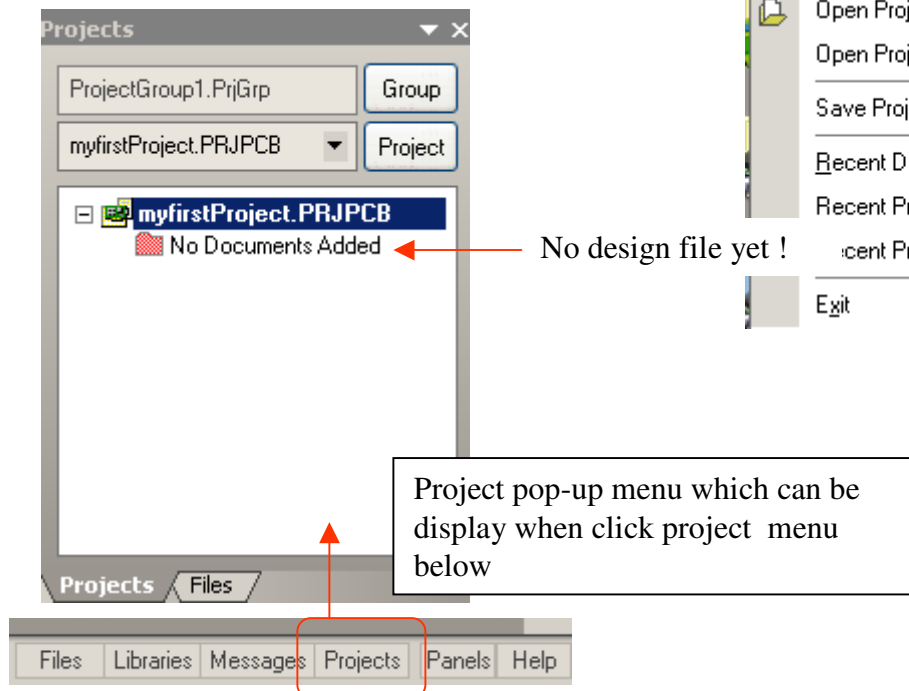
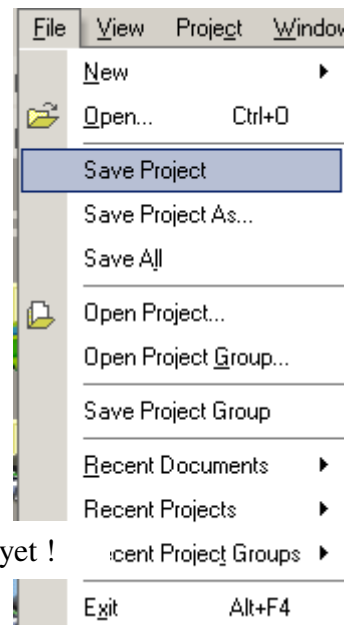
Start a new PCB project

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

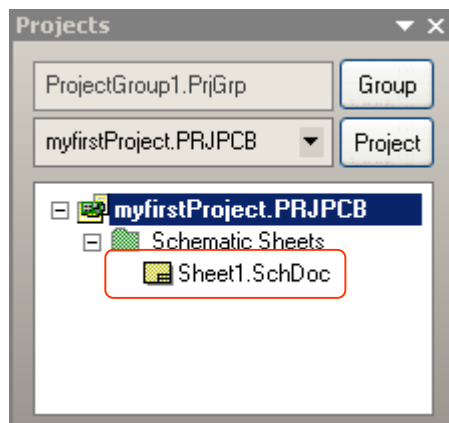
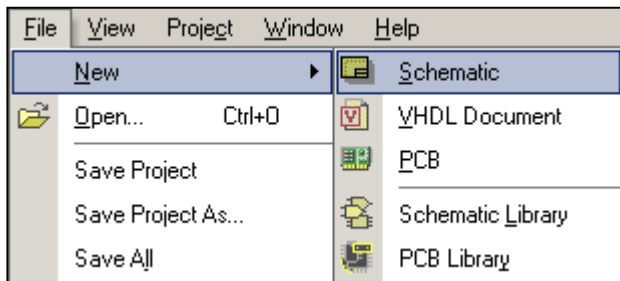


[File] / [Save Project]

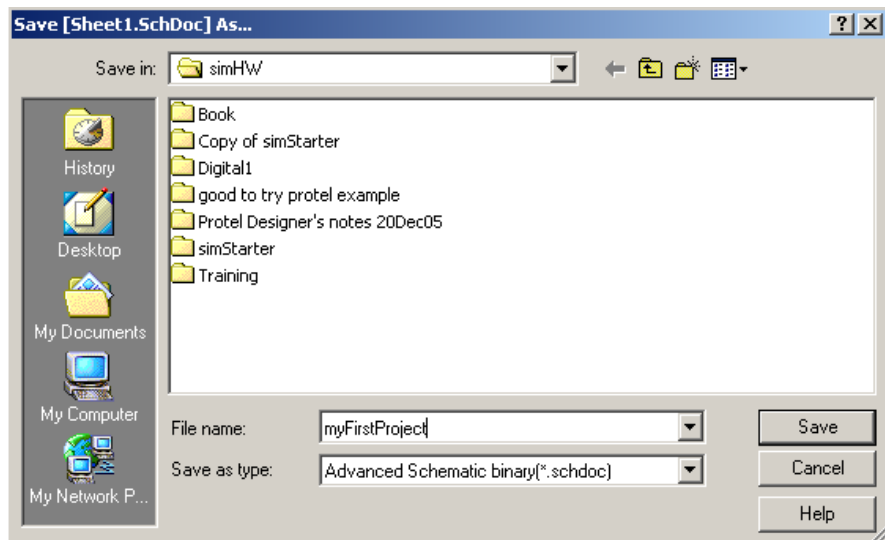
A pop-up menu will appear to enter the project name and the location of the project folder. Enter the project name accordingly and this will appear on the project pop-up menu



Add new schematic page into the project



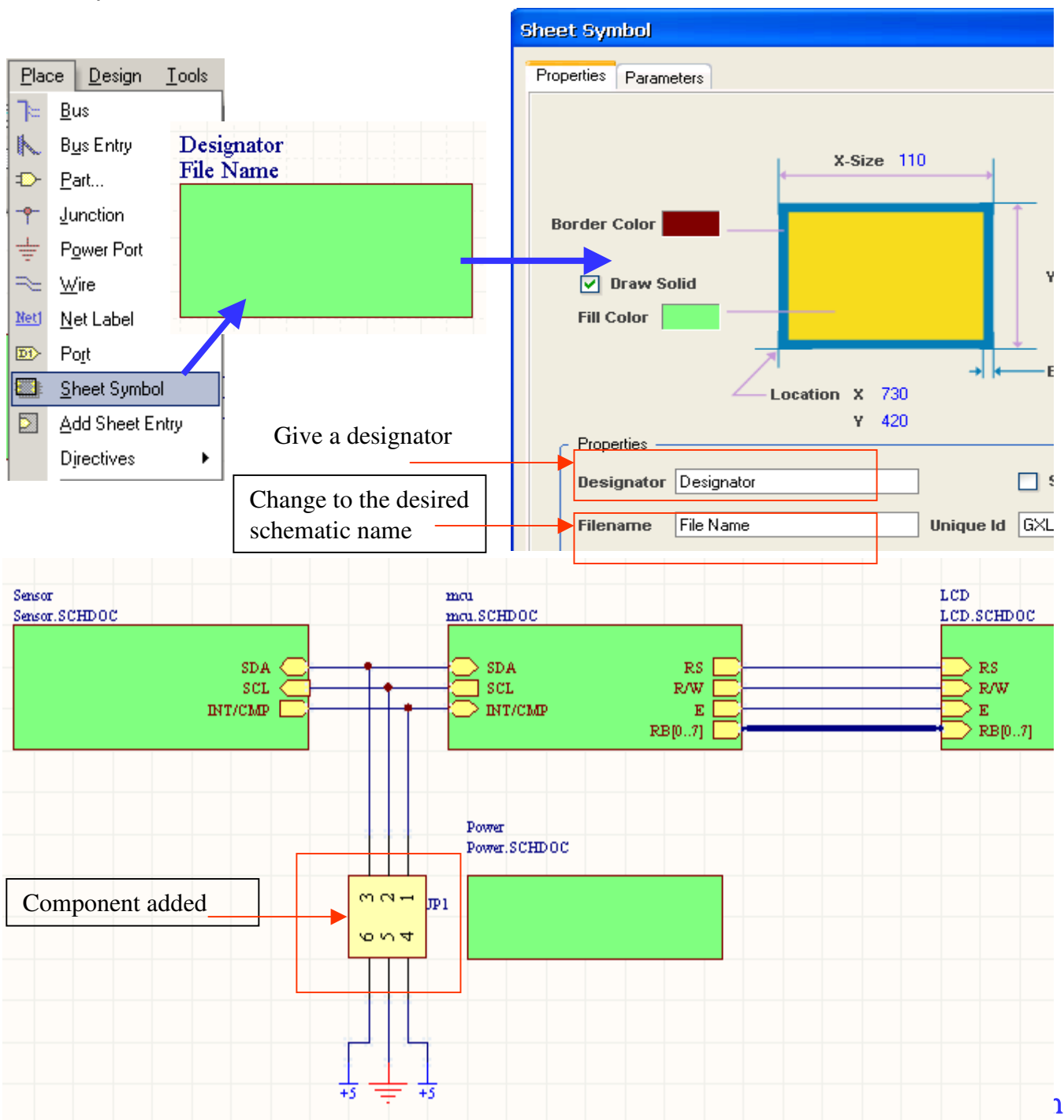
Save the schematic file



Schematic Design concept – Multi-sheet

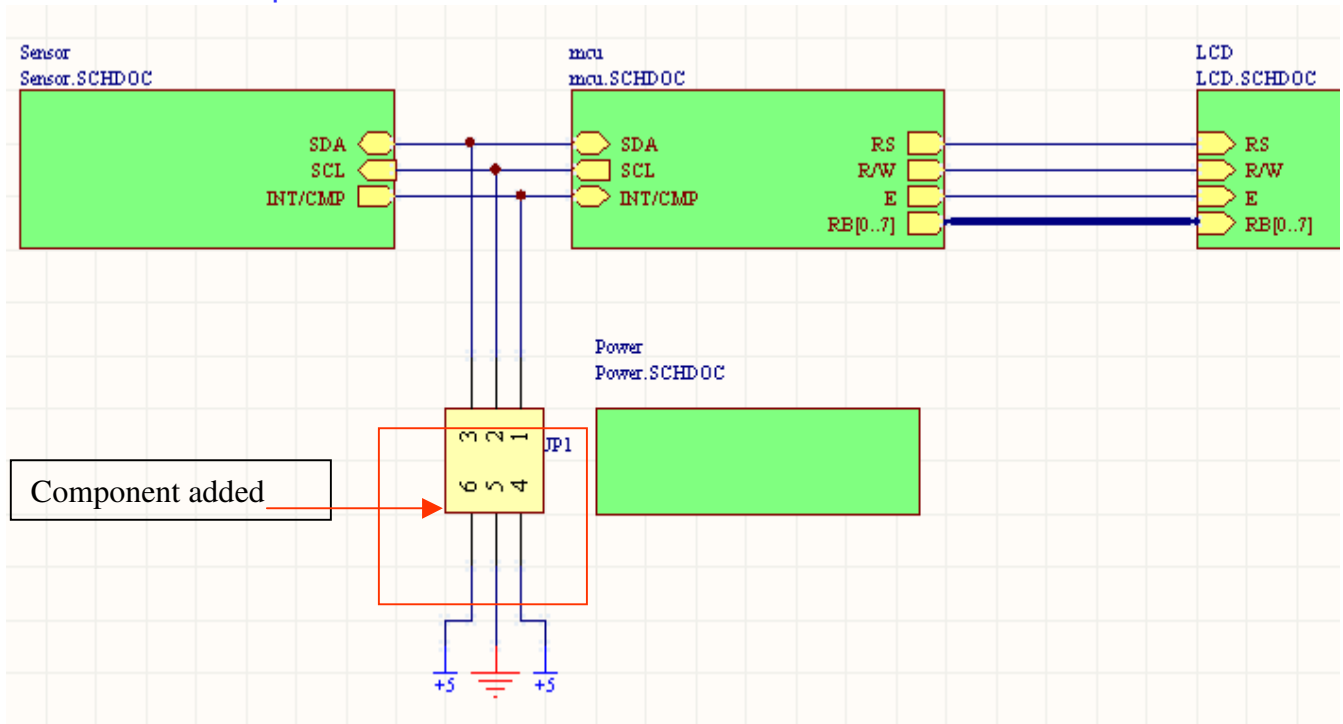
Under one project,

- Create each individual block of circuit on individual schematic sheet.
- For each schematic sheet, use **NetLabels** or **Ports** for input and output
- 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

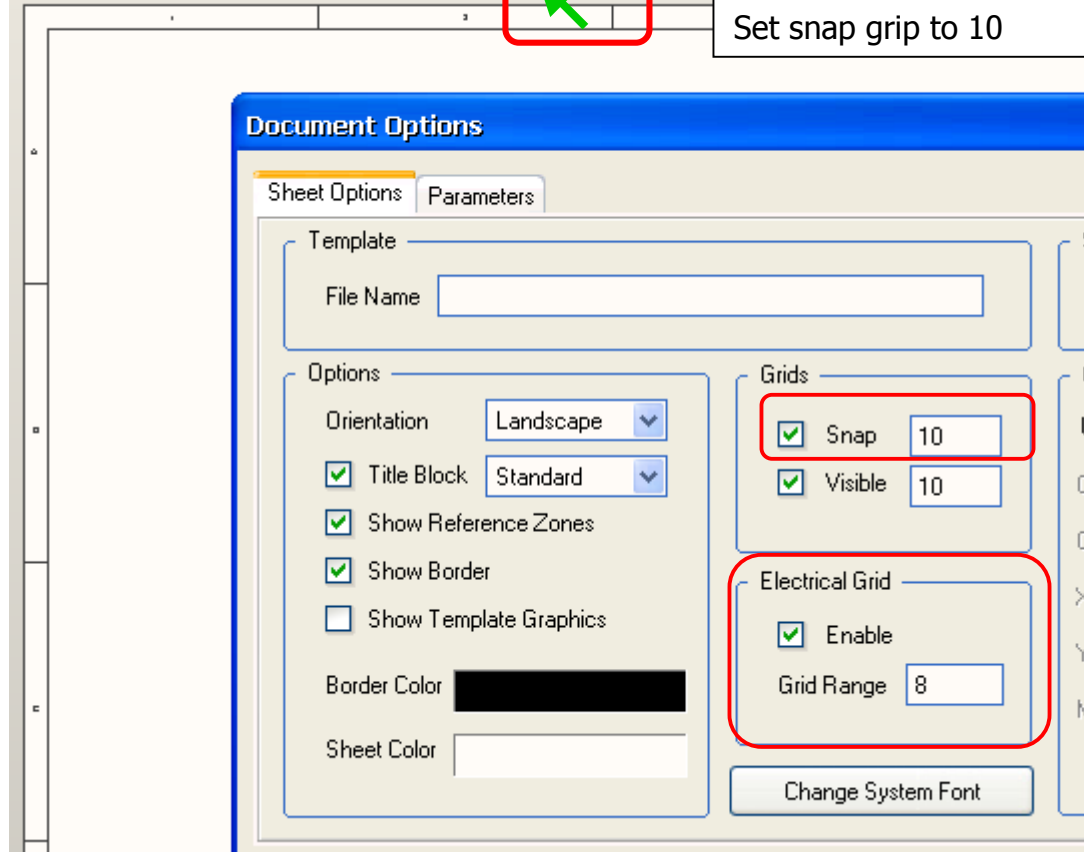
Sheet symbols are used to show which sub sheets link to the top sheet. Each symbol represents a sub-sheet and has a filename attribute that provides the link between the top sheet and the sub sub-sheet with that filename.



Set Grid for design

temperature sensor.PCBDOC Power.SCHDOC

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



Add component to schematic

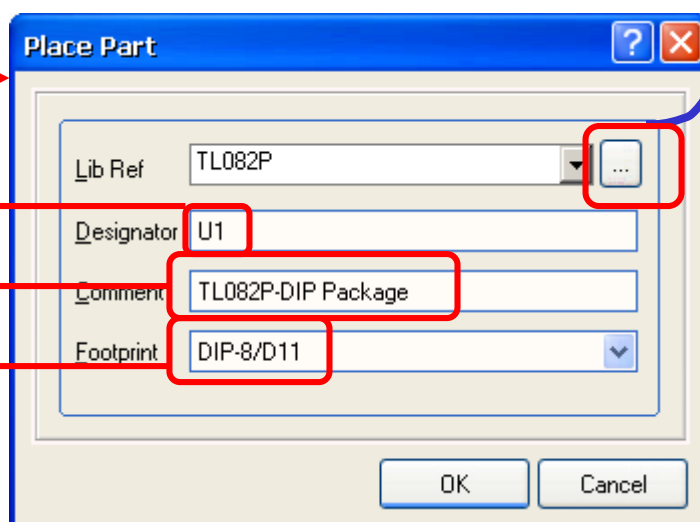
To search more part ,
more libraries

[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

Select the possible installed library

Add New/Remove Existing libraries

For Libraries loaded

For Libraries that is not loaded in the design environment

e.g. Res* - wild card search for resistor

If component found in a library that is not loaded in the design environment, the Install Library button will active

Select the wanted component

Search Libraries

Search **Results**

Scope

☐ Available Libraries

☒ Libraries on Path

Path

Path: C:\Program Files\Altium\Library\

☒ Include Subdirectories

File Mask

File Mask: *

Search Criteria

☒ Name *8051*

☐ Description

☐ Model Type

☐ Model Name

Search **Results**

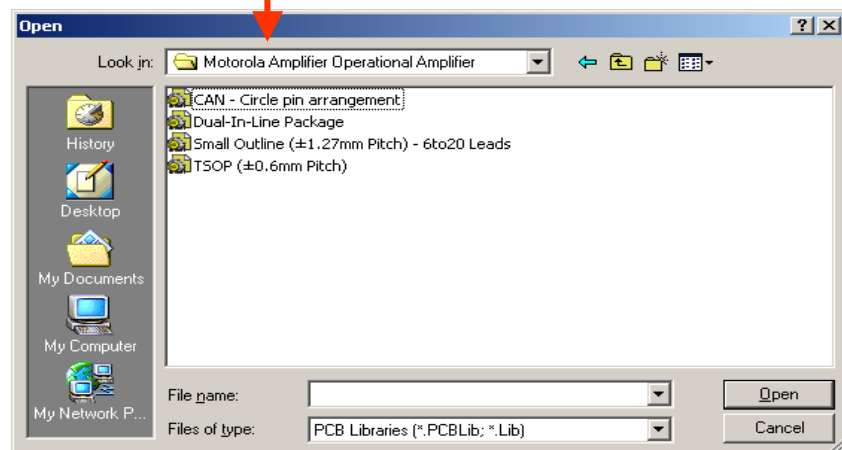
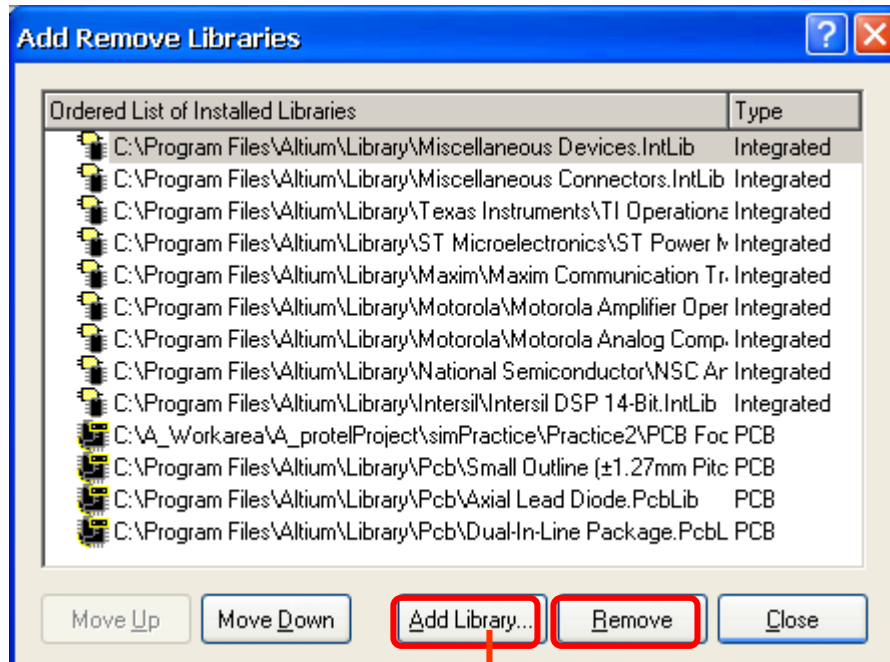
Component Name	Library	Description
AD8051AR	AD Operational Amplifier.IntLib	Low Cost, High-Speed Rail-to-Rail Amplifier
AD8051ART	AD Operational Amplifier.IntLib	Low Cost, High-Speed Rail-to-Rail Amplifier
MC74LVX8051D	ON Semi Logic Multiplexer.IntLib	Analog Multiplexer/Demultiplexer
MC74LVX8051DT	ON Semi Logic Multiplexer.IntLib	Analog Multiplexer/Demultiplexer

Model Name **Type**

SO-G16 PCBLIB

Install Library **Select** **Stop**

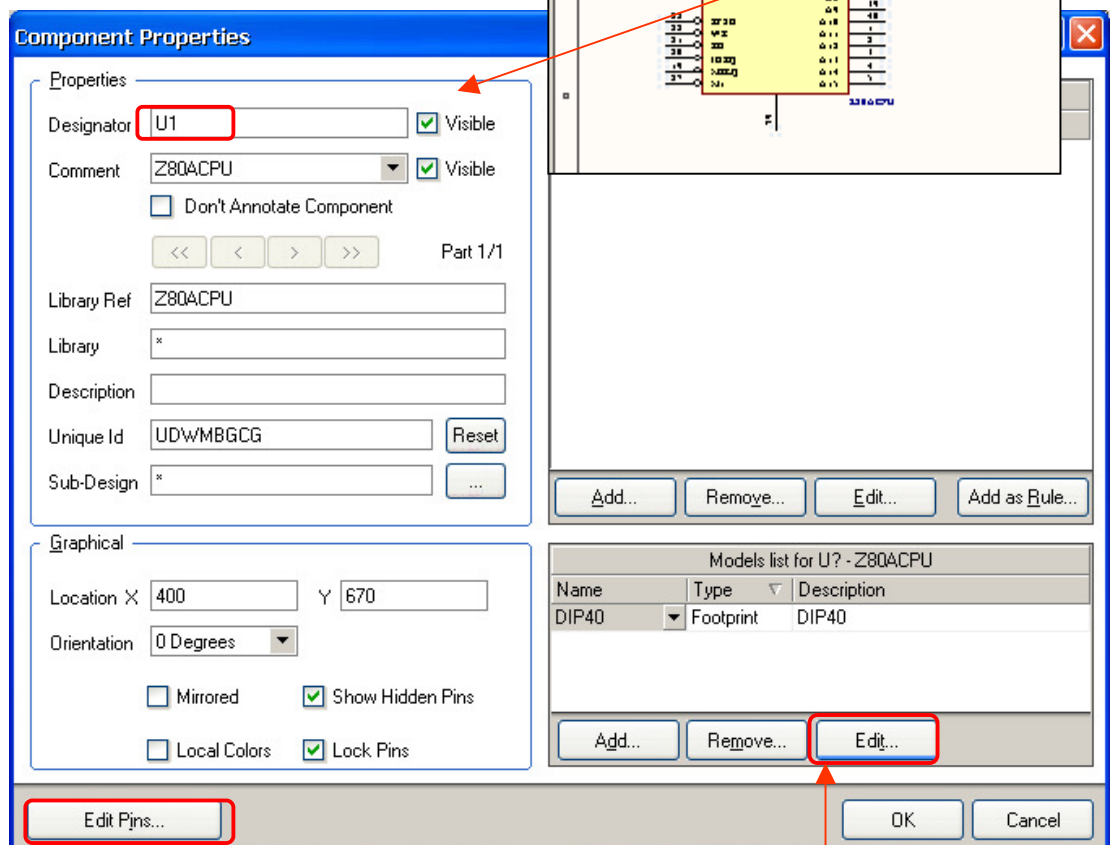
**Libraries that wanted to be use have to be loaded to the working environment.
DXP's library directory is c:\program Files\Altium\Library\<library folders>**



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



Edit schematic's component

Check to make sure foot print is correct

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.

Component Properties

Properties

Designator ☒ Visible

Comment ☒ Visible

☐ Don't Annotate Component

Part 1/1

Library Ref

Library

Description

Unique Id

Sub-Design

Parameters list

Visible	Name	Value
<input type="checkbox"/>	Class	Passive
<input type="checkbox"/>	Manufacturer	Generic Com
<input type="checkbox"/>	Published	08/06/2000
<input type="checkbox"/>	Publisher	Altium Hobart
<input type="checkbox"/>	SubClass	Resistor
<input checked="" type="checkbox"/>	Value	4.7K

Graphical

Location X Y

Orientation

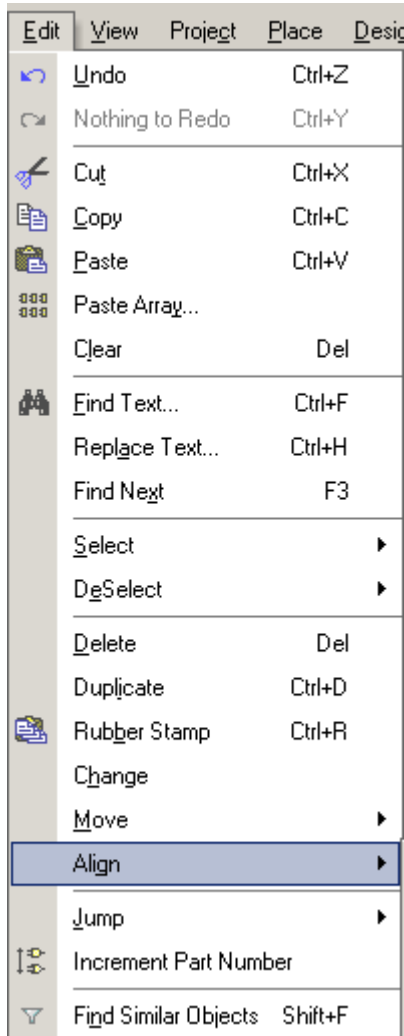
☐ Mirrored ☐ Show Hidden Pins

☐ Local Colors ☒ Lock Pins

Models list for

Name	Type	Descr
RESISTOR	Simulation	Resist
Res1	Signal Integrity	
<input type="text" value="R2012-0805"/>	Footprint	
AXIAL-0.3		
<input type="text" value="R2012-0805"/>		

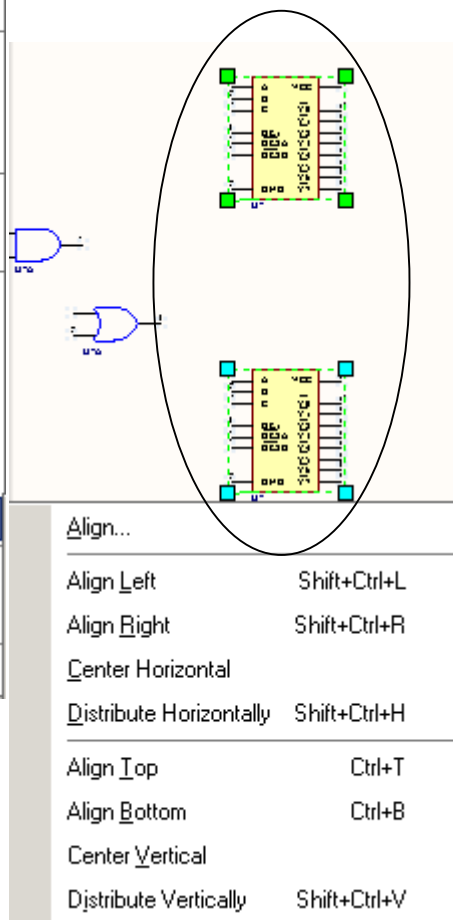
Alignment of Parts



Select items to be aligned

•[Edit] / [Align] / ...

Short Cuts: AL, AR, AB,AT



Draw Connections

[Place] / [Wire]



PW

Change direction of wire : Shift + Space Bar

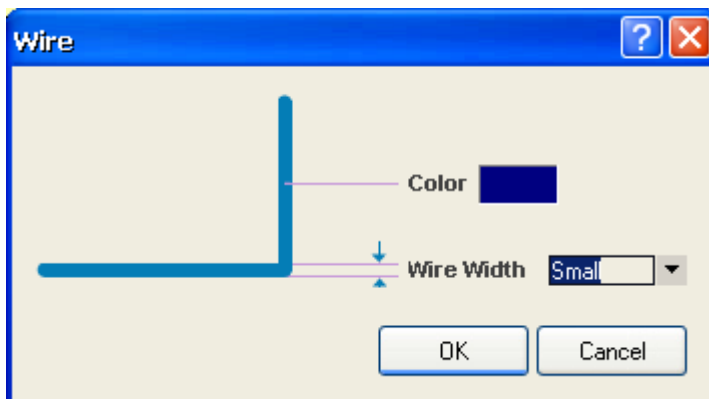
Change ... ? : Space Bar

Release the last portion of wire : Backspace

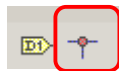
Change property of wire : double click wire

[Shift] + [Space Bar]

BackSpace



Auto Junction

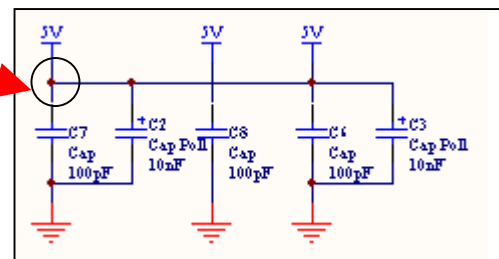
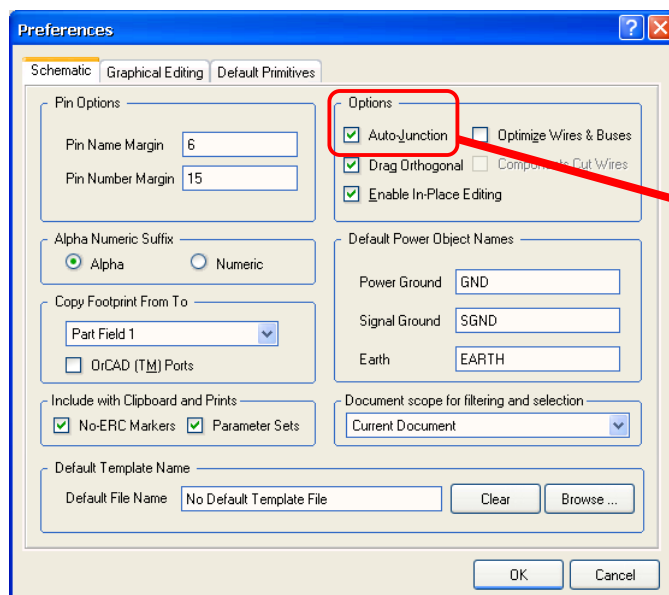


PJ

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"

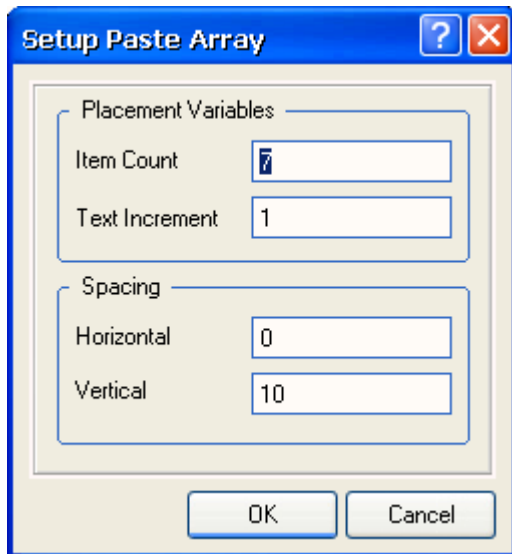


Place Duplicate Items

EY

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

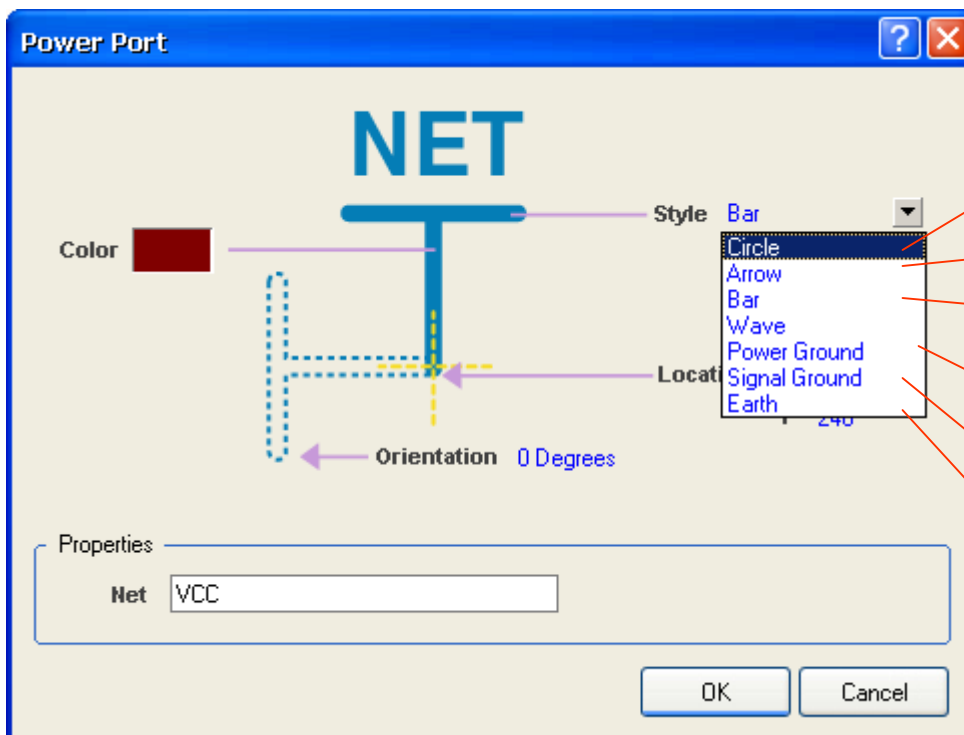
Use EDIT/PlaceArray , define the item count , OK and then move to the location and Click on it.



To select multiple components



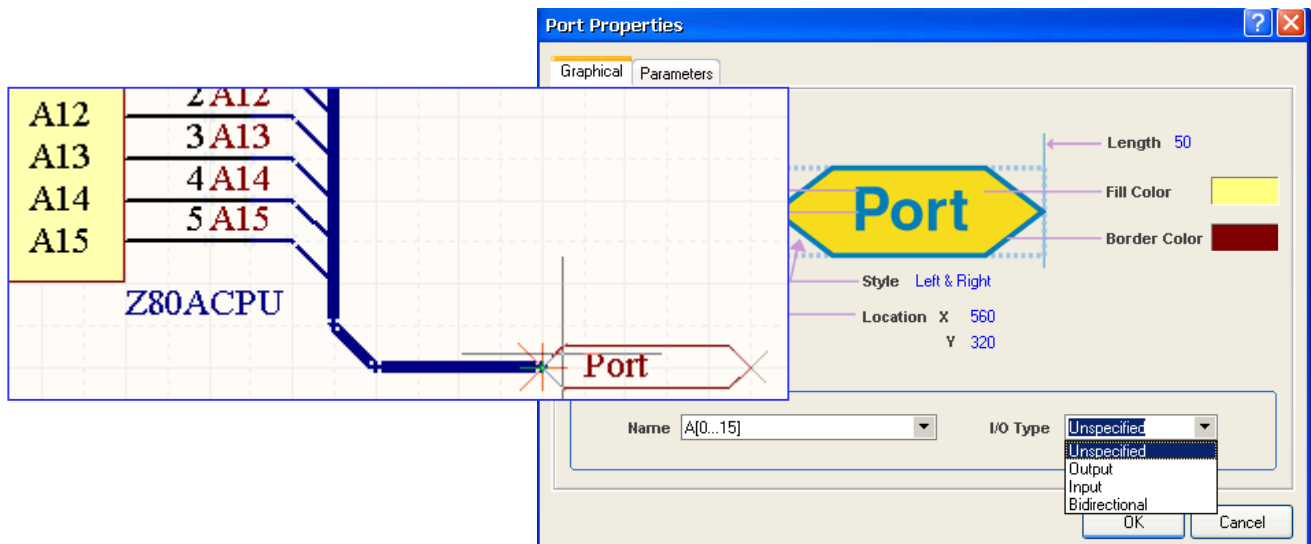
Create Power Port



Place Port

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

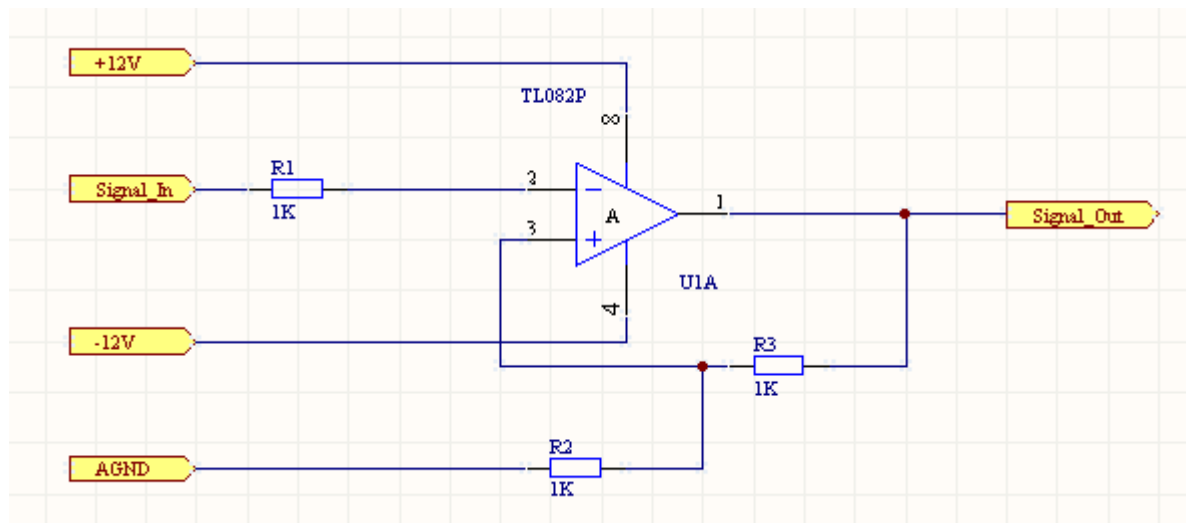
Double click to Popup the property . Enter Port Name and Select IO type.



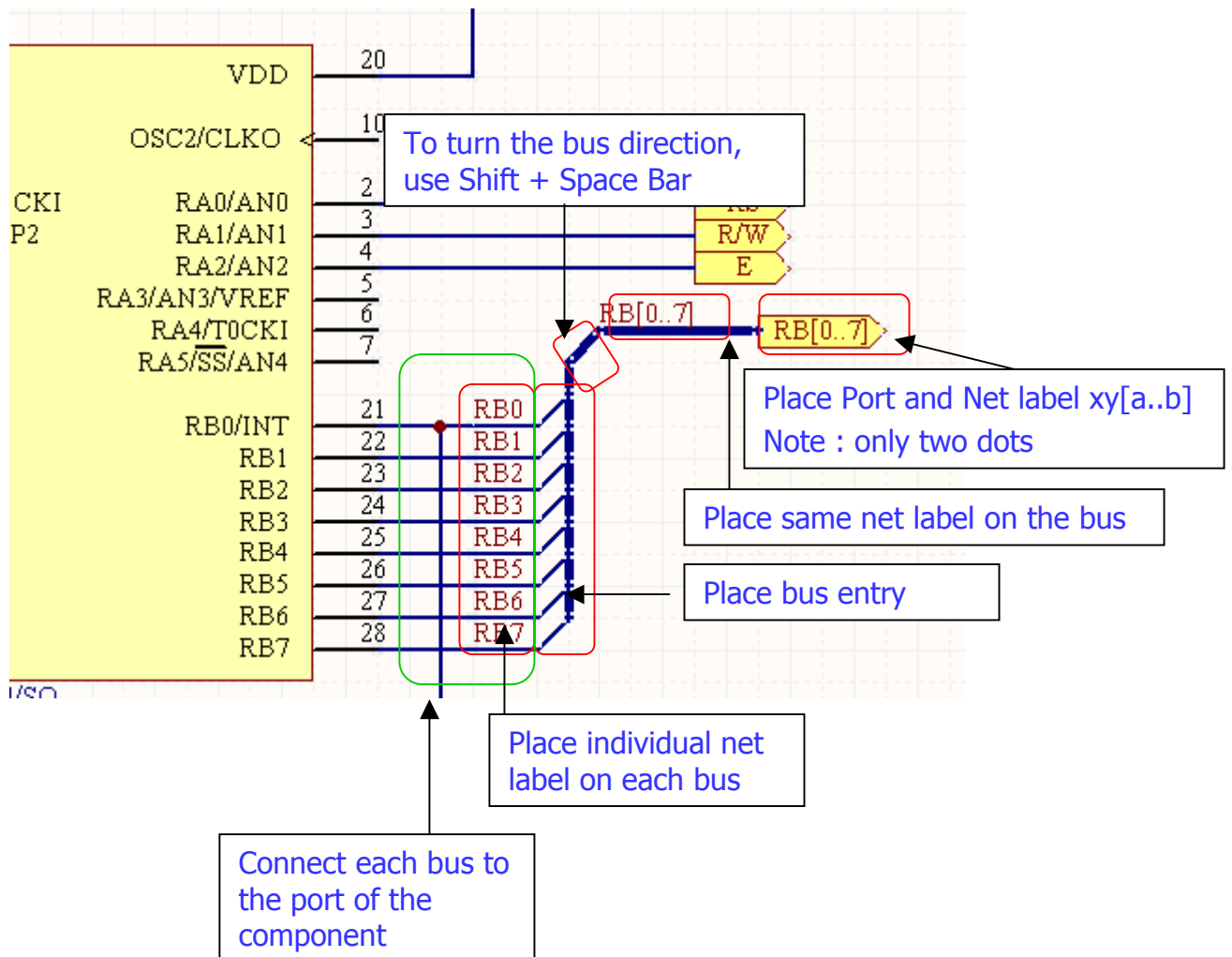
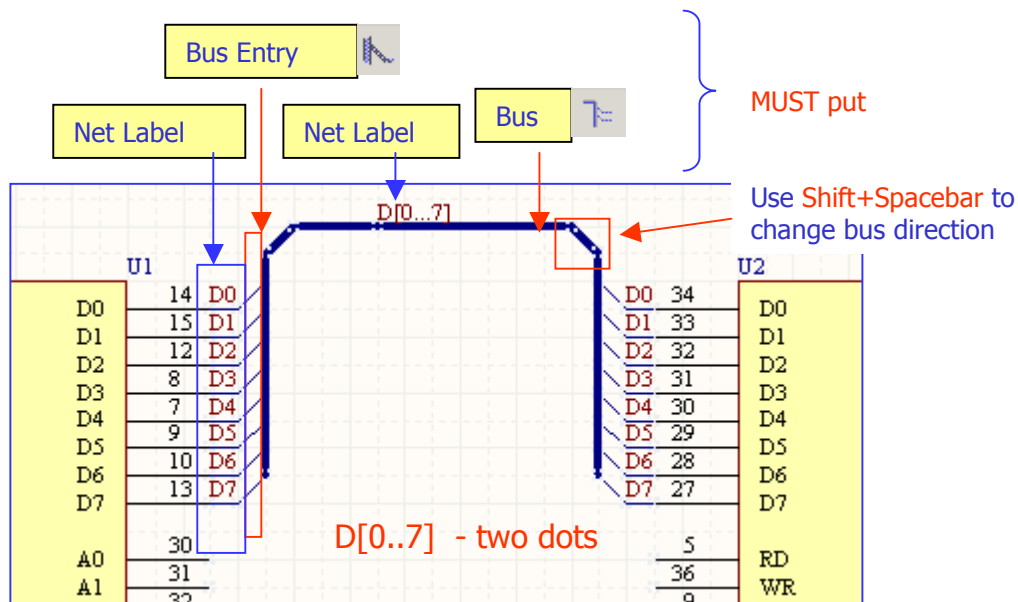
Data bus, Address bus declare as Bidirectional

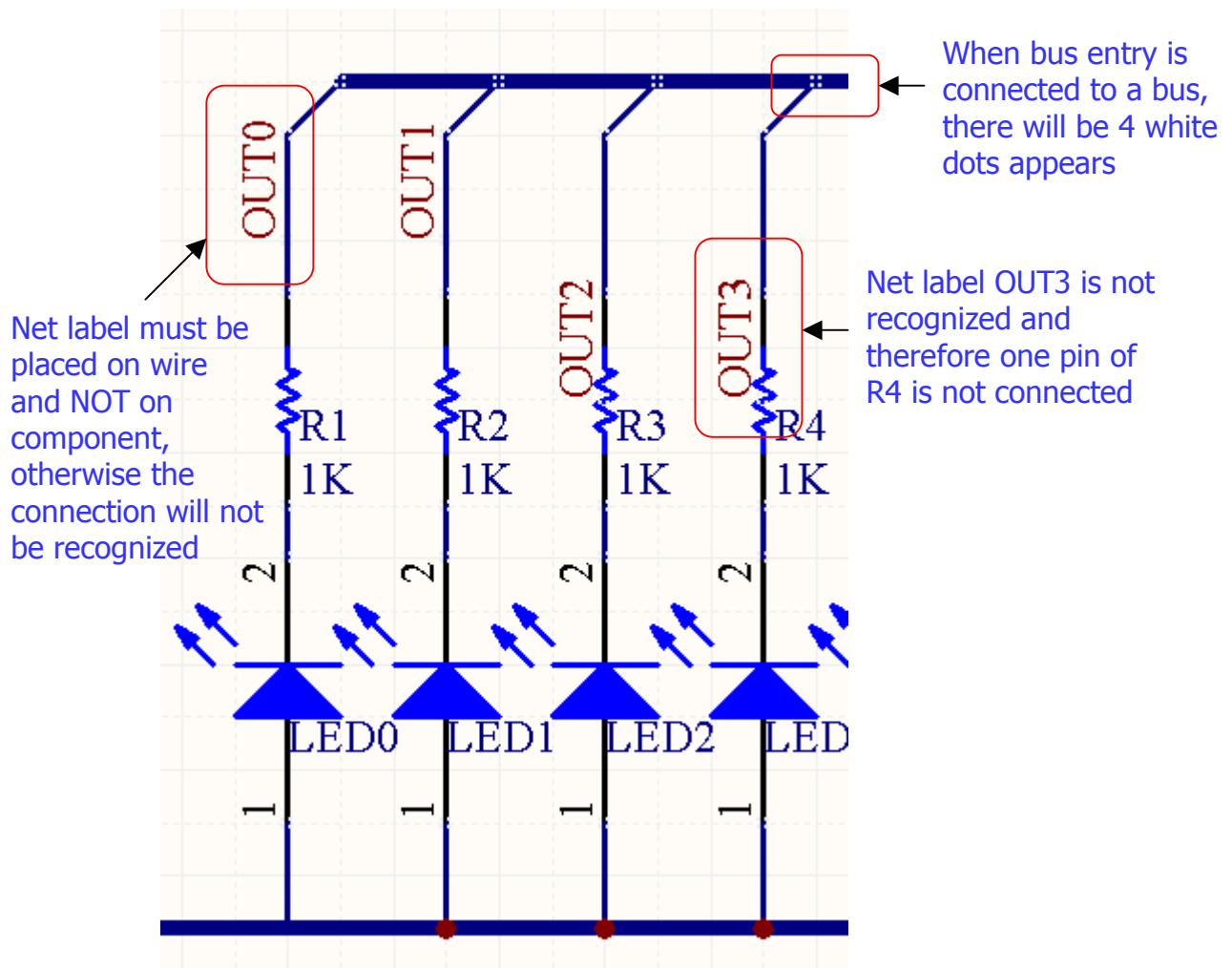
Port type = Output will be connected to Input port of the component

Port type = Input will be connected to output port of the component



Create Bus Connection





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

The image shows a schematic capture software interface. At the top, a menu bar includes 'Inspect', 'List', 'Navigate', 'Files', 'Libraries', 'Messages', 'Projects', 'Panels', and 'Help'. The 'Inspect' menu item is highlighted with a red box. To the right of the menu bar, there are buttons for 'Mask Level' and 'Clear'. A callout box with a blue border points to the 'Inspect' button, containing the text 'Toggle short on/off using' and a yellow button labeled 'F11'.

Below the menu bar, the 'Inspector' window is open. It has a title bar with a dropdown arrow and a close button. The window is divided into several sections: 'Kind' (Object Kind: Part), 'Design' (Owner Docum: Sheet1.SchDoc), 'Graphical' (X1: 980, Y1: 720, Orientation: 0 Degrees, Mirrored: ☐, Show Hidden: ☐) and 'Object Specific' (Description: 8-Bit Microcomputer, Lock Designa: ☐, Pins Locked: ☒, File Name: , Library Path: , Library Refere: MC6805R2L, Component D: U3, Current Part: , Part Comment: MC6805R2L, Current Footpr: DIP-40). The 'Object Specific' section is highlighted with a red box. A red arrow points from the 'Inspect' button to the 'Inspector' window. Another red arrow points from the 'Inspector' window to a schematic diagram of an MC6805R2L microcomputer. The schematic diagram shows the microcomputer with its pins connected to various components, including XTAL2, XTAL1, XTAL, TIMER, RST, PA0, PA1, PA2, PA3, PA4, PA5, PA6, PA7, PC0, PC1, PC2, PC3, PC4, PC5, PC6, PC7, PD0/AN0, PD1/AN1, PD2/AN2, PD3/AN3, PD4/V RL, PD5/V RL, PD6/INT2, PD7, GND, and VCC. A yellow box highlights the microcomputer component. A callout box with a yellow border points to the microcomputer, containing the text 'Click'.

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.

Set what to Navigate

When blank, need to click Compile and then Navigator

Selected component Will be stand out

Select sheet and component on sheet

Set Contrast level

Clear to reset (remove contrast)

The Navigator interface includes the following sections:

- Buttons:** Analyse, Compile, Hierarchy (with up/down arrows).
- Navigation Mode:** ☒ Navigate Components, ☐ Navigate Nets, ☐ Navigate Violations. Sub-options: ph ☐, Mask ☒, Zoom ☒, Select ☒.
- Table:** Sheet Name | File Name
- Table:** Components | Information
- Table:** Pins | Net | I/O
- Bottom Buttons:** Mask Level, Clear

Component	Information
C1	Cap Pol1
C2	Cap
C3	Cap
JP?	Header 2H
U1	Z80ACPU
U2	8255

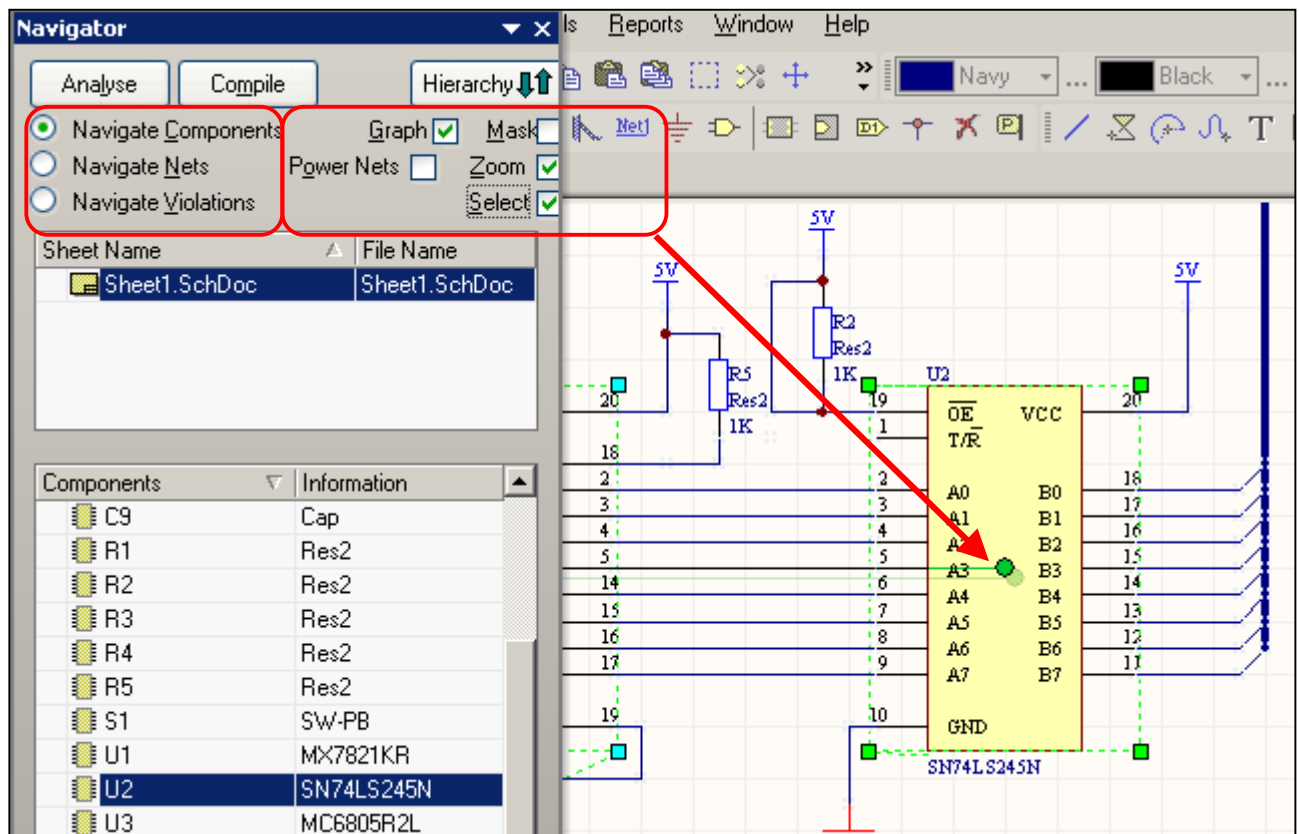
Pins	Net	I/O
C2-1	VCC	Passive
C2-2	GND	Passive

View connection of a group of components – Navigator/Graph

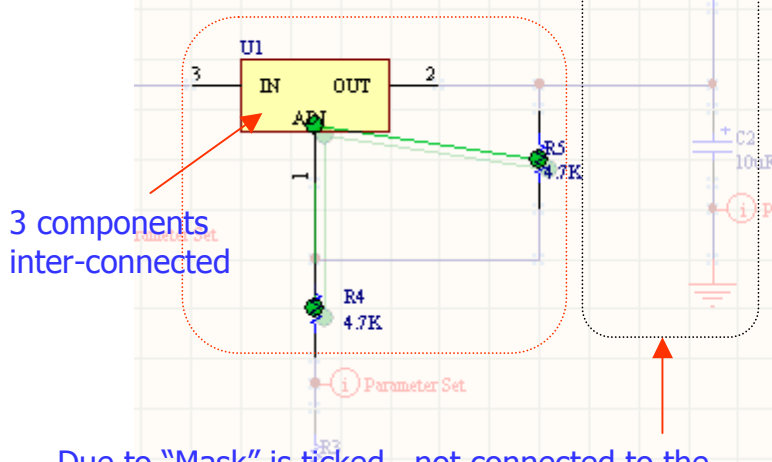
In the navigator, there is a few more function that you can use to see the connection information.

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

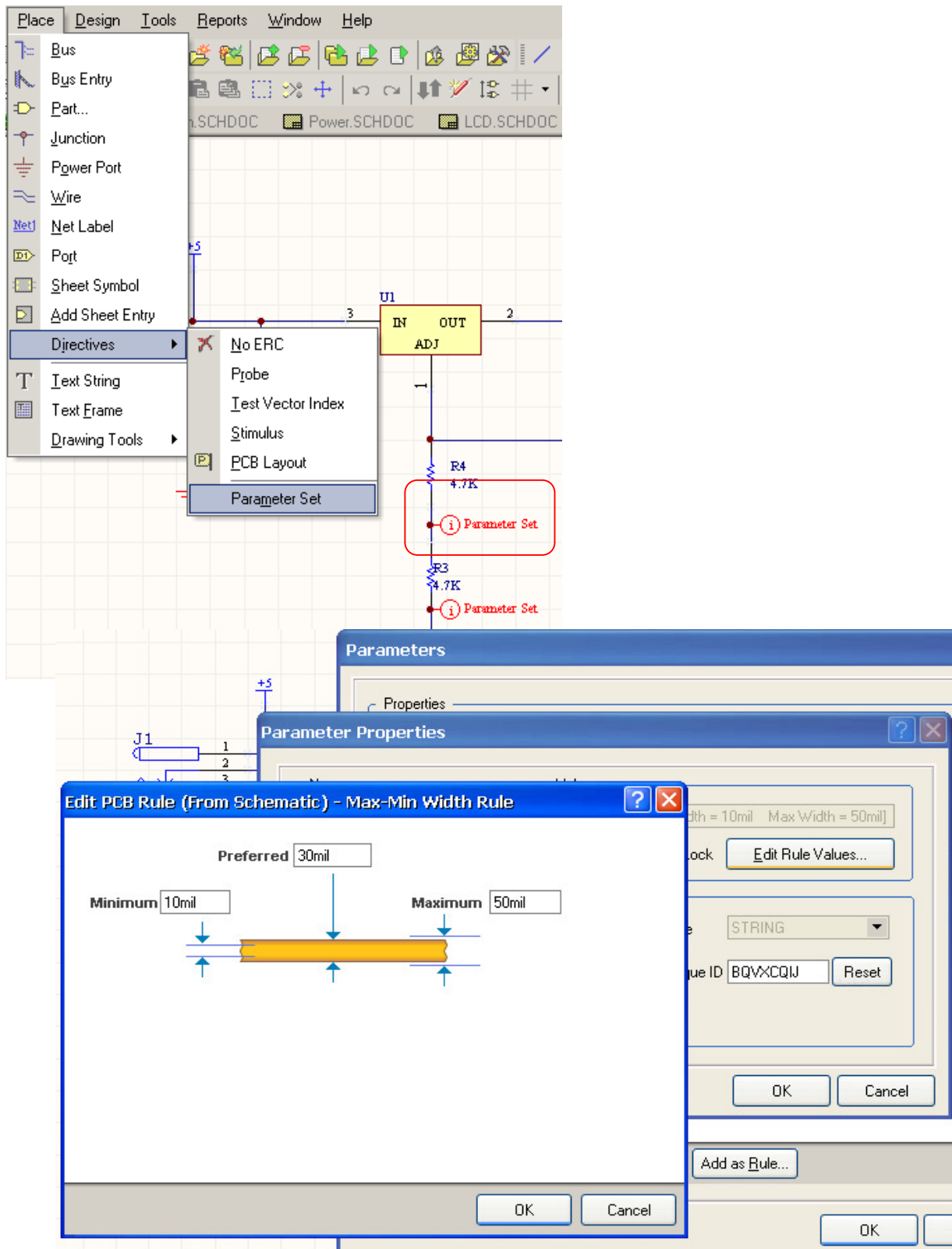


3 components inter-connected

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

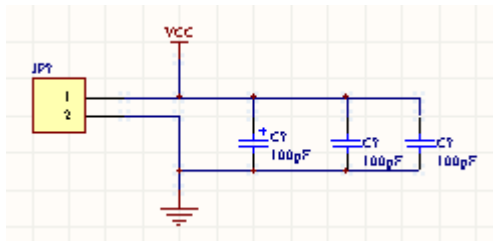
Prepare by : HK Sim simhkeng@gmail.com

Specify a particular routing width on schematic – Parameter Set



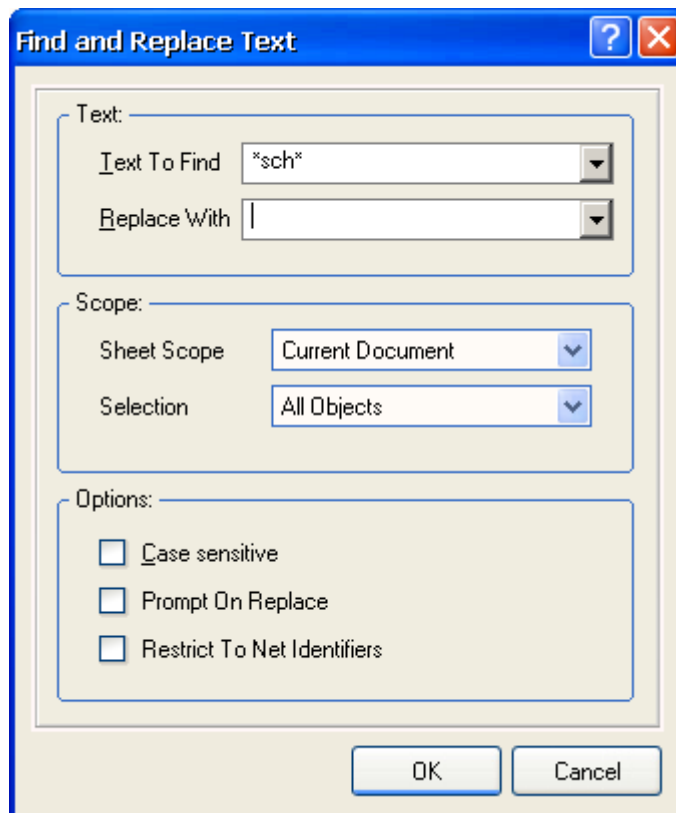
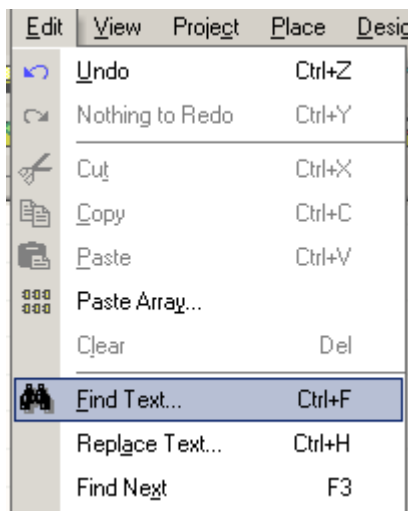
VCC and GND Connection

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 be connected to Power source input by connector. Otherwise, these two pin will not be connected.



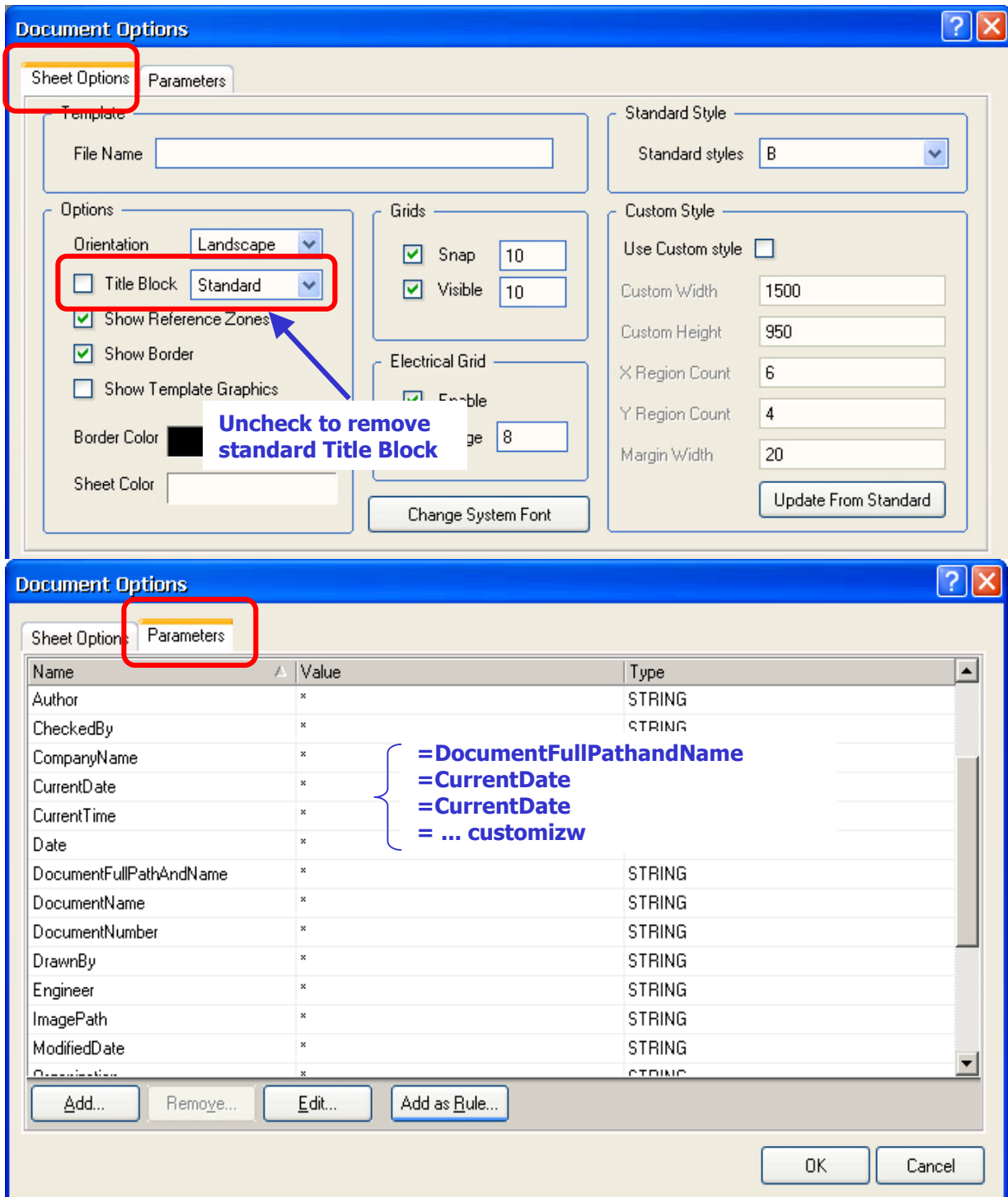
Find/Replace text

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



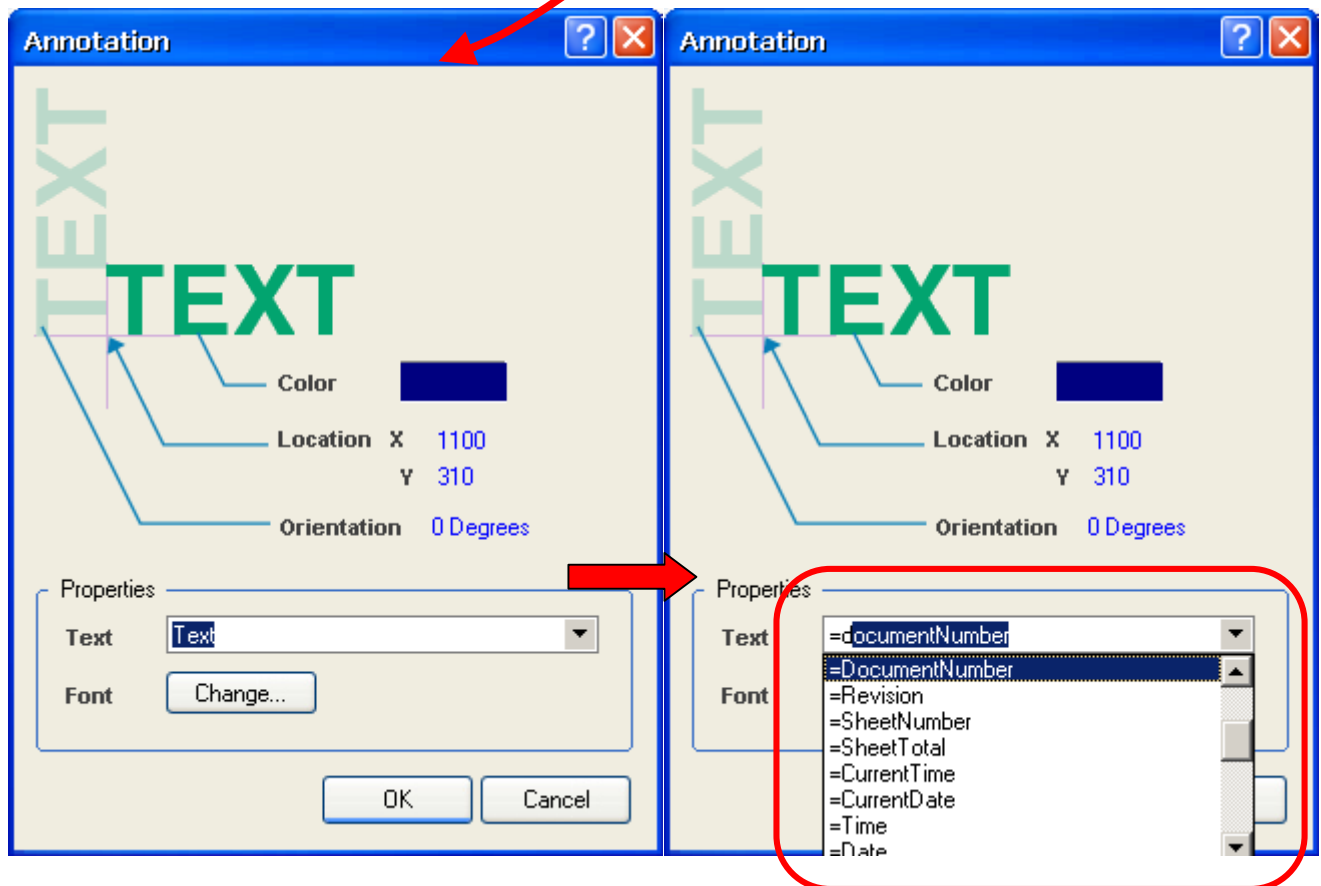
Customize Document Title Block

[Design]/[Option]



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

?????



Pace/Port

Double click

Start Changes



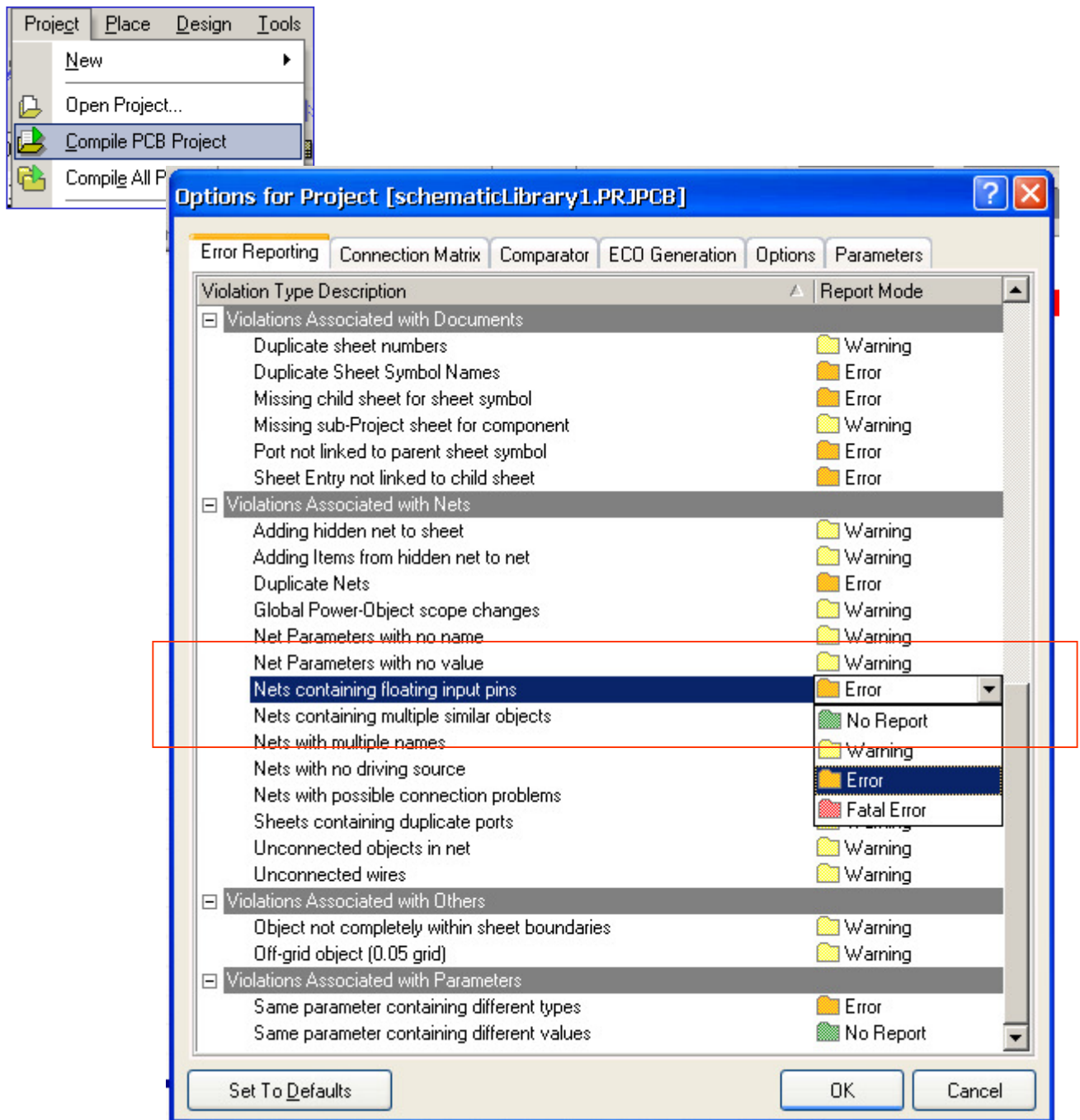
Prepare by : HK Sim simhkeng@gmail.com

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



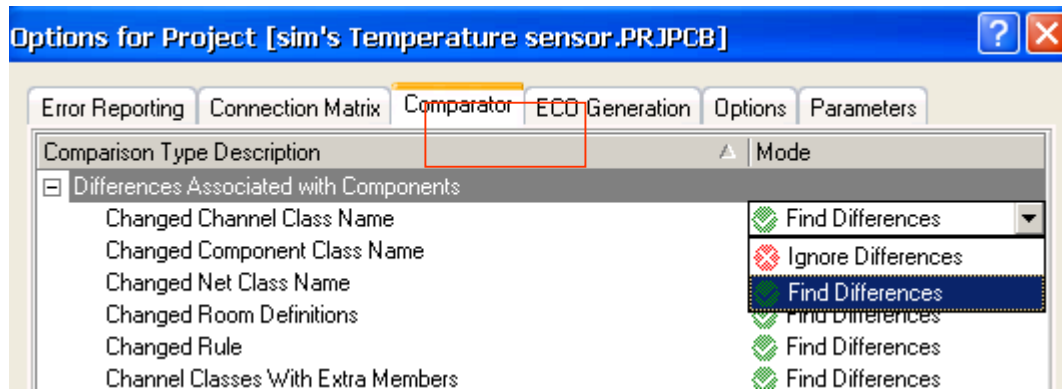
Set error message color

[Project]/[Project Option] – Connection Matrix



Set comparison criteria between a new and old schematic ?

[Project]/[Project Option] – Comparator



Avoid duplicate designator

System assign designator - [Tool]/[Annotate]

Annotate

Schematic Annotation Configuration

1 Up then across
2 Down then across
3 Across then up
4 Across then down

Match By Parameters

- ☐ Class
- ☒ Comment
- ☐ Gender
- ☐ Manufacturer
- ☐ Note
- ☐ Orientation
- ☐ Positions

Schematic Sheets To Annotate

Schematic Sheet	Designator Index Control	Add Suffix
	Start Index	Suffix
<input checked="" type="checkbox"/> amp.SCHDOC	<input type="checkbox"/> 1	
<input type="checkbox"/> IV.SCHDOC	<input type="checkbox"/> 1	
<input type="checkbox"/> main.SCHDOC	<input type="checkbox"/> 1	
<input type="checkbox"/> Power.SCHDOC	<input type="checkbox"/> 1	

Select schematics to be annotate
When all selected, all components will have unique name

Proposed Change List

Current Designator	Sub	Proposed Designator	Sub	Location of Part
R2		R2		IV.SCHDOC
R2		R2		main.SCHDOC
R2		R2		amp.SCHDOC
R3		R3		main.SCHDOC
R3		R3		amp.SCHDOC
R3		R3		IV.SCHDOC
R4		R4		main.SCHDOC
R5		R5		main.SCHDOC
R6		R6		main.SCHDOC
R7		R7		main.SCHDOC
R8		R8		main.SCHDOC
R9		R9		main.SCHDOC
R?		R?		main.SCHDOC
R?		R?		main.SCHDOC
R?		R?		main.SCHDOC
R?		R?		main.SCHDOC
R?		R?		main.SCHDOC
R?		R?		main.SCHDOC
S?		S?		main.SCHDOC
U1	1	U1	1	amp.SCHDOC
U1				
U?		U?		main.SCHDOC
U?		U?		Power.SCHDOC
U?		U?		Power.SCHDOC
U?		U?		Power.SCHDOC
Y1		Y1		main.SCHDOC

Change all to "?" e.g. R?

Update Changes List Reset Designators

Engineering Change Order

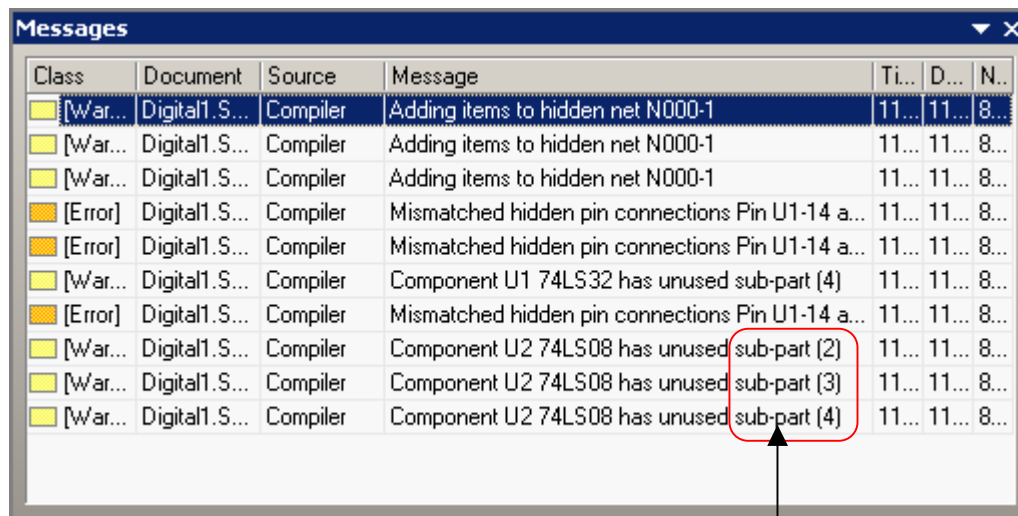
Indicate changes

Action	Affected Object	Modifications	Affected Document	Status
				Check Done
Annotate Component(4)				
Modify	R? -> R1	In	amp.SCHDOC	
Modify	R? -> R2	In	amp.SCHDOC	
Modify	R? -> R3	In	amp.SCHDOC	
Modify	U?(1) -> U1(1)	In	amp.SCHDOC	

Validate Changes Execute Changes Report Changes...

Start Changes

Error example



Class	Document	Source	Message	Ti...	D...	N..
[War...]	Digital1.S...	Compiler	Adding items to hidden net N000-1	11...	11...	8...
[War...]	Digital1.S...	Compiler	Adding items to hidden net N000-1	11...	11...	8...
[War...]	Digital1.S...	Compiler	Adding items to hidden net N000-1	11...	11...	8...
[Error]	Digital1.S...	Compiler	Mismatched hidden pin connections Pin U1-14 a...	11...	11...	8...
[Error]	Digital1.S...	Compiler	Mismatched hidden pin connections Pin U1-14 a...	11...	11...	8...
[War...]	Digital1.S...	Compiler	Component U1 74LS32 has unused sub-part (4)	11...	11...	8...
[Error]	Digital1.S...	Compiler	Mismatched hidden pin connections Pin U1-14 a...	11...	11...	8...
[War...]	Digital1.S...	Compiler	Component U2 74LS08 has unused sub-part (2)	11...	11...	8...
[War...]	Digital1.S...	Compiler	Component U2 74LS08 has unused sub-part (3)	11...	11...	8...
[War...]	Digital1.S...	Compiler	Component U2 74LS08 has unused sub-part (4)	11...	11...	8...

Indicate un-use parts

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	Re	SW DIP-8	DIP-SW_8WAY_SMD	
R5	U1	TL082P	JFET-Input Operational Amplifier DIP-8/D11	TL082P
R6	Re	L7805CP	Positive Voltage Regulator SFM-T3/132V	L7805CP
R7	Re	L7812CV	Positive Voltage Regulator SFM-T3/132V	L7812CV
R8	Re	L7912CV	Negative Voltage Regulator SFM-T3/132V	L7912CV
R9	Re	U5	TL082P	JFET-Input Operational Amplifier DIP-8/D11
R10	Re	U6	AT89C2051	AT89C2052
R11	Re	U7	MAX232ACPE	+5V Powered, Multi-Channel V-DIP-16/14-5
U1	Re	Y1	XTAL	Crystal Oscillator BCY-W2/D3.1
U2	LM			
U3	TC			
U4	PIC			

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 display	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 Connect	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 Microcontroller	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)