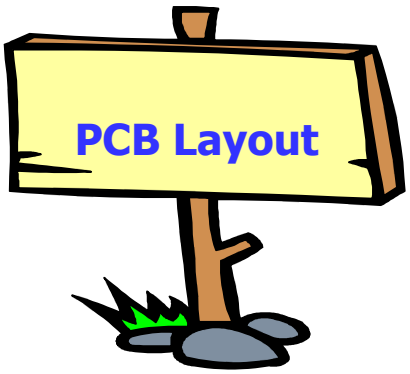
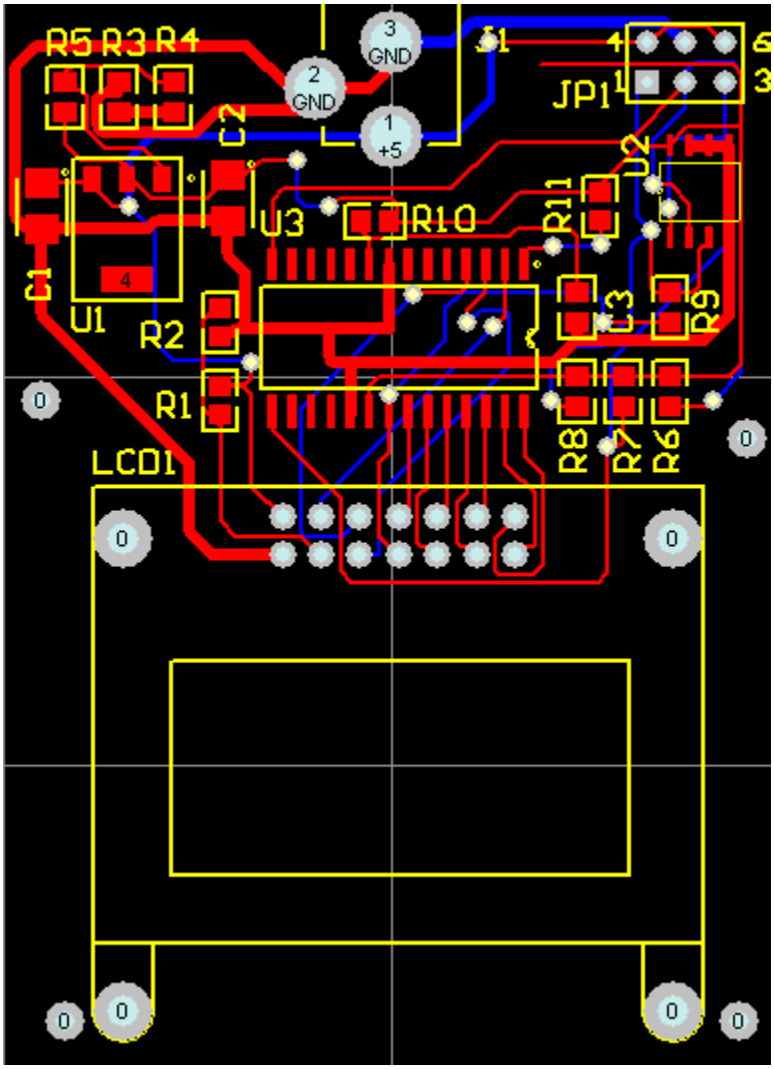


Prepare by : HK Sim simhkeng@gmail.com

Date : 22 Dec 05



Prepare by : HK Sim simhkeng@gmail.com

Main steps from Schematic to PCB

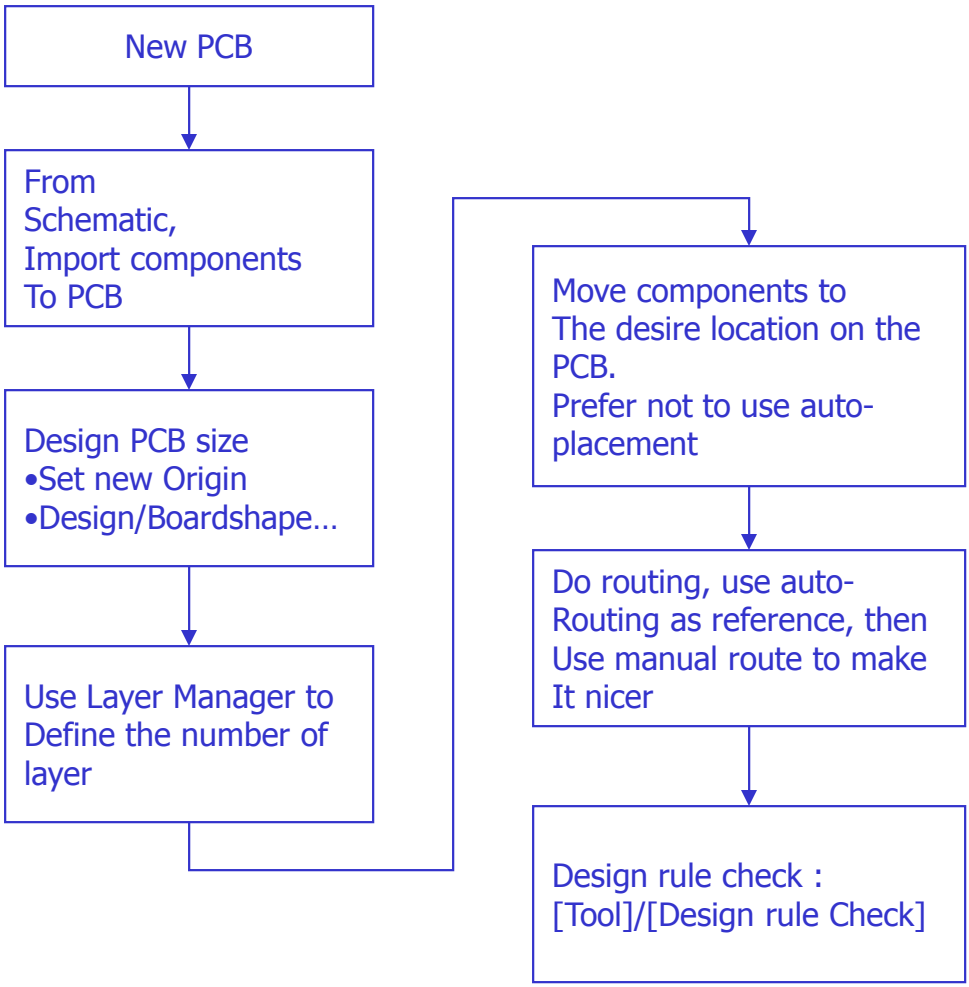
Move from schematic to PCB

Define PCB size

Bring component from schematic to PCB

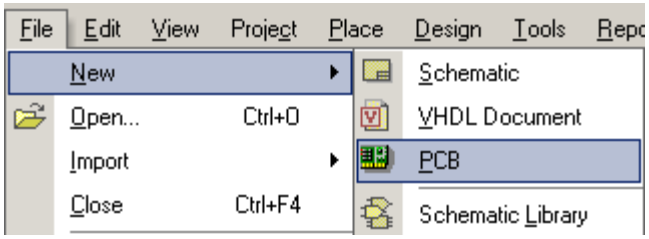
Move the components to the desire position

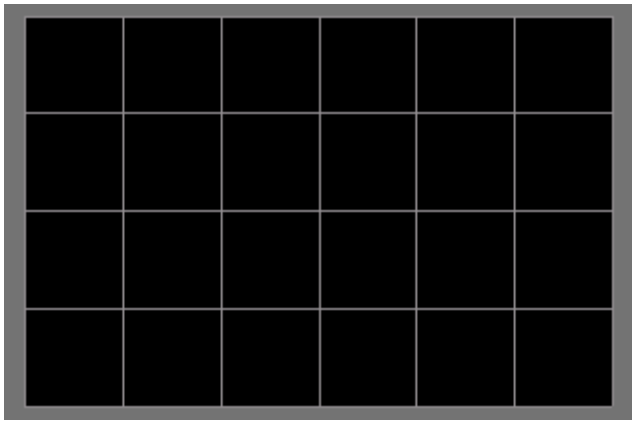
Layout the path



Prepare by : HK Sim simhkeng@gmail.com

Move from schematic to PCB

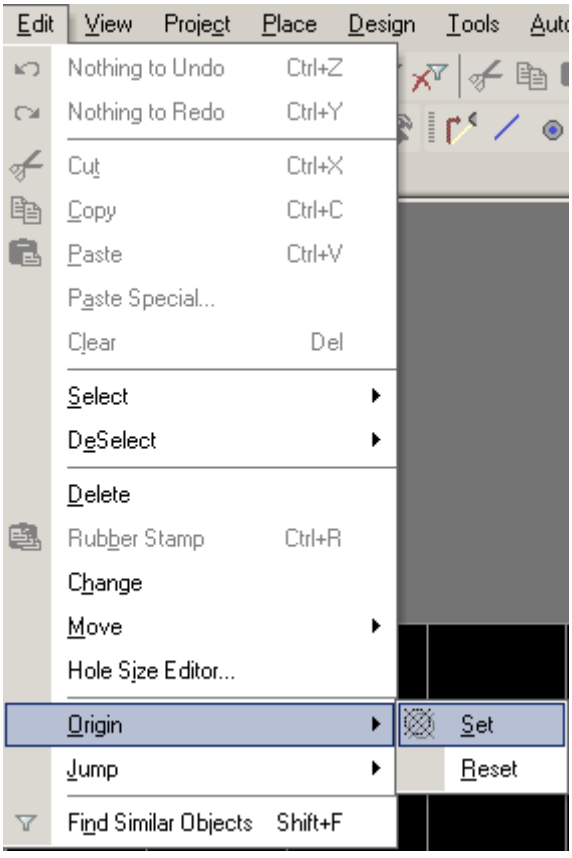


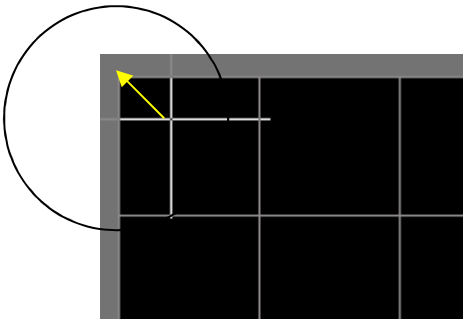


Define PCB size

Set reference Origin on PCB

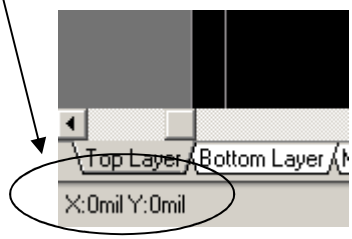
[Edit] / [Origin] / [Set]





Move the cursor to the point that you want to set as origin and click.

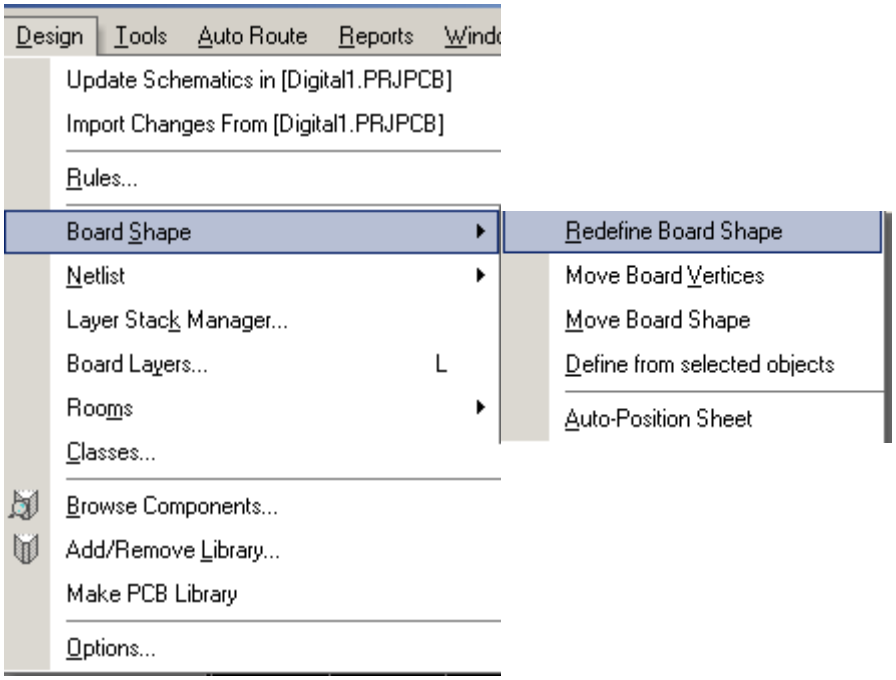
After that, this point's position should read as [0,0]



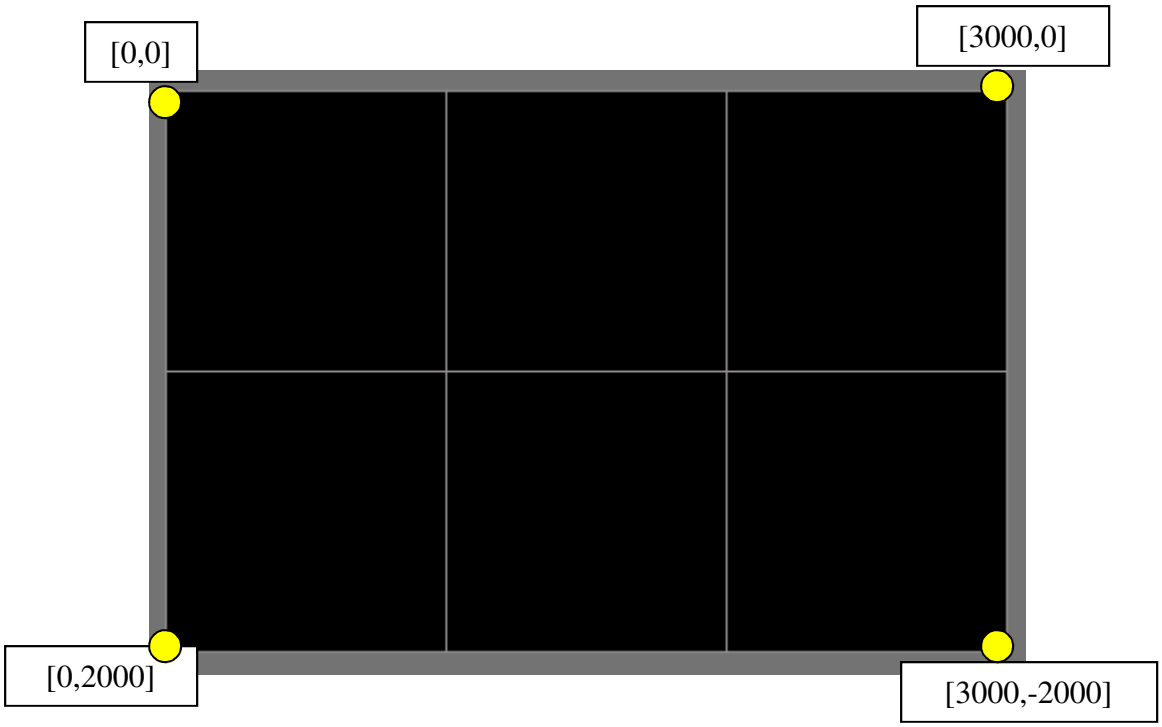
Prepare by : HK Sim simhkeng@gmail.com

Change the board shape use

[Design]/[Board Shape]/[Redefine Board Shape]



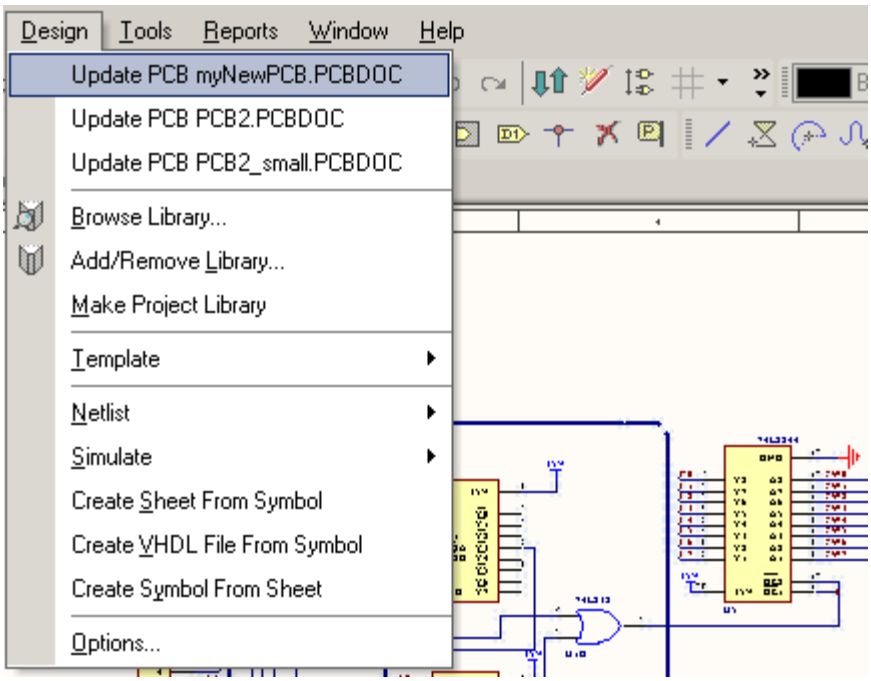
The PCB will change from black to green. Start from the set origin [0,0], use the mouse to move around in a close rectangle box to define the board side. See the bottom co-ordinate to know the dimension.



Prepare by : HK Sim simhkeng@gmail.com

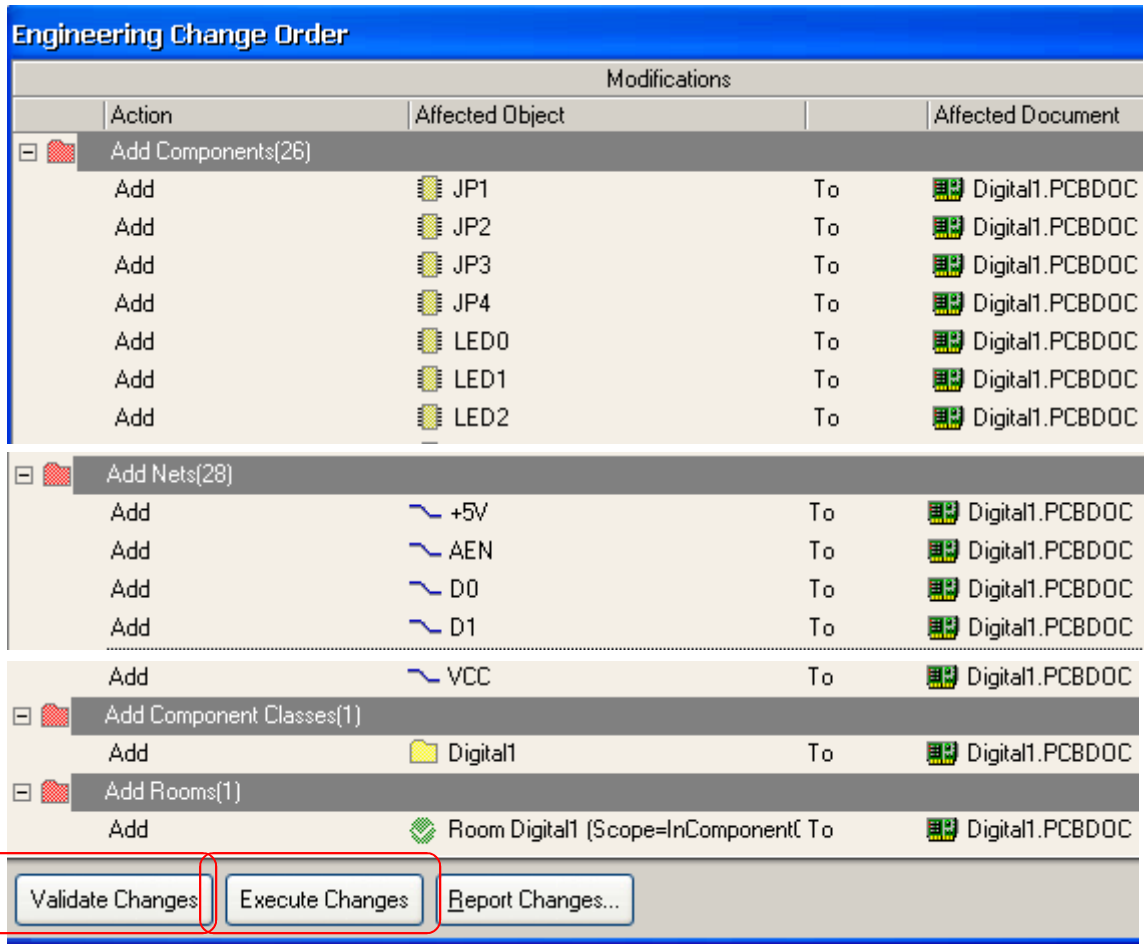
Bring component from schematic to PCB

To bring components from schematic into PCB , while you are in the schematic ,use the following to bring the component to the correct PCB



Prepare by : HK Sim simhkeng@gmail.com

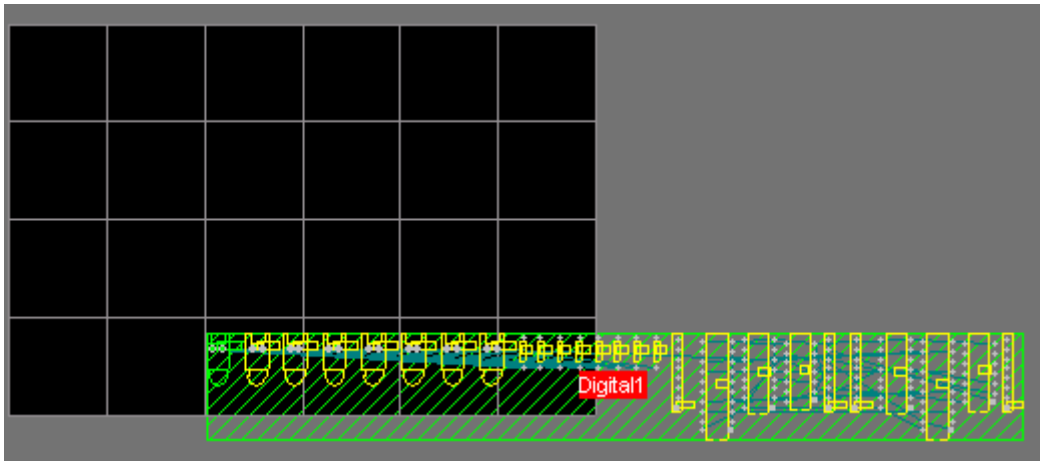
This because when the PCB have no components, update the specified PCB will bring all components from the schematic into the PCB.



Click [Validate Changes] to check if there is any error .

Click [Execute Changes] to initiate the change.

After click Execute Changes, component footprints are brought into PCB



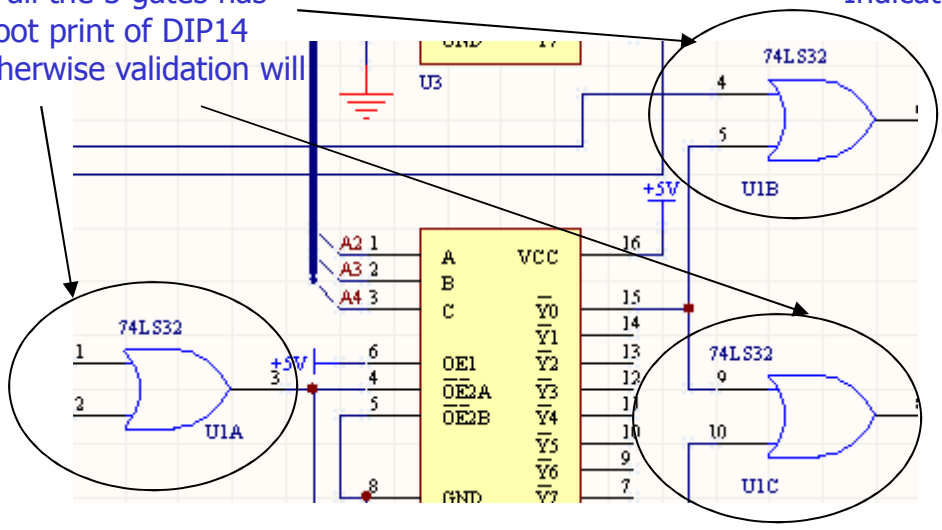
Prepare by : HK Sim simhkeng@gmail.com

Error example.

Add	R8	To	Digital1.PCBDOC
Add	U1	To	Digital1.PCBDOC
Add	U2	To	Digital1.PCBDOC
Add	U3	To	Digital1.PCBDOC
Add	U4	To	Digital1.PCBDOC
Add	U5	To	Digital1.PCBDOC
Add	U6	To	Digital1.PCBDOC

Indicate there is error

Make sure all the 3 gates has common foot print of DIP14 picture, otherwise validation will give error.



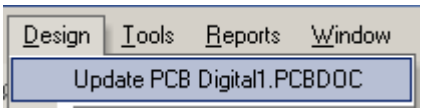
Prepare by : HK Sim simhkeng@gmail.com

Update changes between Schematic and PCB

During the layout of PCB, when make changes on schematic, can use the following method to update the PCB :

[Project] / [compile PCB project] – if there is no logical error, the compilation will be quiet . I.e Nothing display

Use update PCB as follows :

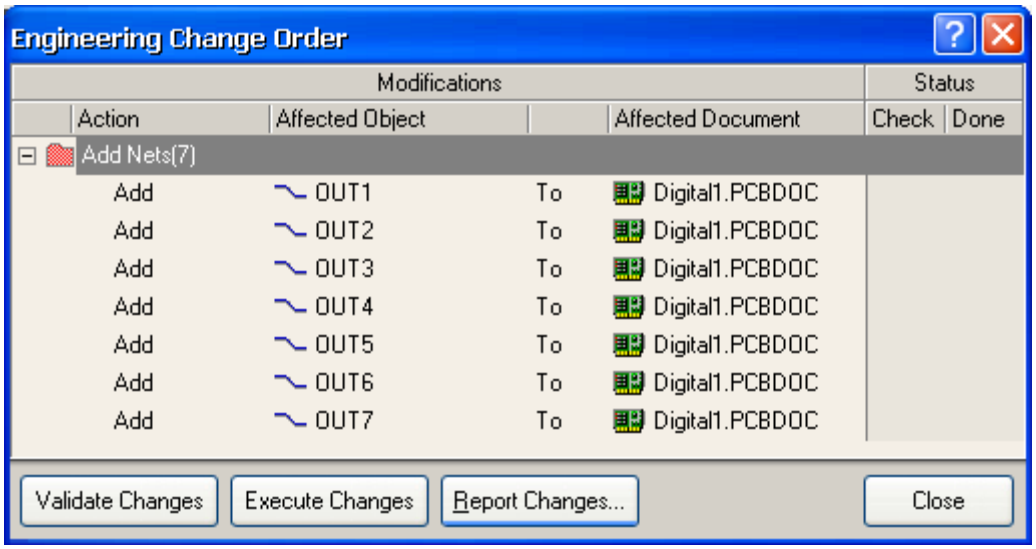


A list of different items (different from the previous PCB) is then displayed.

Use [Validate Changes] – to check the changes

Use [Execute Changes] – to implement the changes. Then the PCB will be updated with the new changes

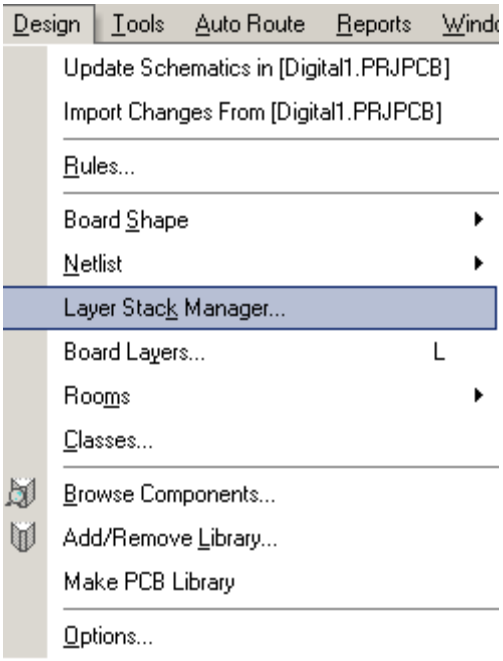
Use [Close] – to end changes



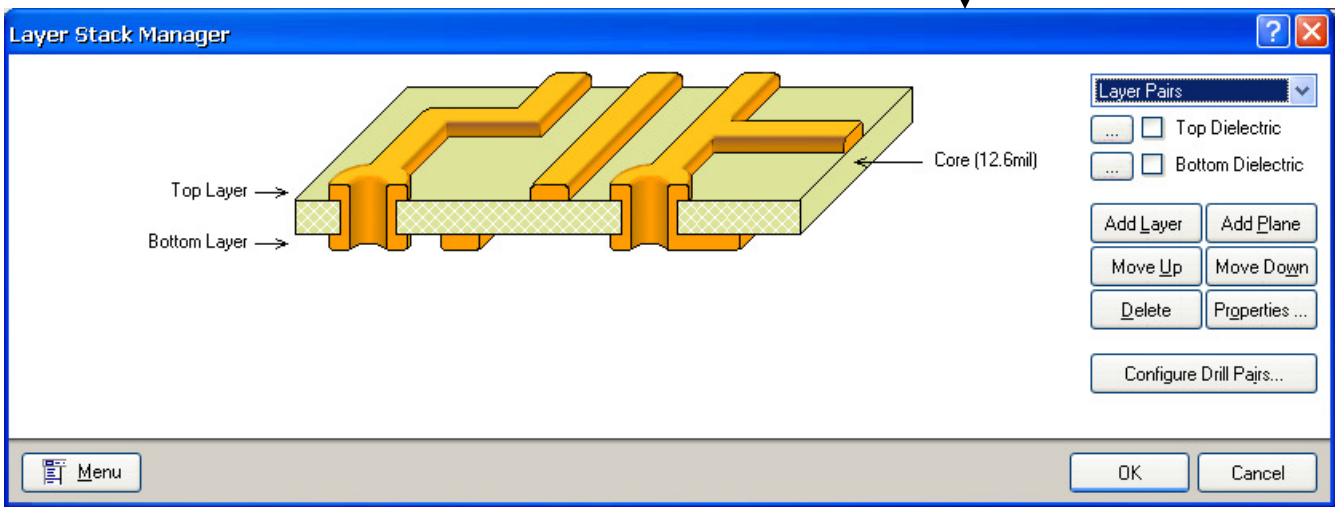
Prepare by : HK Sim simhkeng@gmail.com

Add more layer to the PCB

[Design]/[Layer Stack Manager]



By default is a 2 layer PCB



Prepare by : HK Sim simhkeng@gmail.com

Layer definition

Signal layers

Top layer

mid layer 1 to 30

Bottom layer

Internal Planes

- 16 layers available for Power/Gnd planes.
- Net can be assigned to these layers .
- Mutli-layer pads and via automatically connect to these planes. Plane layers can be split into any number of regions and each regions can be assigned to different net
- Objects that are placed on the plane define regions of no copper

Silkscreen layers

Top and bottom layers for User to display component outlines

Mechanical layers

16 mechanical layers are provided for fabrication and assembly detail such as dimensions, alignment targets, annotation or other details.

Solder Mask

Top and bottom solder mask layer

Paste mask

Top and bottom paste mask layers are provided to generate the artwork which is used to manufacture stencils to deposit solder paste onto surface mount pads on PCB with surface mount devices.

Drill Drawing

Showing a unique symbol for each hole size at each location. 3 symbol styles : coded symbol, alphabetical codes (A, B, C etc) or the assigned size.

Drill Guide

Plots all holes in the layout. Drill guide is also call pad masters

Prepare by : HK Sim simhkeng@gmail.com

Keep out area

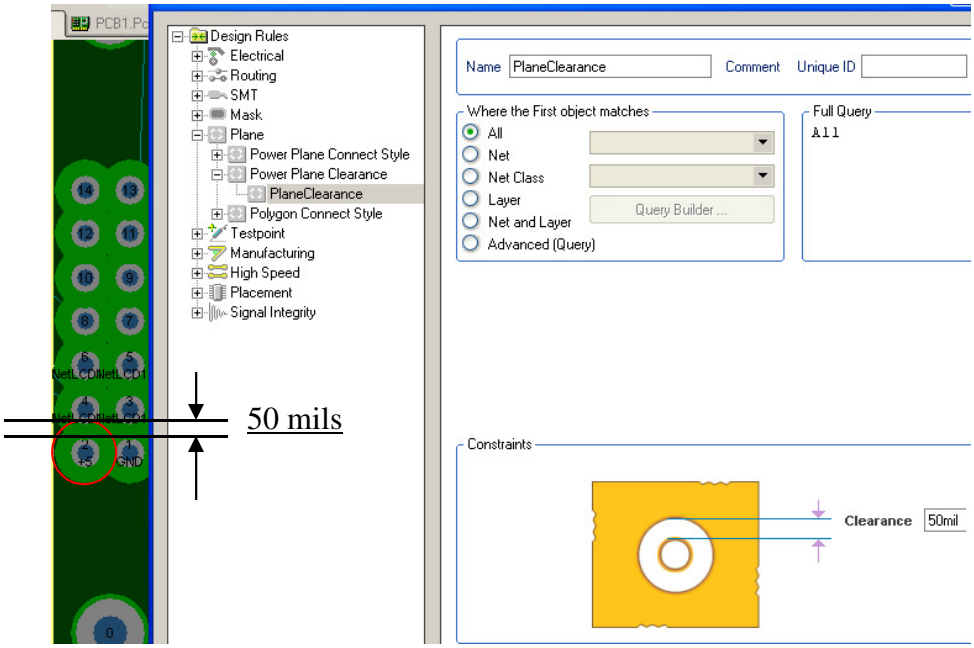
Define the regions where components and routes can validly be placed.

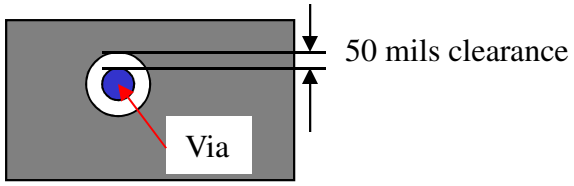
Components cannot be placed over an object on the keep out layer and routes cannot cross an object on the keep out layer

Multi-layer

Objects placed on this layer will appear on all copper layers. This is typically use for through hole pads and via.

Power Planes Clearance setting

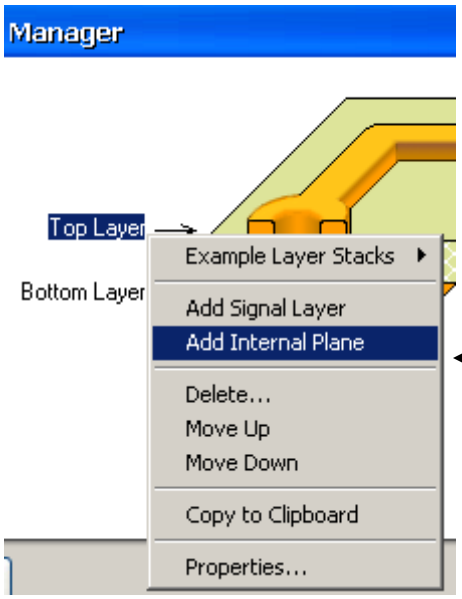




Prepare by : HK Sim simhkeng@gmail.com

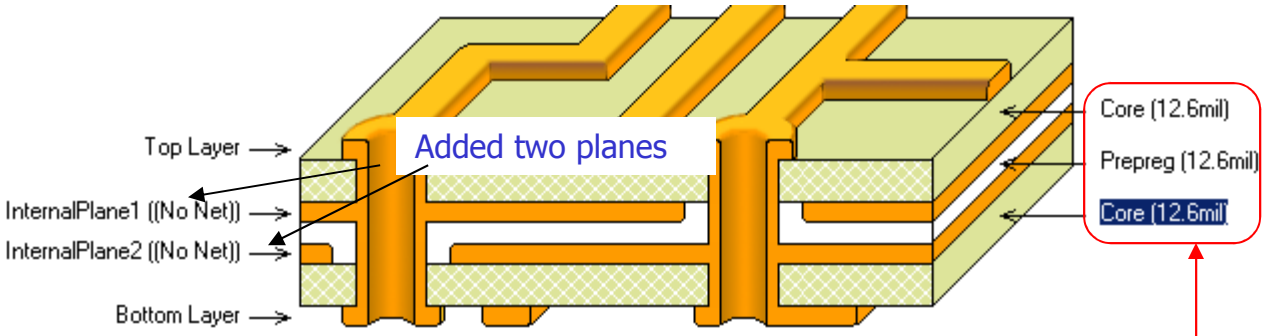
Add internal Planes

Add internal plan means Power / ground plan. It is normally 2nd plane is ground plan and 3rd plan is power plan



Select top layer, Right click and select

Add internal plane or Signal layer

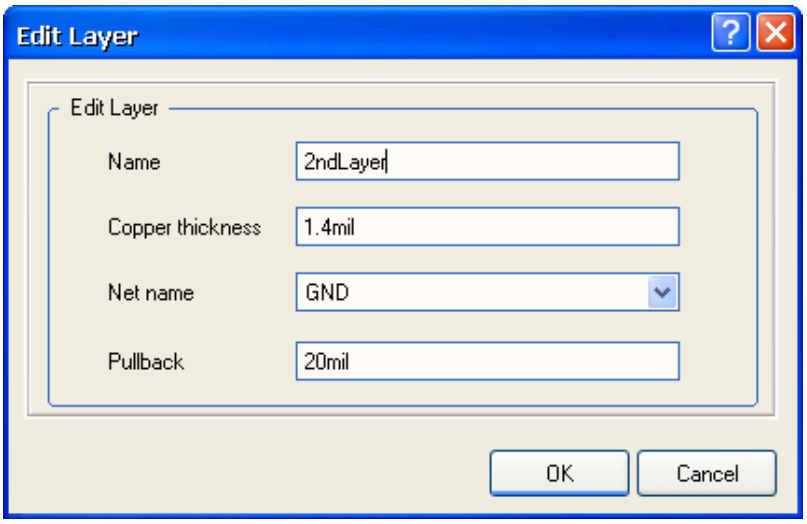


Mechanical layer thickness can be changed

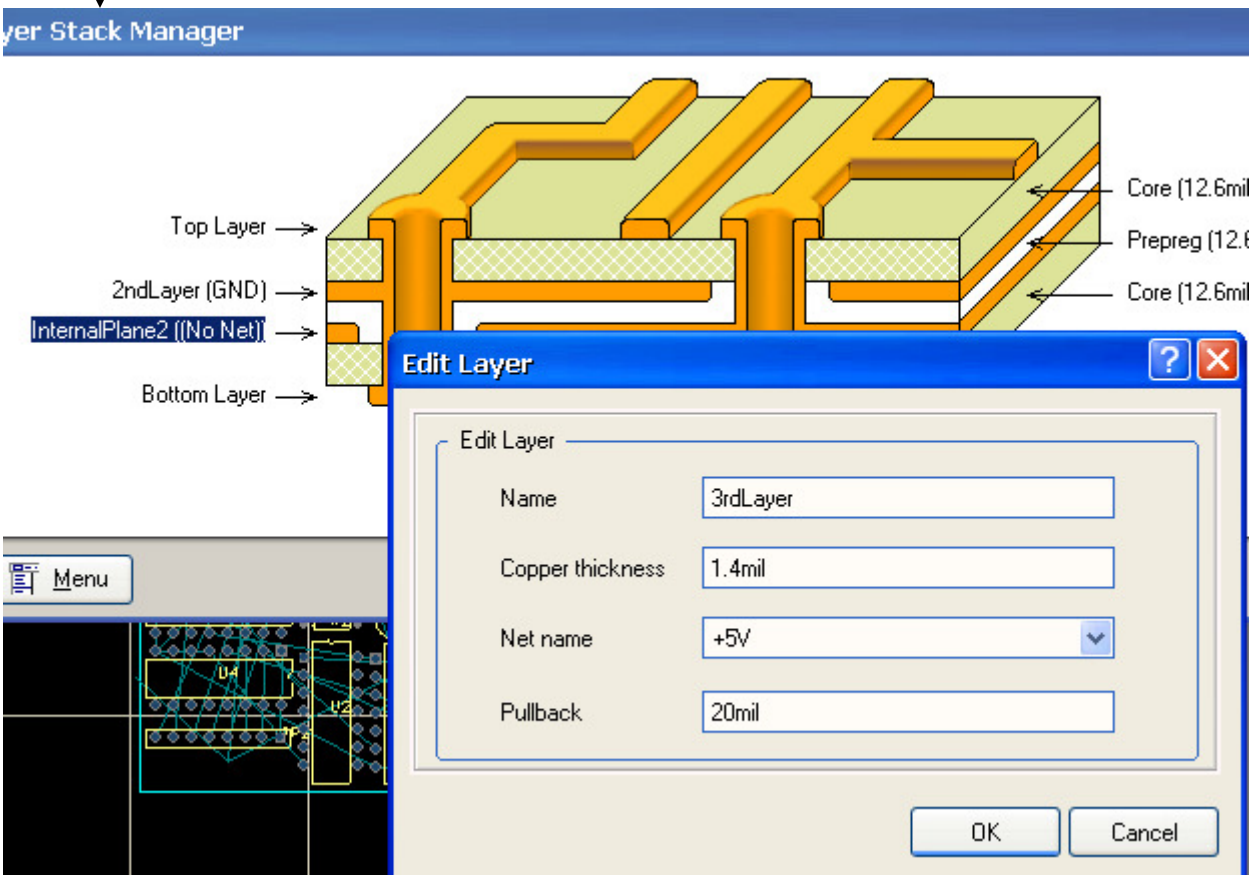
Prepare by : HK Sim simhkeng@gmail.com

Add internal Planes

Add ground plane, Net name is GND as in schematic



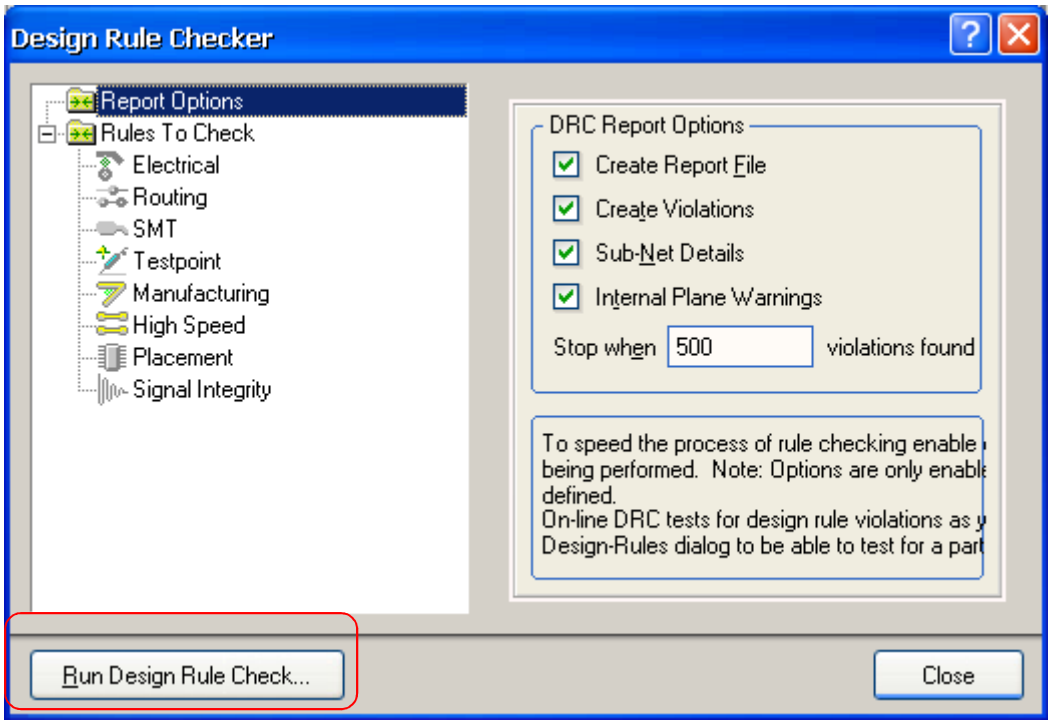
Add Power (+5V) plane, Net name is +5V as in schematic



Prepare by : HK Sim simhkeng@gmail.com

Design Rule Check for PCB Layout

[Tool]/[Design Rule Check]



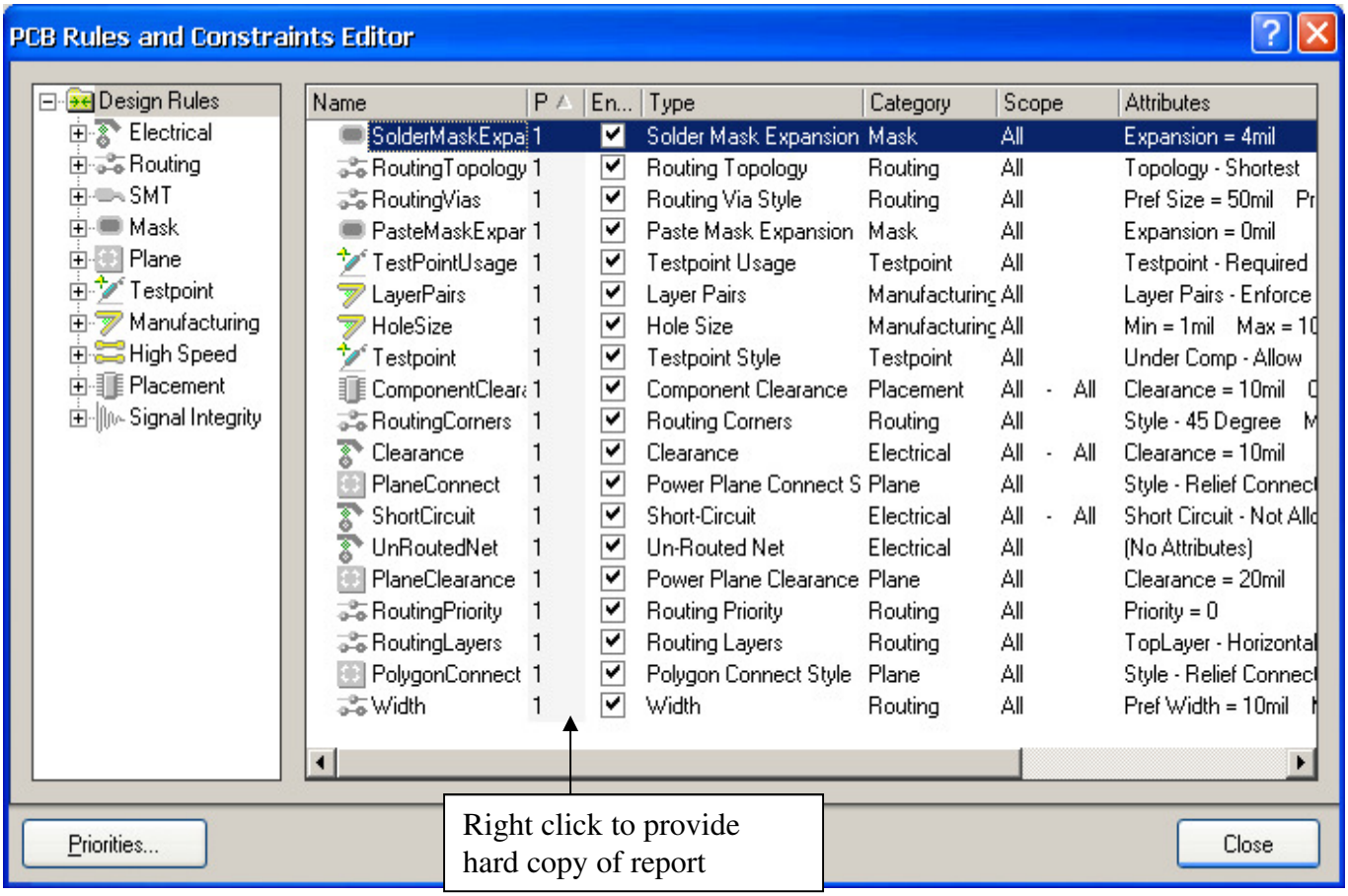

```
Protel Design System Design Rule Check
PCB File : \simHW\Digital1\PCB2_small.PCBDOC
Date     : 12/6/2005
Time     : 7:08:14 PM

Processing Rule : Hole Size Constraint (Min=1mil) (Max=100mil) (All)
Rule Violations :0

Processing Rule : Width Constraint (Min=10mil) (Max=10mil) (Prefered=10mil) (All)
Rule Violations :0

Processing Rule : Clearance Constraint (Gap=10mil) (All),(All)
Rule Violations :0

Processing Rule : Broken-Net Constraint ( (All) )
Rule Violations :0

Processing Rule : Short-Circuit Constraint (Allowed=Not Allowed) (All),(All)
Rule Violations :0

Violations Detected : 0
Time Elapsed        : 00:00:00
```

Zero error !

Prepare by : HK Sim simhkeng@gmail.com

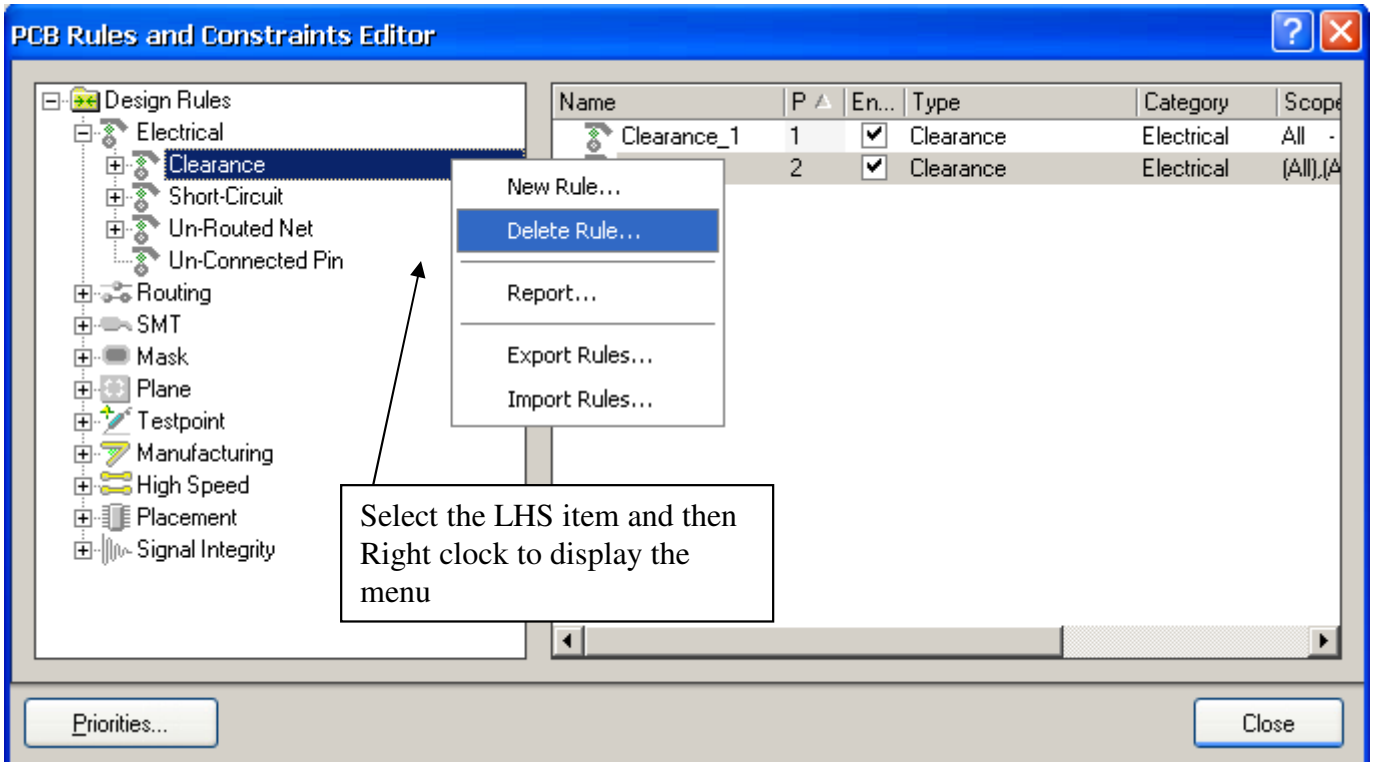
Design Rule Setting

[Design] / [Rule]



Prepare by : HK Sim simhkeng@gmail.com

Select item at the LHS box and do a right click, you can add / delete rules … etc



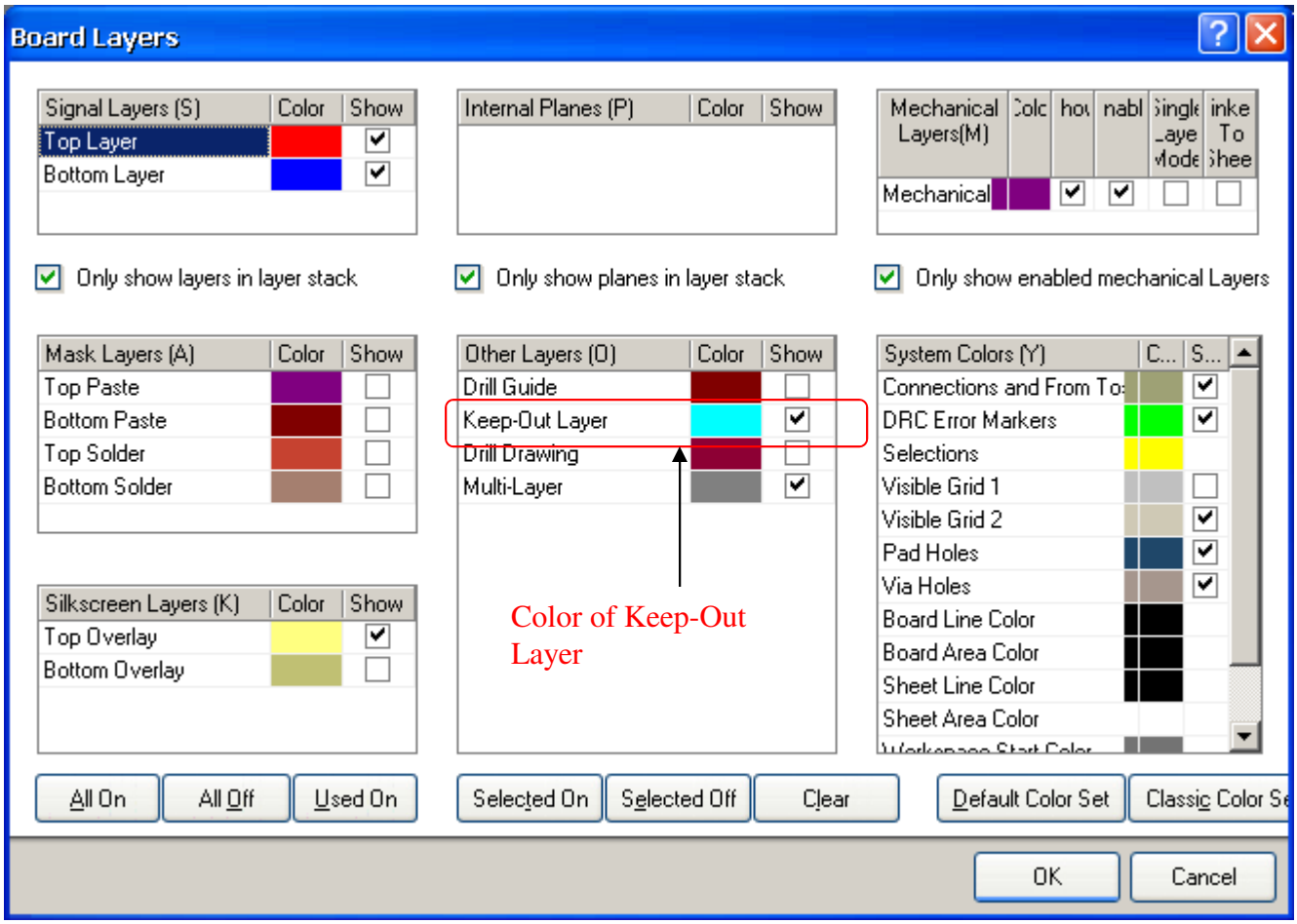
Prepare by : HK Sim simhkeng@gmail.com

Auto Placement

It is recommended that key component should not use auto-placement due to performance and appearance.

To do an auto-placement, you **must define** the keep-out area where no routing will be pass through that area..

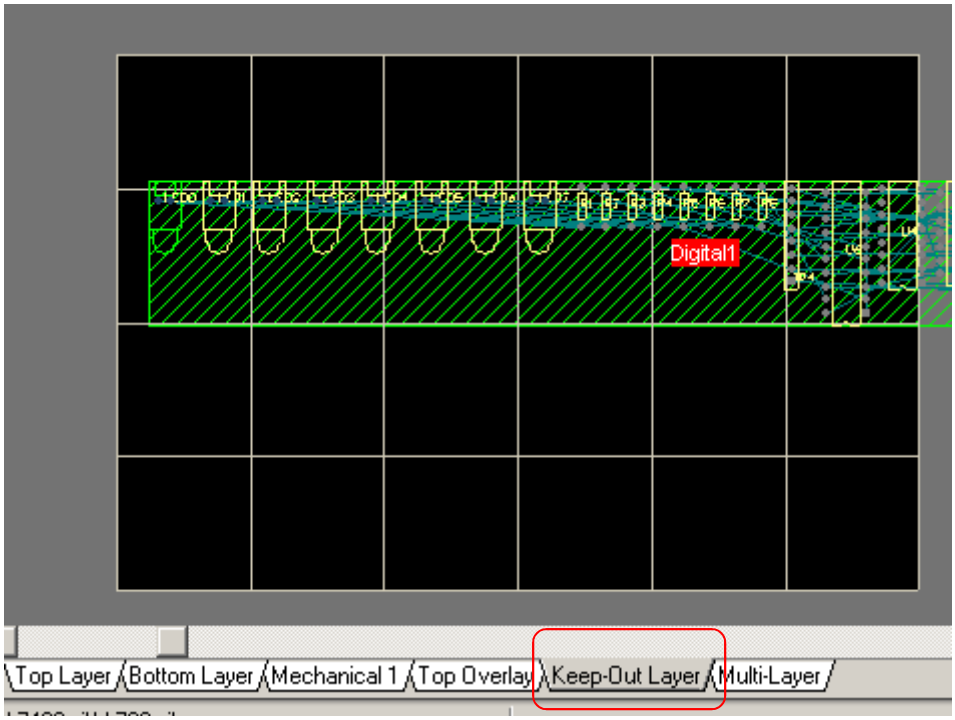
Keep-out area can defined on the Keep-out Layer

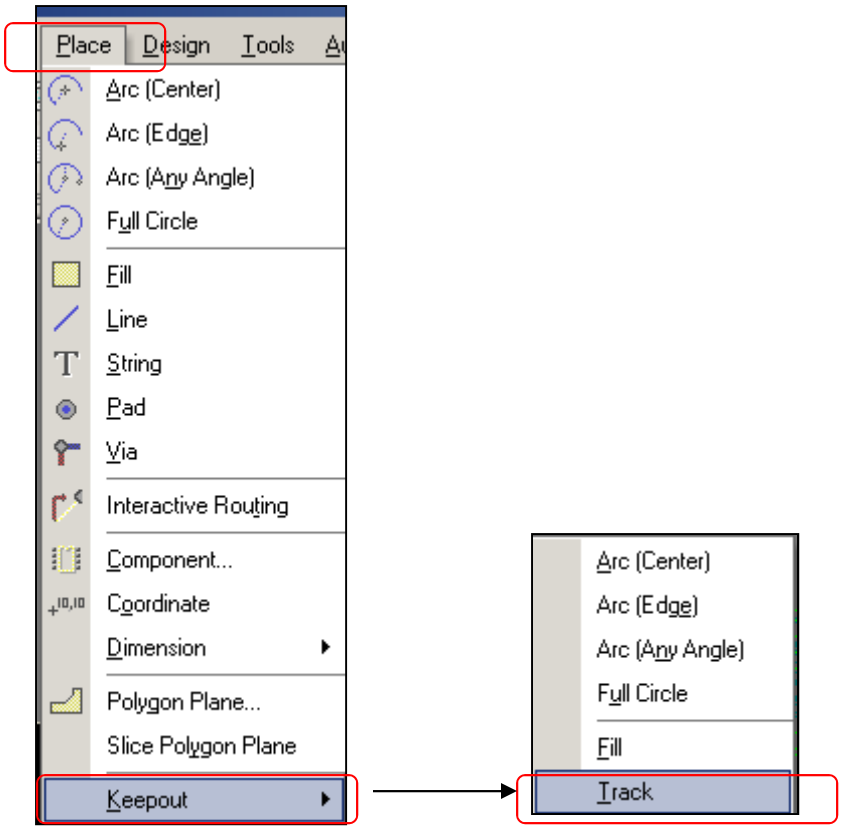


Short cut to get this menu Press **L** on the PCB board

Prepare by : HK Sim simhkeng@gmail.com

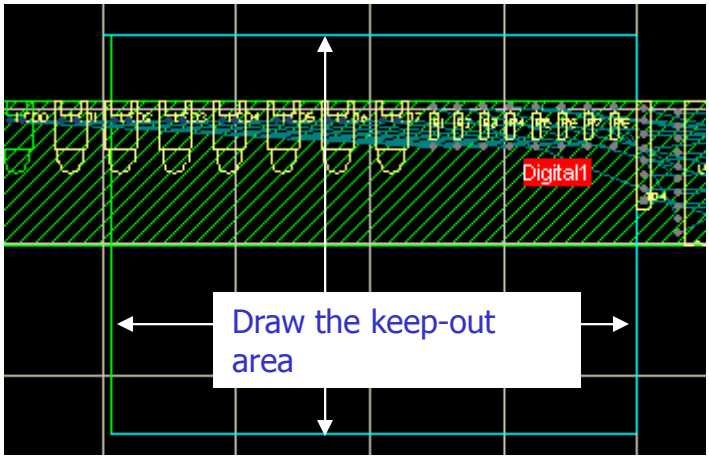
Define Keep Out Area



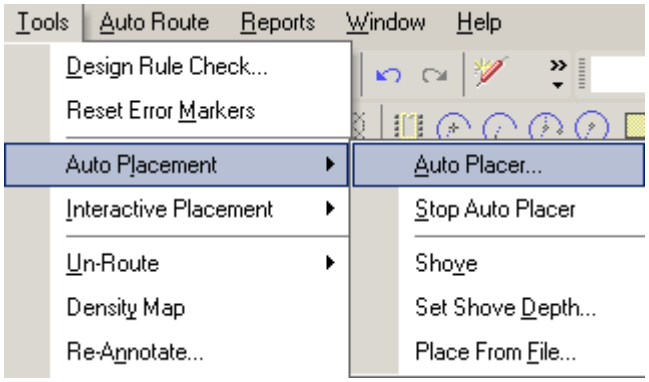


Prepare by : HK Sim simhkeng@gmail.com

Draw the keep out area



After defined the keep-out area, use [Tool]/[Auto Placement]/[Auto Placer] to auto-place the component. Once can define the constraint of the auto-placing.

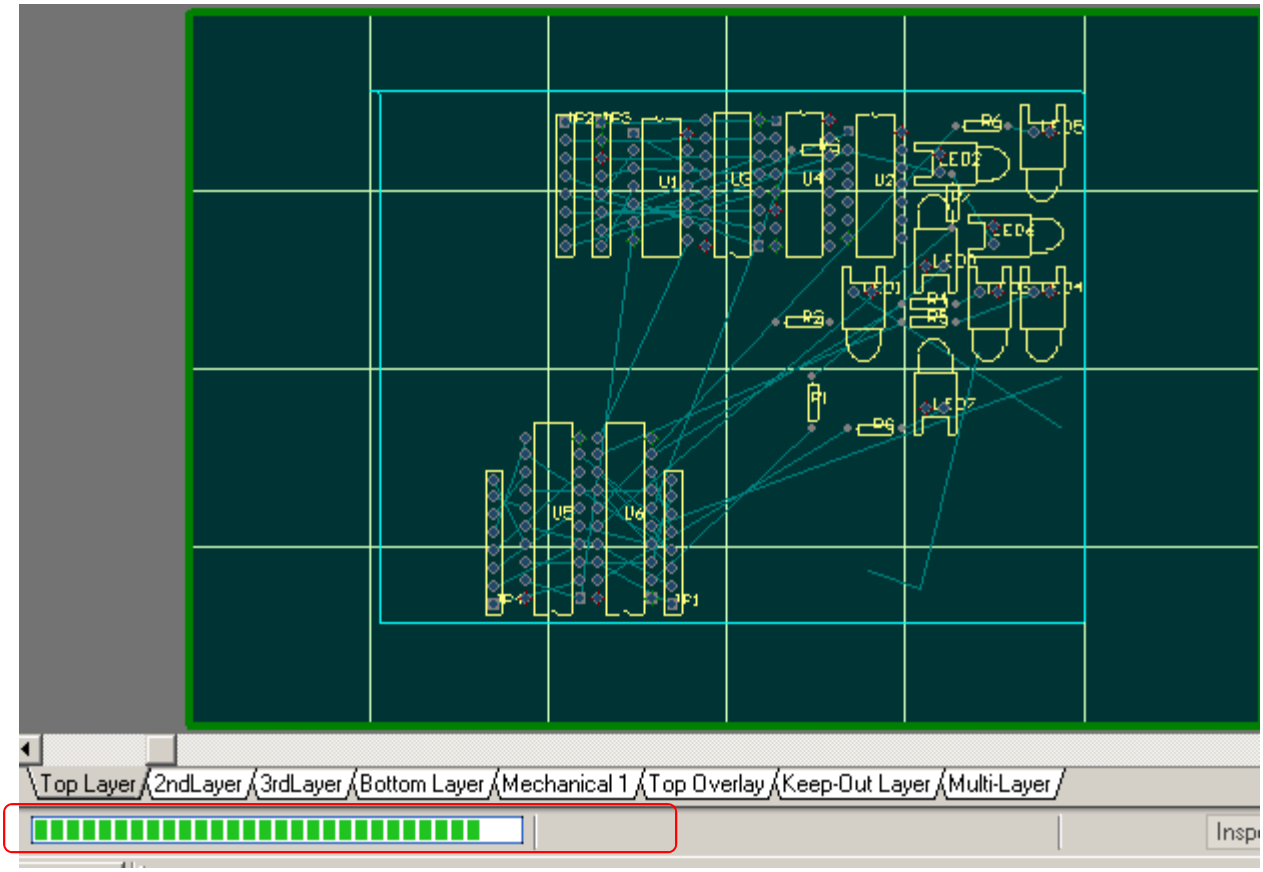


Prepare by : HK Sim simhkeng@gmail.com

Auto-placement is in progress

All components are within the keep-out area

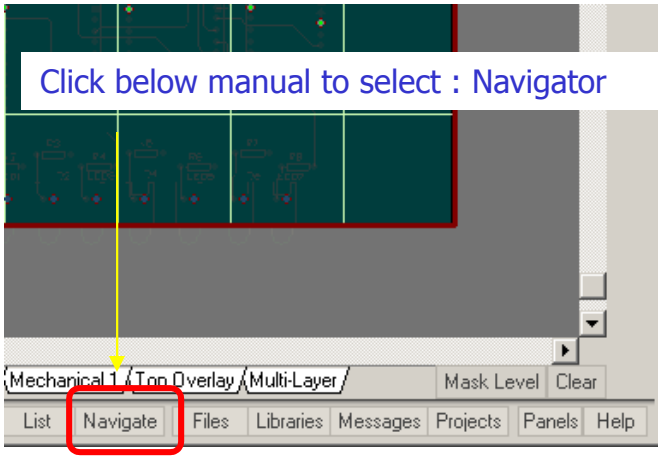
Status bar indicate the progress

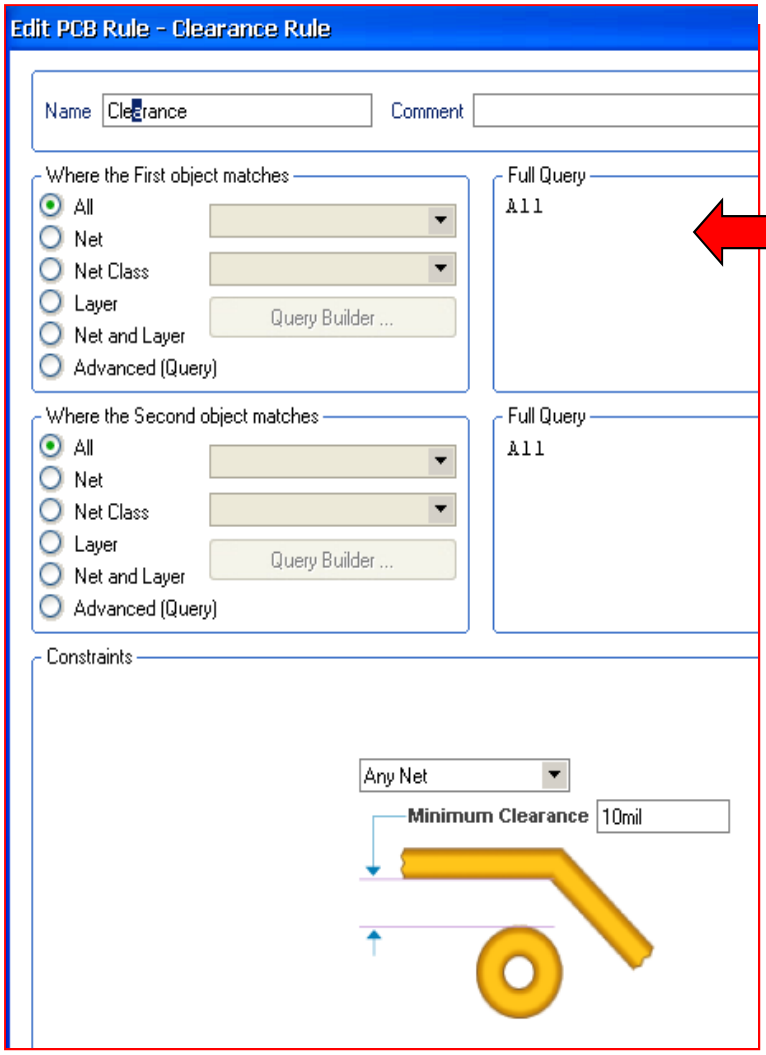


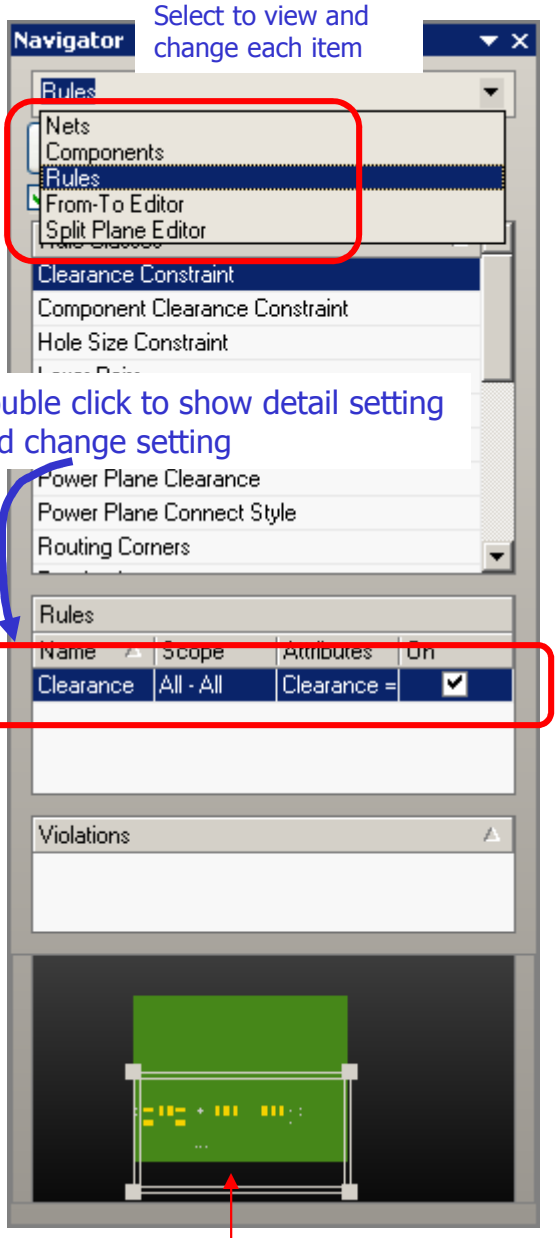
Prepare by : HK Sim simhkeng@gmail.com

See a component or net on a PCB

Use Navigator







Double click to show detail setting and change setting

Use the frame to position PCB for viewing

Prepare by : HK Sim simhkeng@gmail.com

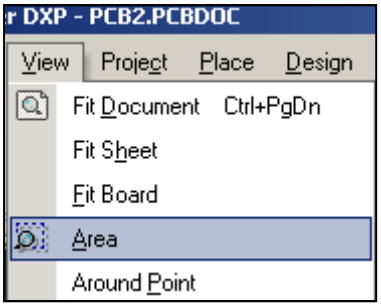
Position a PCB for viewing

[View]/[Fix Board] - VF

[View]/[Fix Sheet] - VH

[View]/[Fix Document] - VD

Important

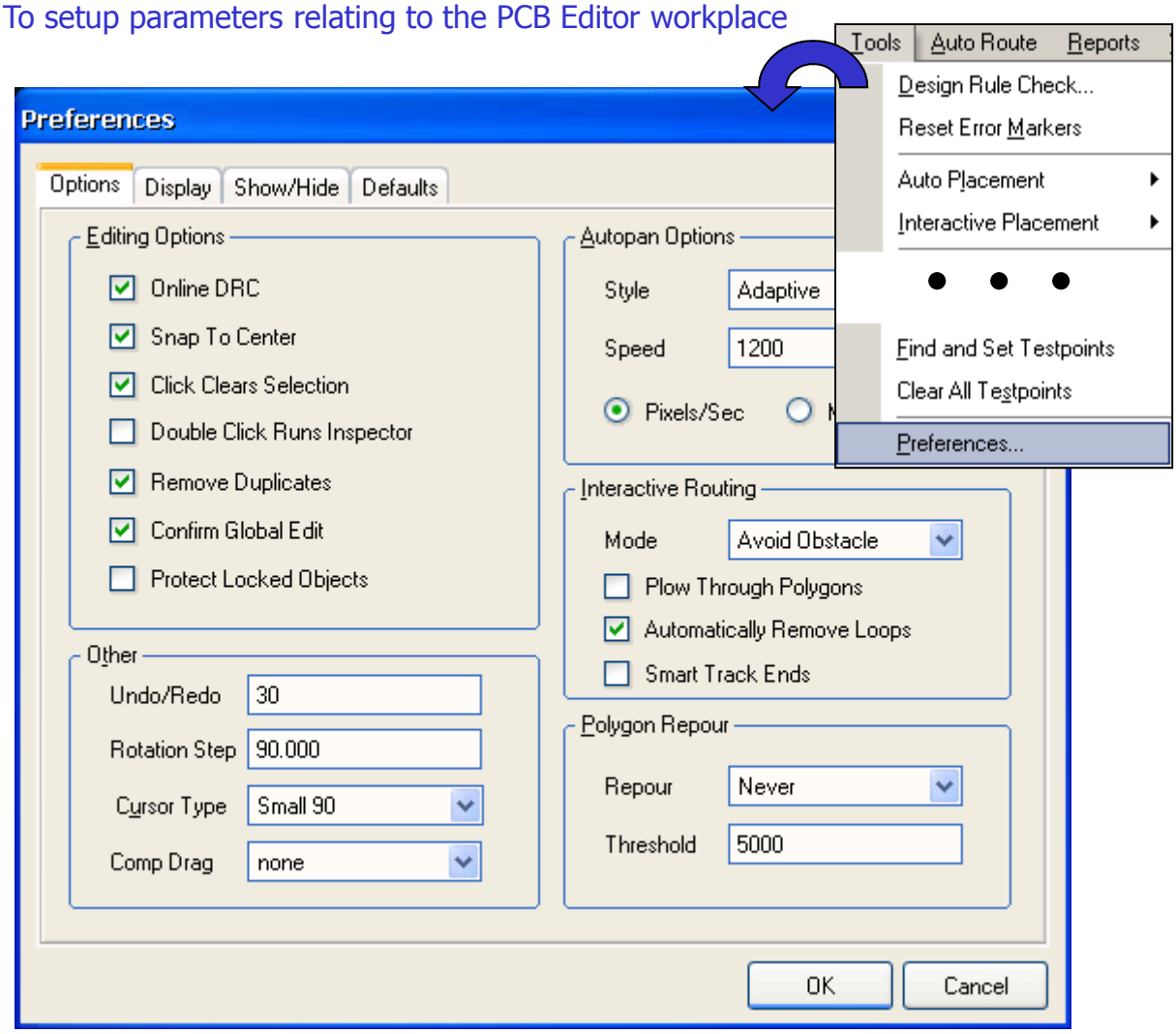


Magnify a area for viewing

[View]/[Area] - VA

Preference Dialog

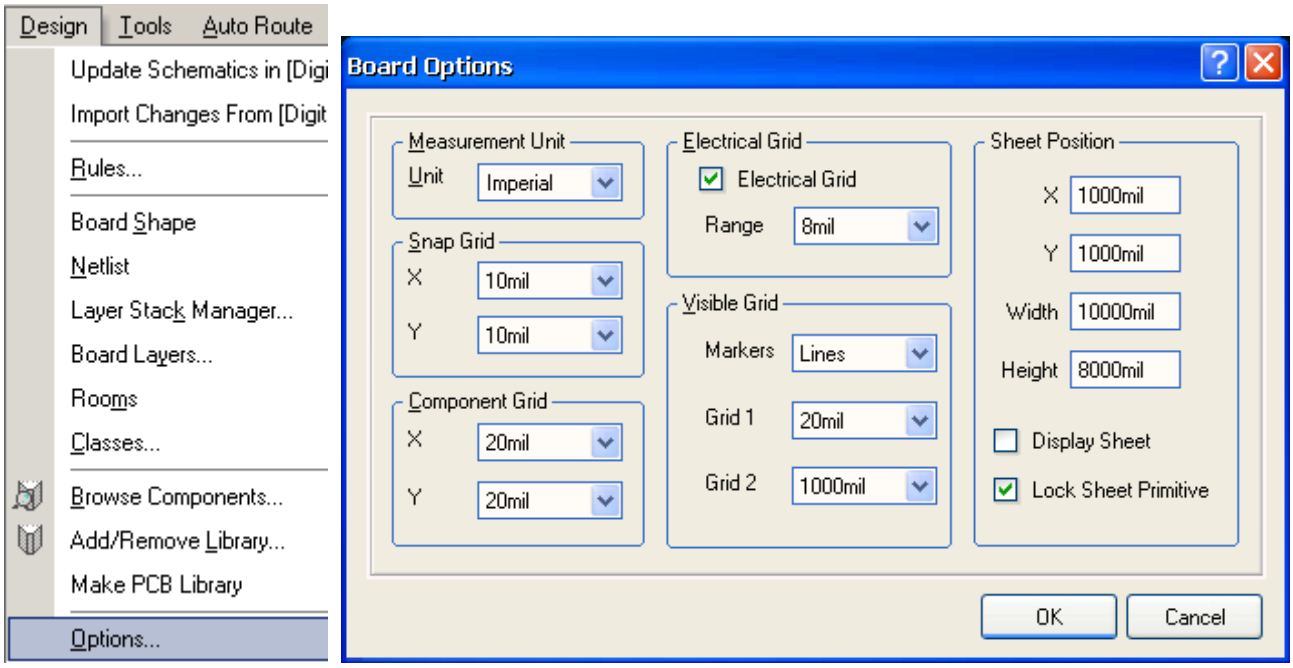
To setup parameters relating to the PCB Editor workplace



Prepare by : HK Sim simhkeng@gmail.com

Design Setting

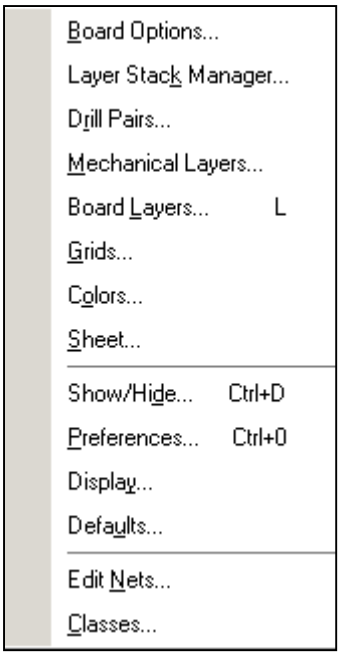
[Design] / [Option]



Short cut key for setup option

Press a "O" will pop-up a manual

O →

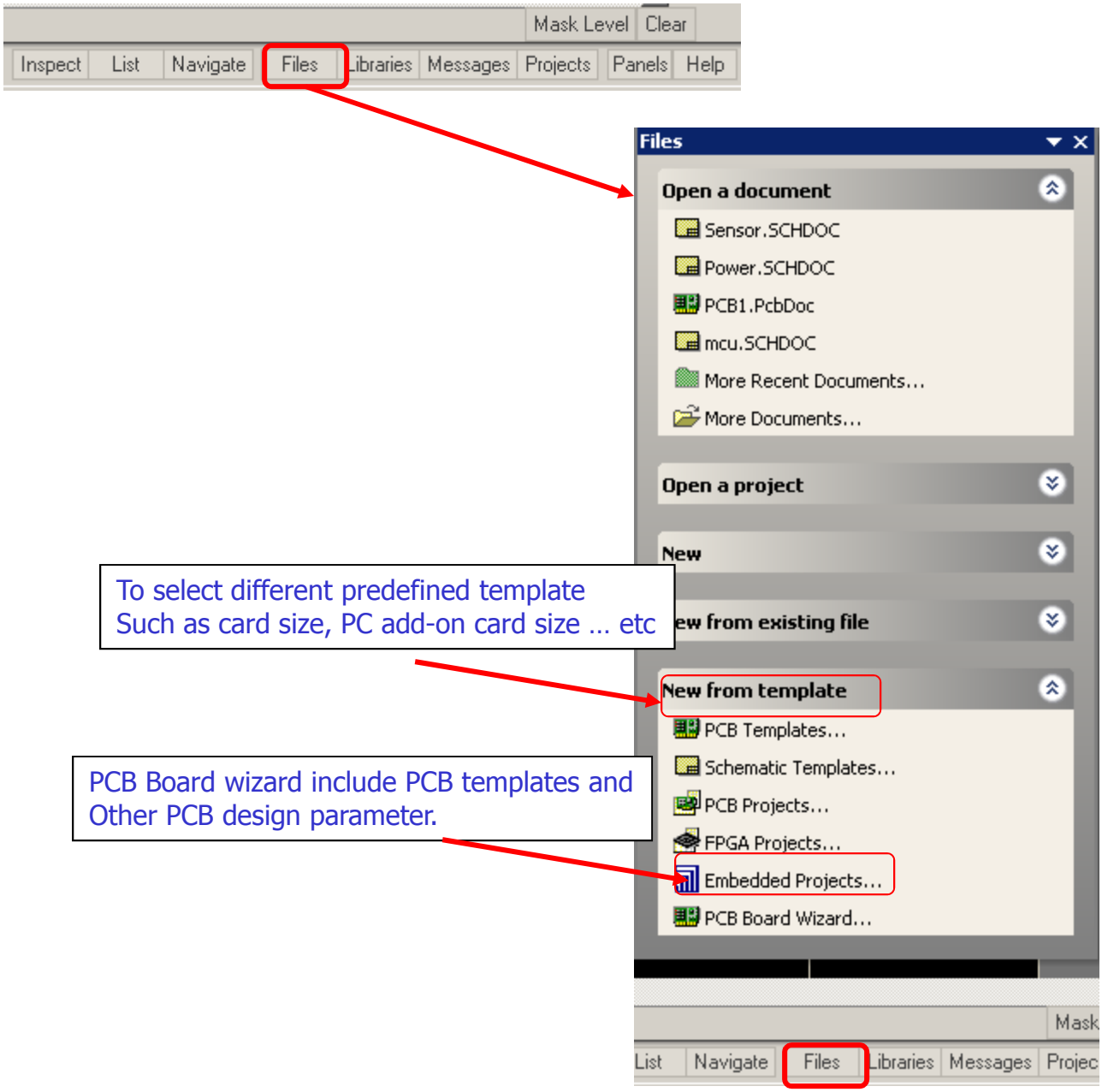


Prepare by : HK Sim simhkeng@gmail.com

Using PCB Template / Wizard to define a new PCB

Special FILE menu

The normal tool bar File menu is slightly different from the File menu from below



To select different predefined template
Such as card size, PC add-on card size ... etc

PCB Board wizard include PCB templates and Other PCB design parameter.

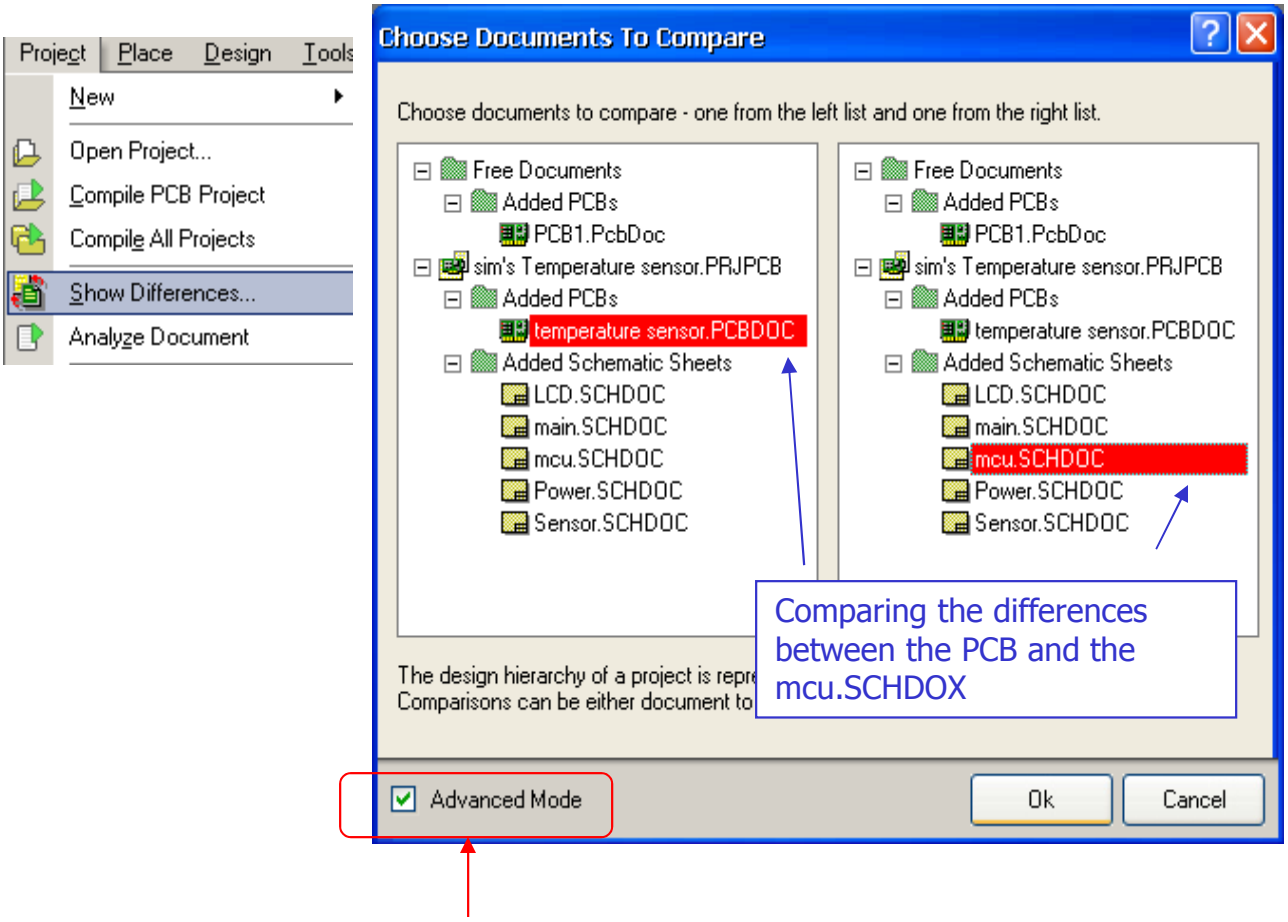
Prepare by : HK Sim simhkeng@gmail.com

Resolve synchronization Error

Two problems associate with Synchronization Error

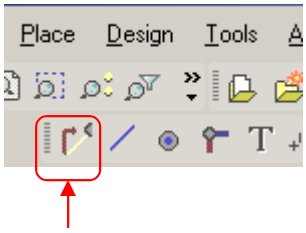
- Missing component foot print information in schematic
- New foot print does not match old foot print. Pins on schematic symbol do not match the pads on the foot print

Show difference



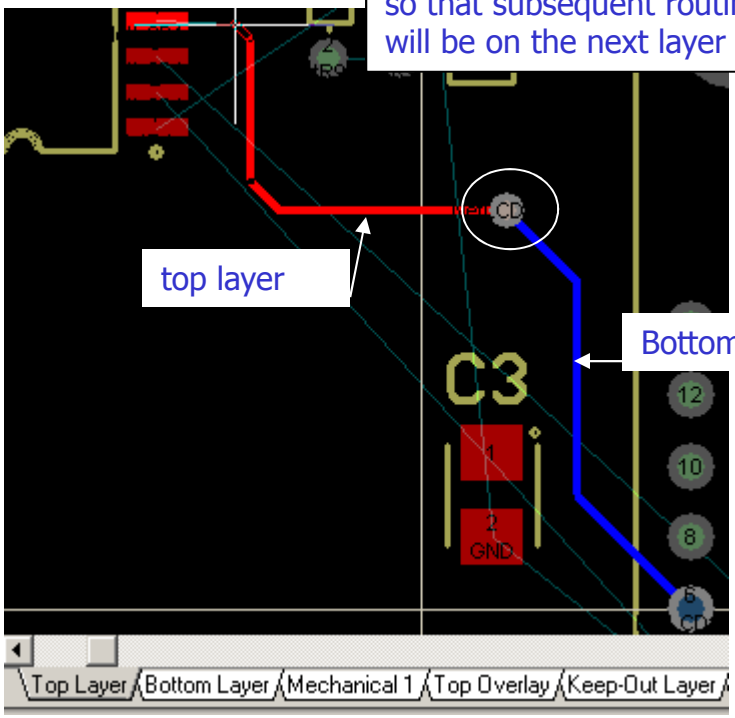
Prepare by : HK Sim simhkeng@gmail.com

PCB Trace layout



Trace icon

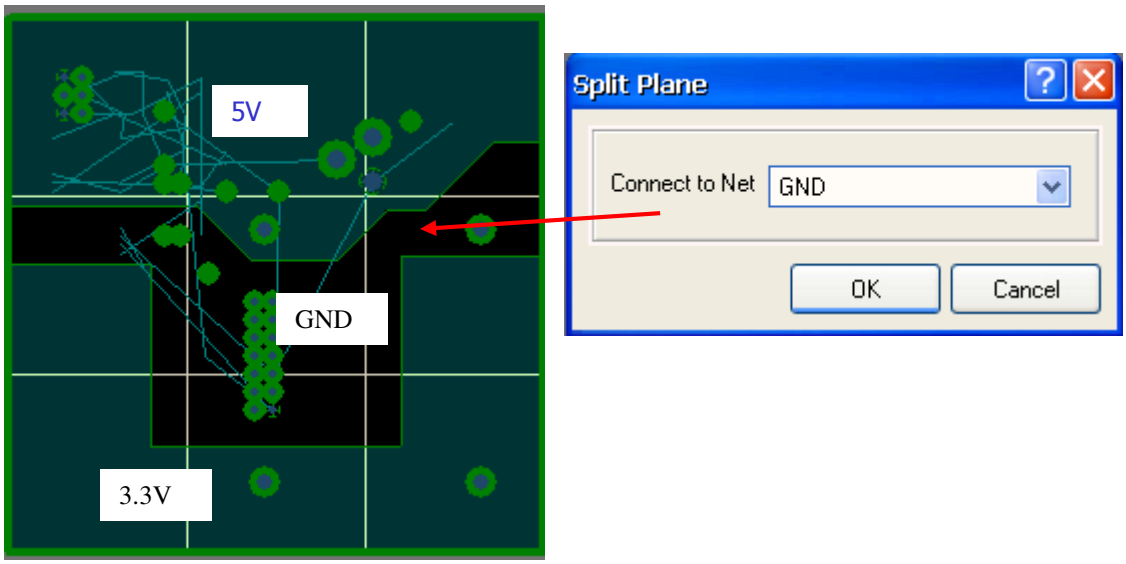
At this point, use + button and a via automatic add in so that subsequent routing will be on the next layer



Prepare by : HK Sim simhkeng@gmail.com

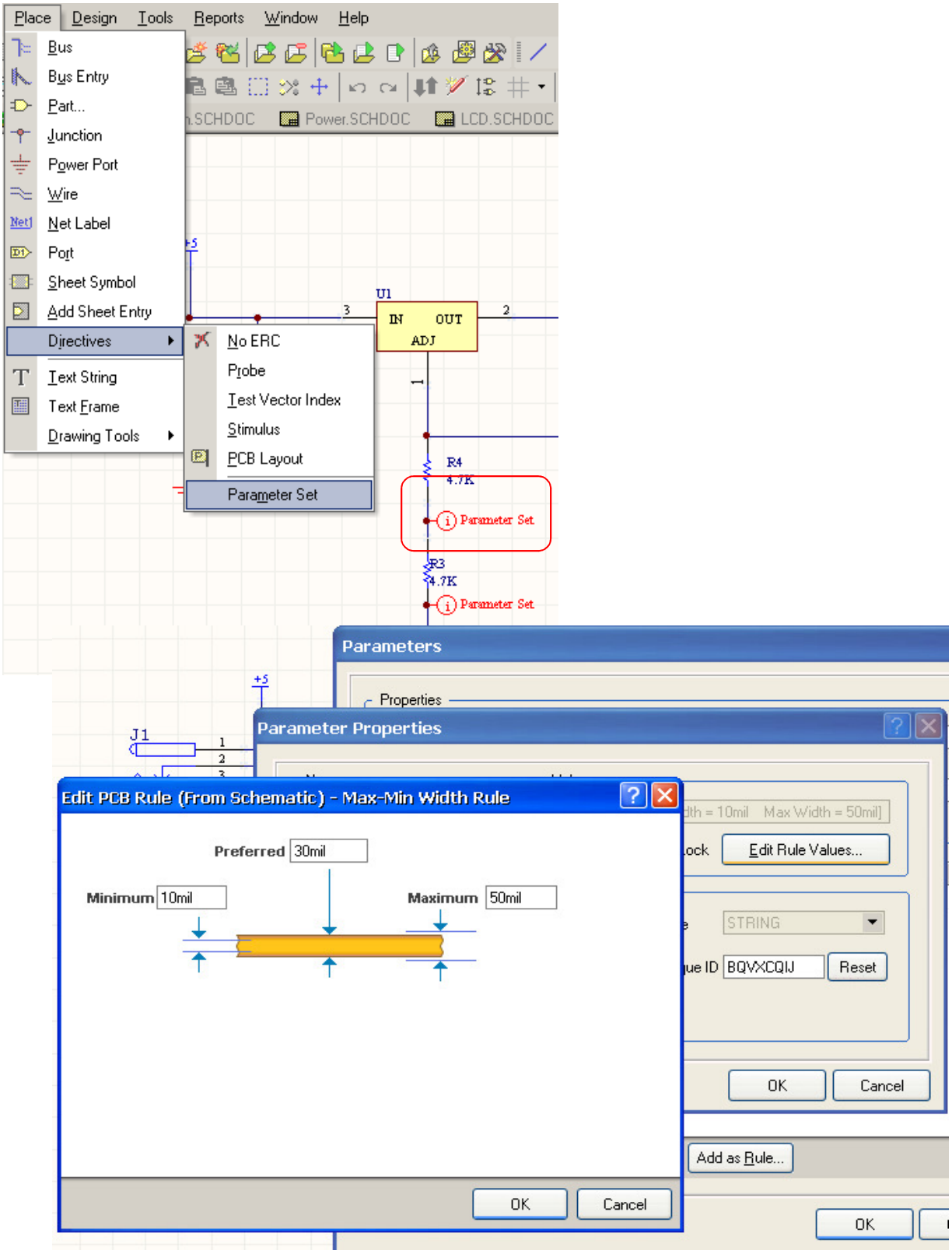
Split Power Plane

Internal planes can be split and shared amongst multiple nets.
Use line [Place] / [Line] to separate the plane into multiple regions
Define Net name connect to each plane



Prepare by : HK Sim simhkeng@gmail.com

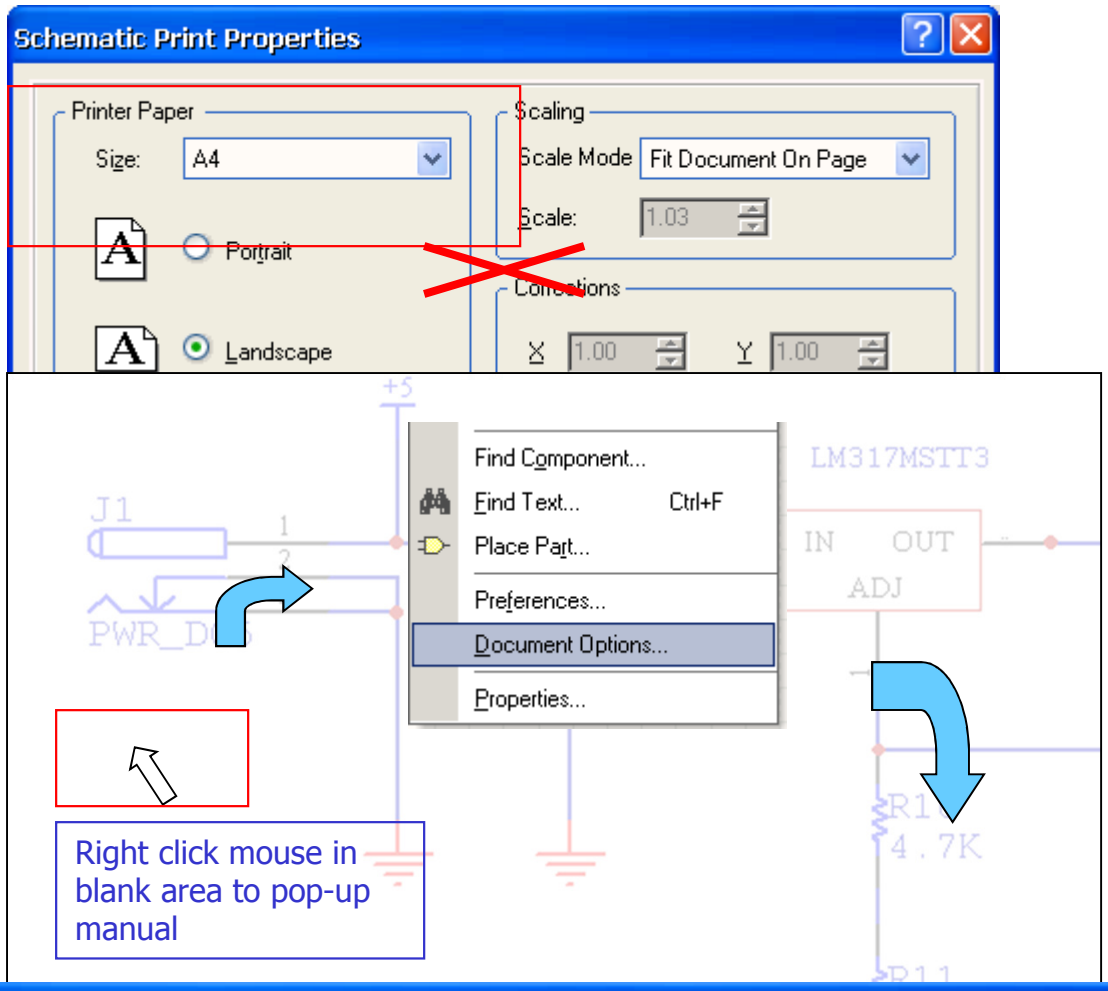
Specify a particular routing width on schematic

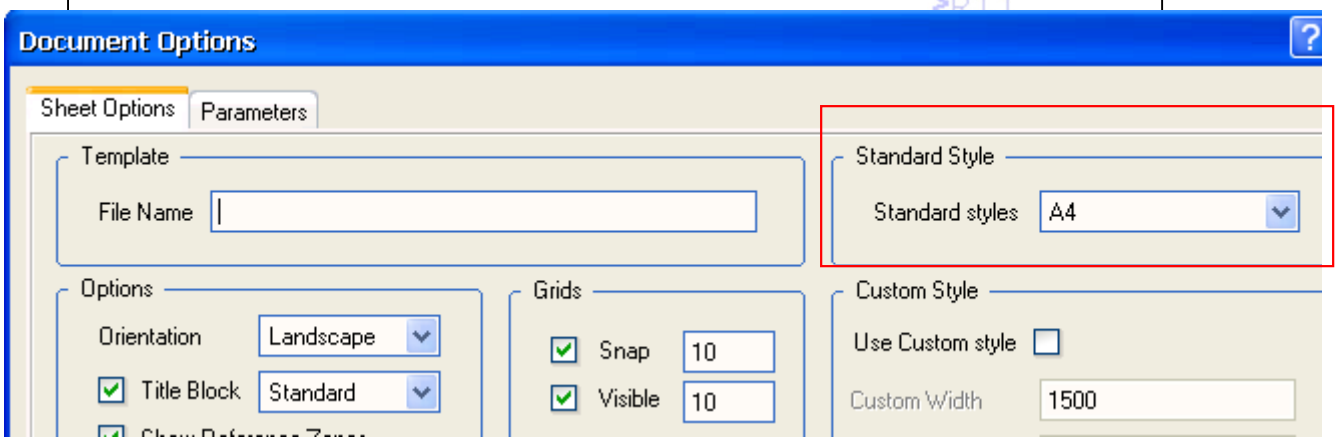


Prepare by : HK Sim simhkeng@gmail.com

Print out a viewable hardcopy

In schematic sheet, your File/Page Setup may be in A4 size as shown below. But you have to make sure that your Document Setup is also in A4 size (NOT by default). Otherwise your print out may be too small





Prepare by : HK Sim simhkeng@gmail.com

PCB Short cut commands

Measure Distance
CTRL + M and then select two points to get the distance

Layer Movement
+ next layer
- Previous layer
* Next signal layer

L – display layer setup
G – set grid value
Q – Toggle units (Metric/imperial)

Rotate Component
ClickComponent +SpaceBar - rotate object anti-clockwise
ClickComponent + Shift + SpaceBar – rotate object clockwise

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