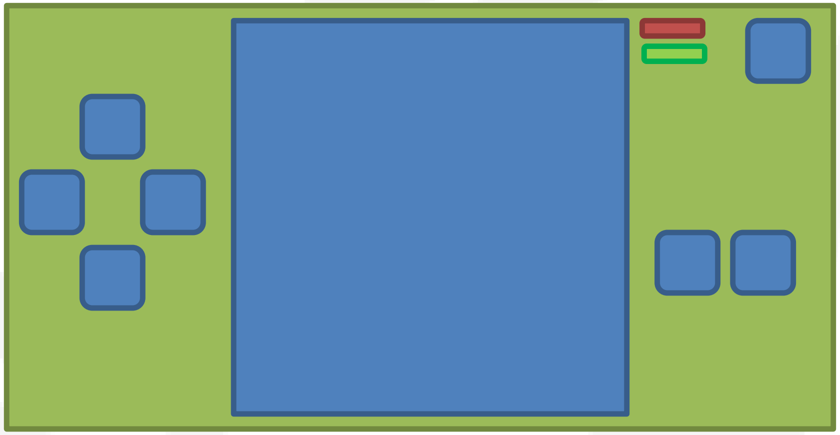
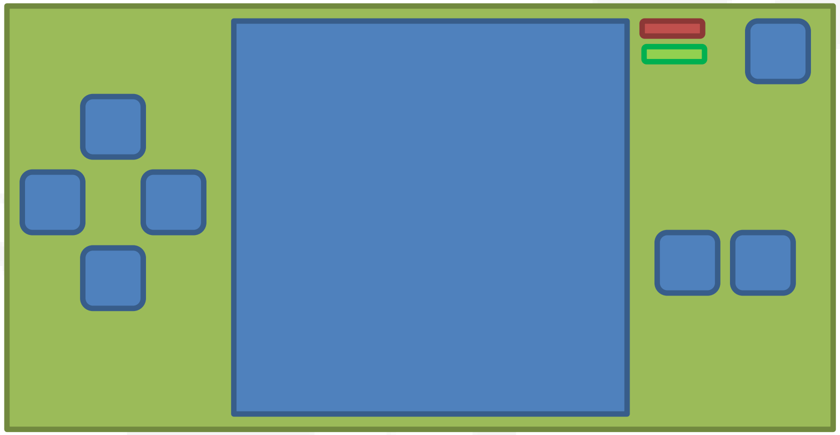
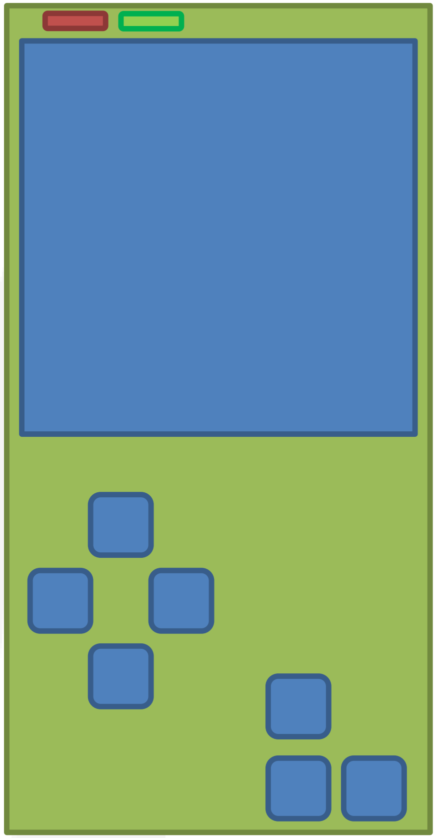
Embedded PCB Design Guide



Download
www.Sahilkhanna.org/embedded

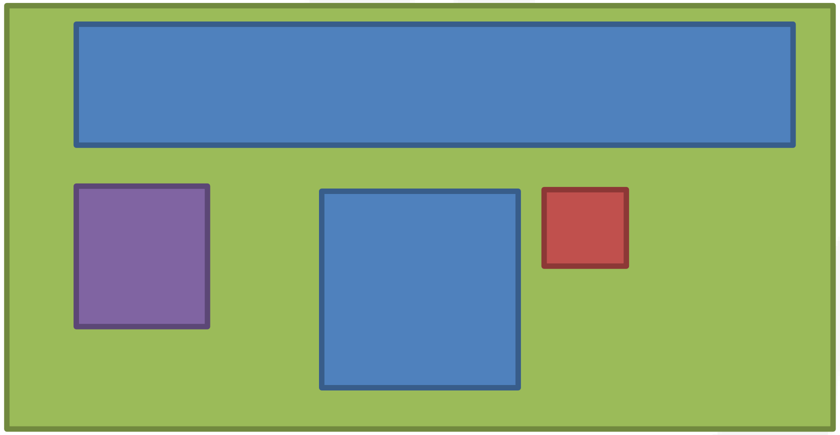
1. Basic Layout





- Choose orientation
- Place components which occupies most space like LCD and battery
- Then move on to Microcontroller

2. Placement



- Spend most of time placing components.
- Divide your design different sections such as power module, ram module etc.
- Start routing the important signals (high speed) first.

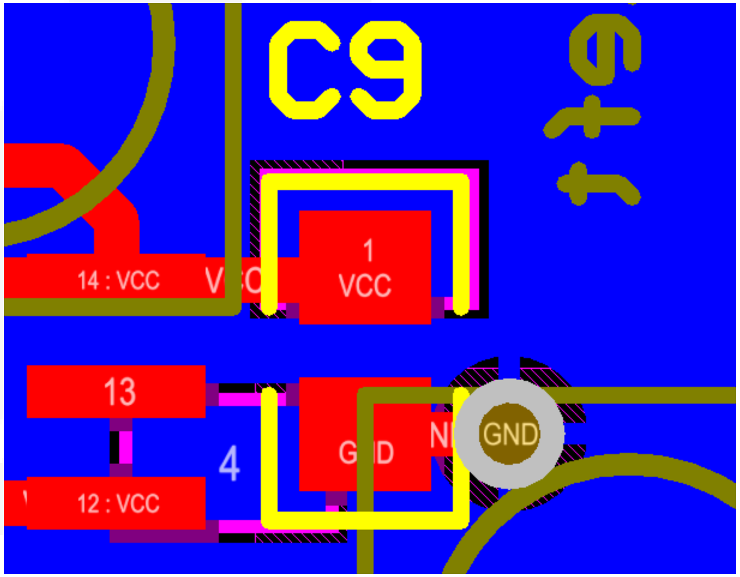
3. Prioritise

Signal description	Priority
Crystal	1
SCK, MOSI DATA	2
Other IOs	3

- Although having decoupling capacitors close to MCU is good practice, having the clock as close as possible is more important. So prioritise that over the decoupling capacitor.
- Components should be placed in such a way that clock lines(SCK) is uninterrupted i.e. no vias and should be kept as short as possible.
- General IOs can always be changed later on i.e. your button signals, so don't fixate on your initial pin assignment.
- Leave ground connection to the end as all of those could just be connected to the bottom layer GND plane by placing a via near the pins.

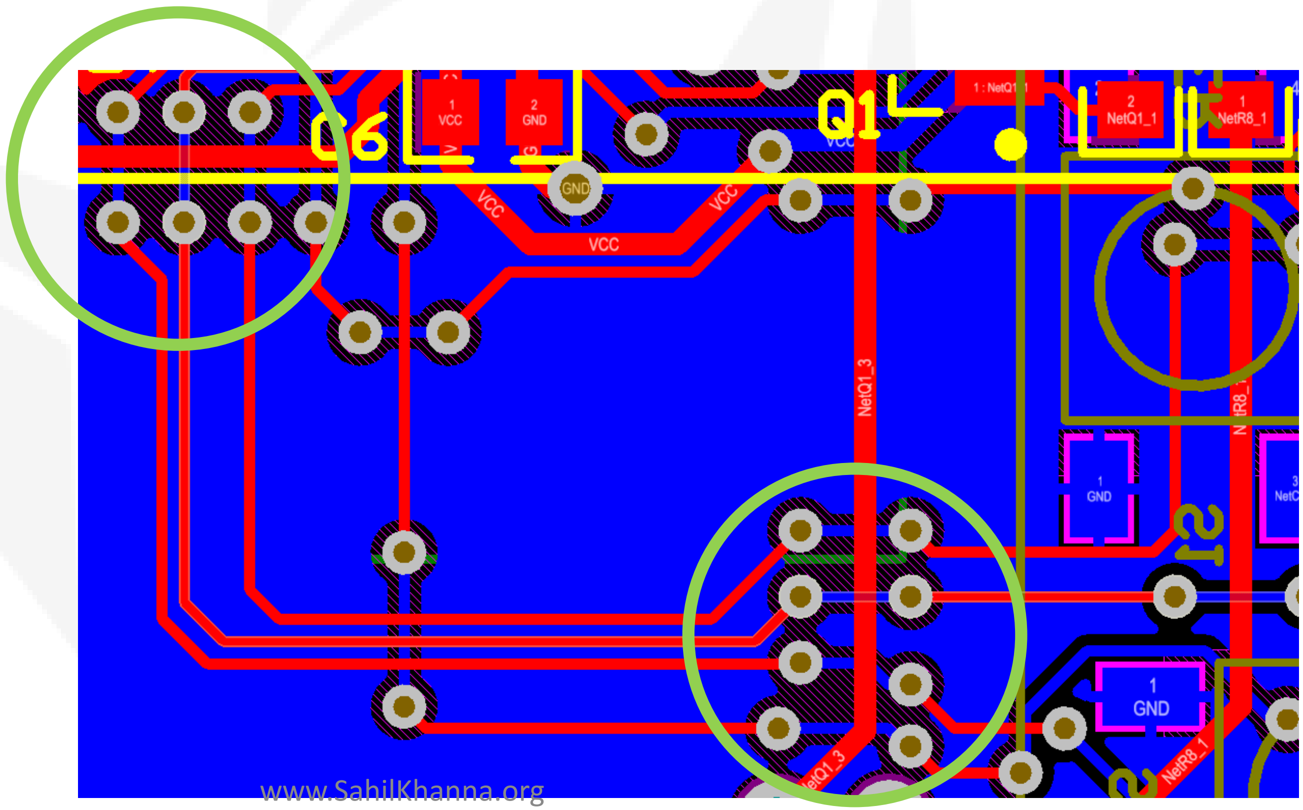
VIAs

- Acts as a bridge for tracks between different layers.
- Also reduces EMI effects
- Helps in heat sinking.
- Provides RF shielding
- Use them to connect the ground pins to the ground plane on the other side of the PCB



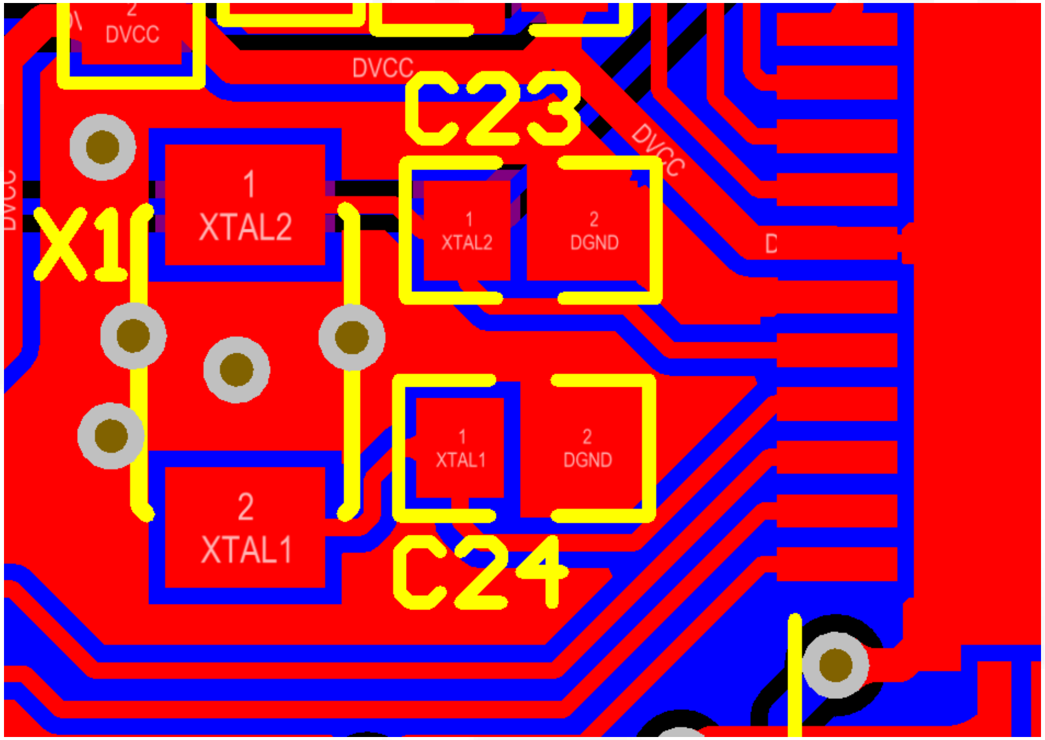
VIAs

- Form small bridges where ever there are crosslinks.
- Aim is to have least amount of breakages in the GND plane.



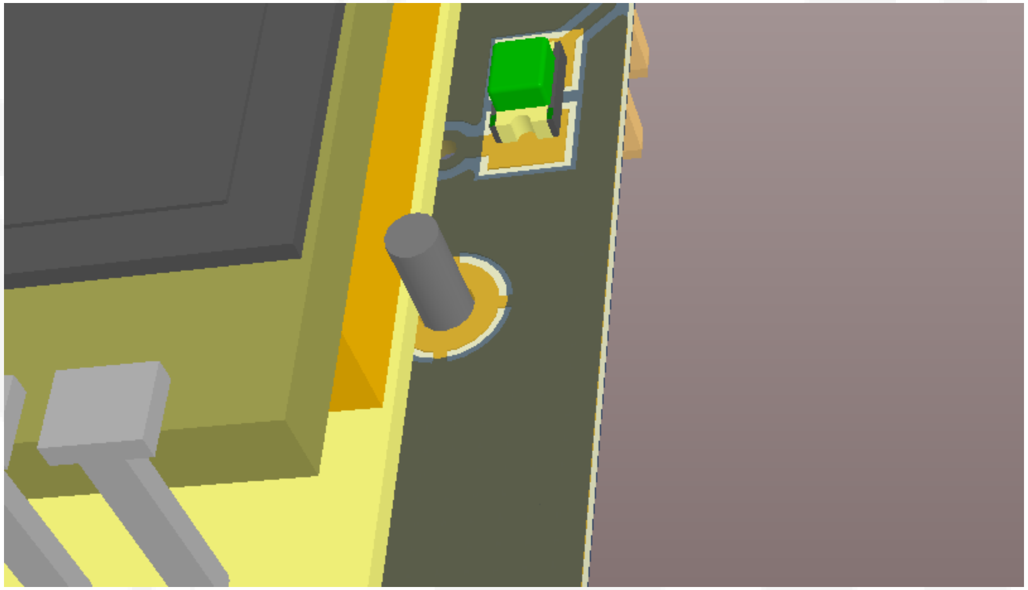
Crystal

- The most vulnerable signal on your PCB.
- Follow ATMEL guide to improve your crystal design
  http://bit.ly/2cdsdBB



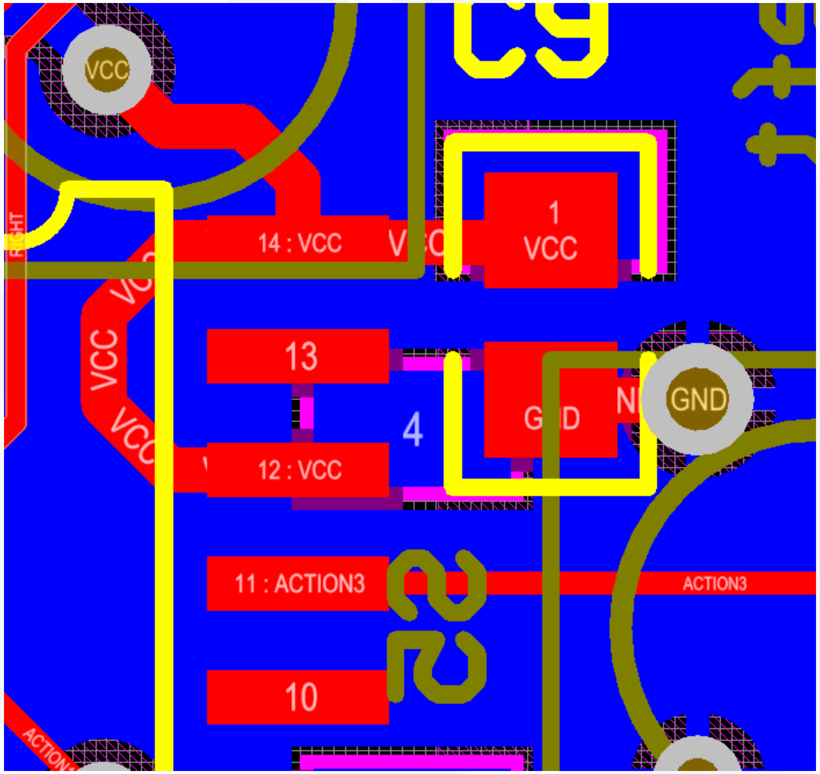
Careful of

- Through hole components.
- Battery and LCD has pins coming out on the other side.
- There will be collisions if not soldered properly.



Make sure

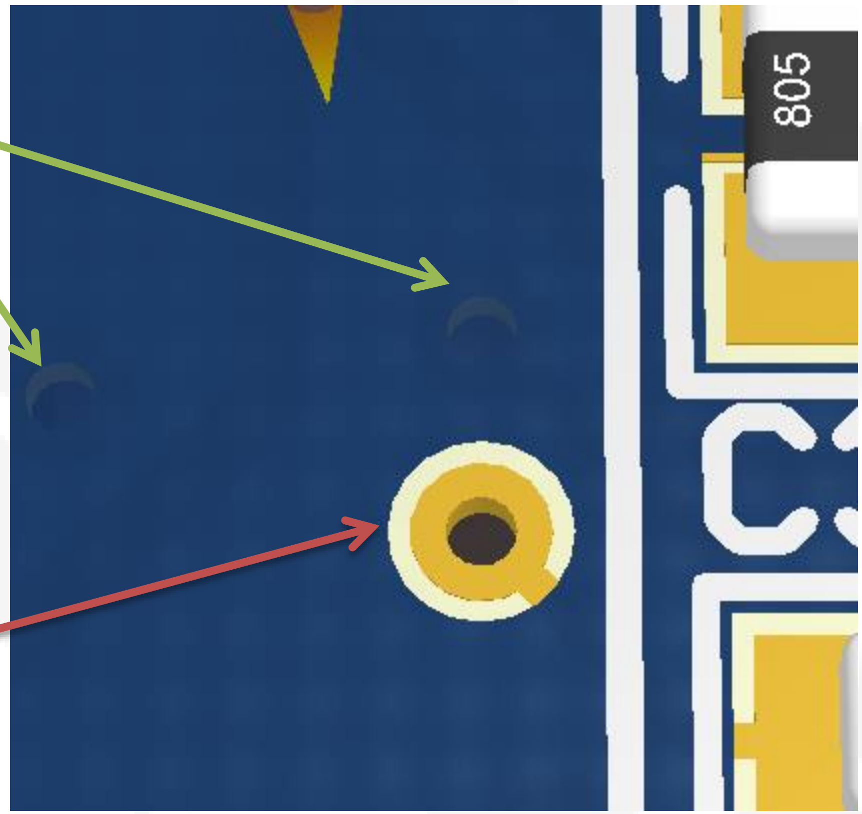
- Decoupling pins are as close as possible to the power pins of their respective ICs



Make sure

To “TENT” the vias. It forms electrical isolation on vias.

Tented Vias

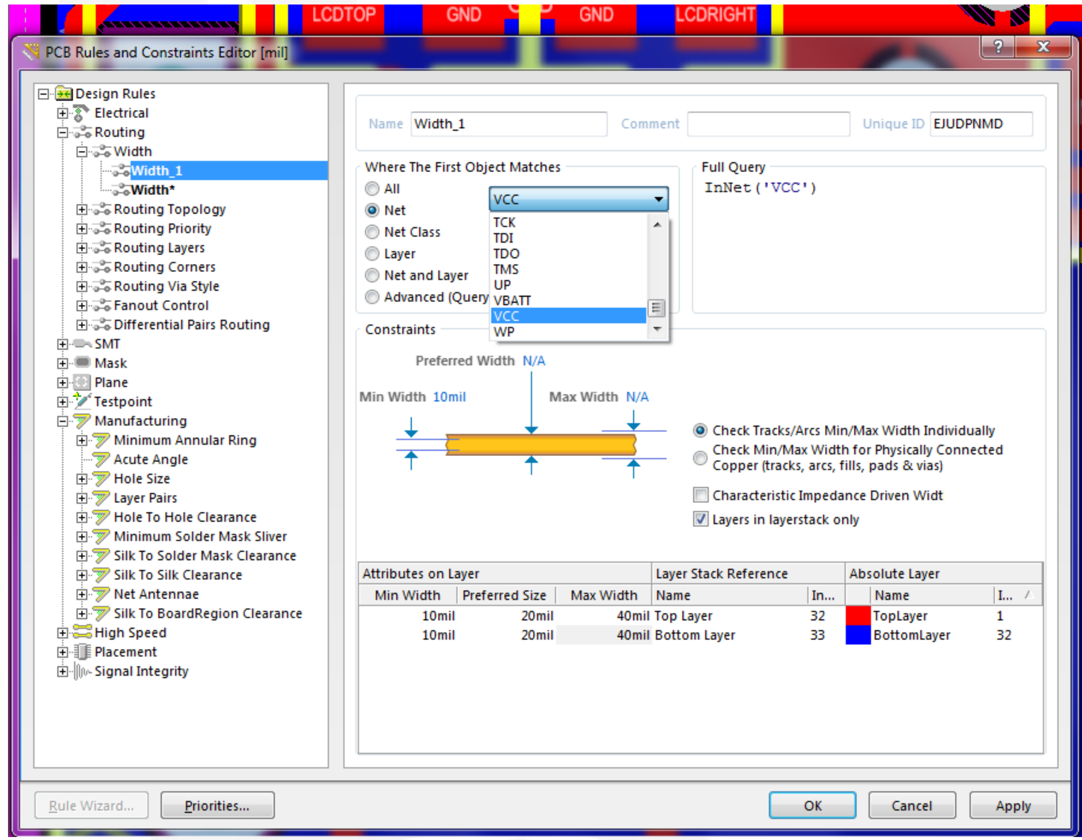


Untented Vias

Make sure

Your Power lines are thick, preferably 20 mil. You can make a rule to implement this automatically.

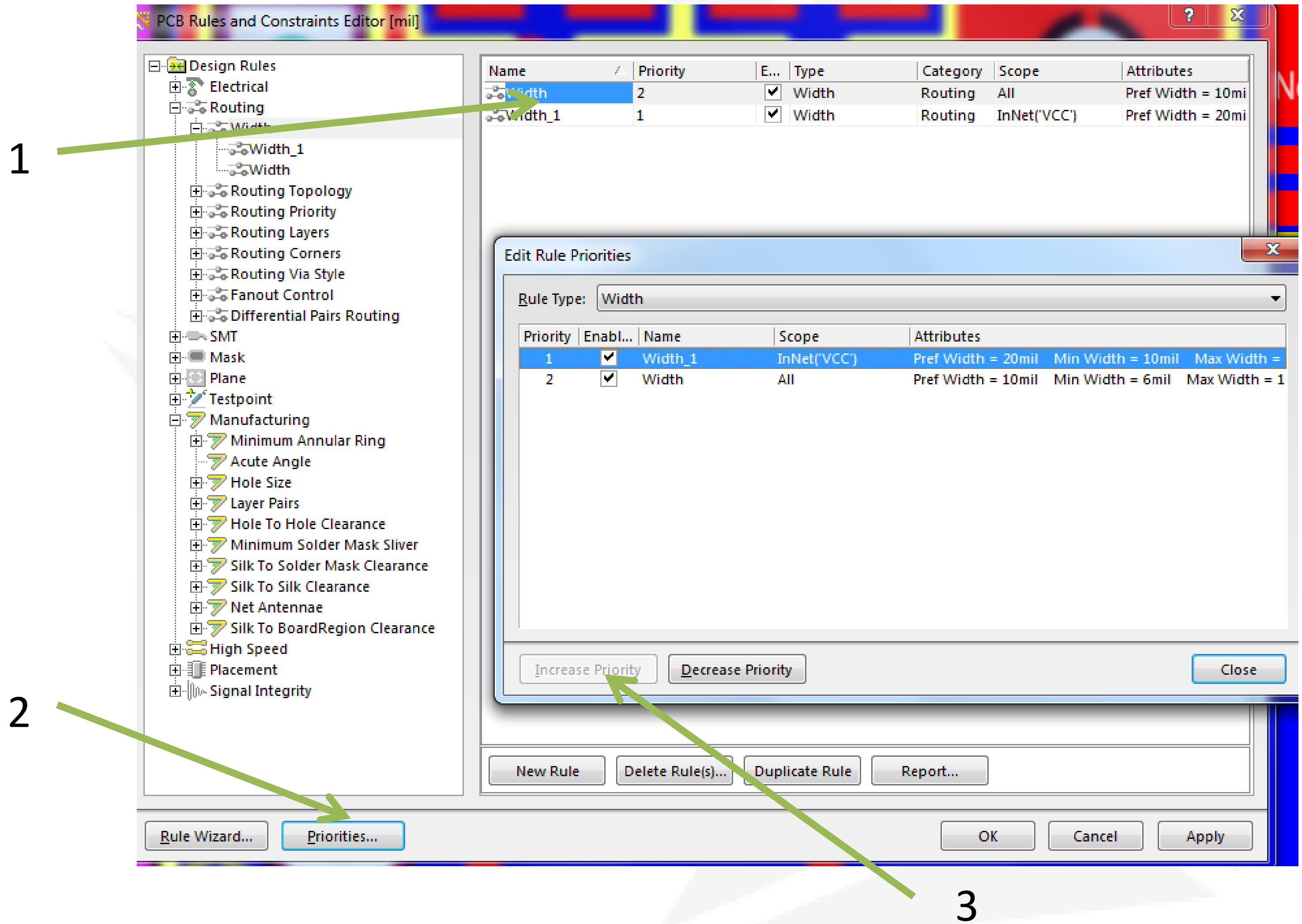
- Go to Design>Rules..
- Expand Routing and then click on width
- Double click on the new width rule
- Select the VCC or whatever net you want to change under "Net" option
- Set the preferred size to 20 mil as shown
- Click ok



Now whenever you lay VCC track, its default width would be 20 mil

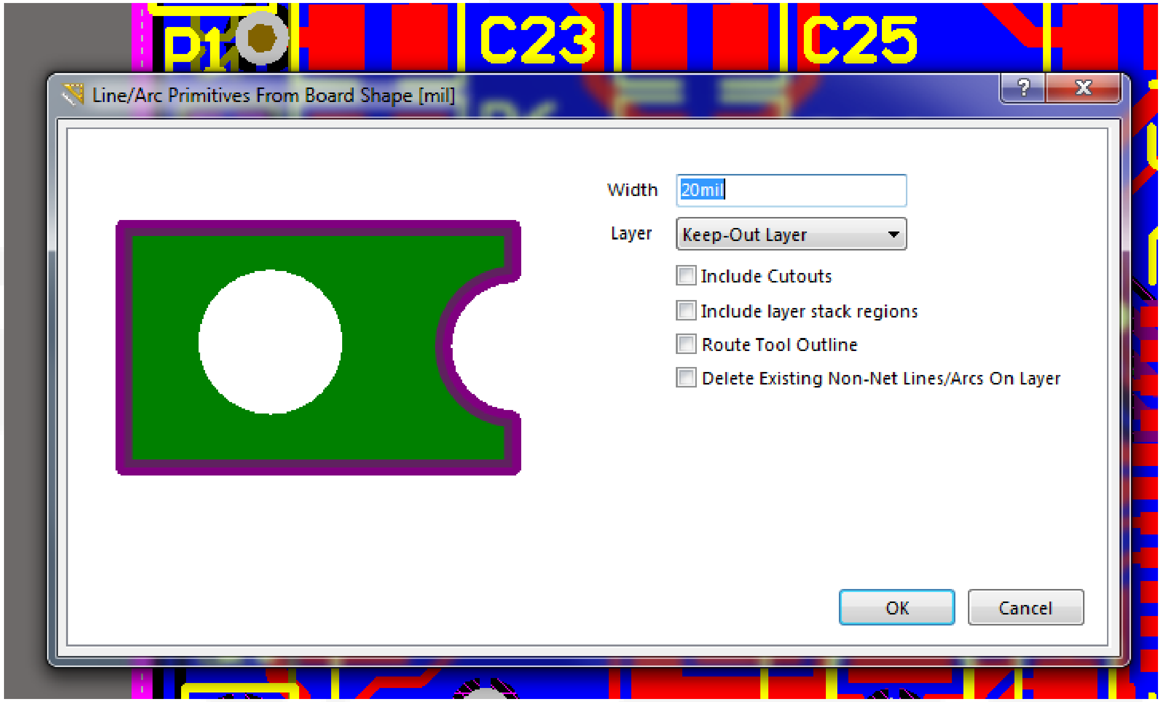
Make sure

If it didn't work then make sure the rule has the highest priority



Make sure

There is nothing in range of 10 mil closer to the edge of your PCB.
Use keep out layer to provide a guide to avoid this issue.
Go to Design> Board Shape > Create Primitives From Board Shape

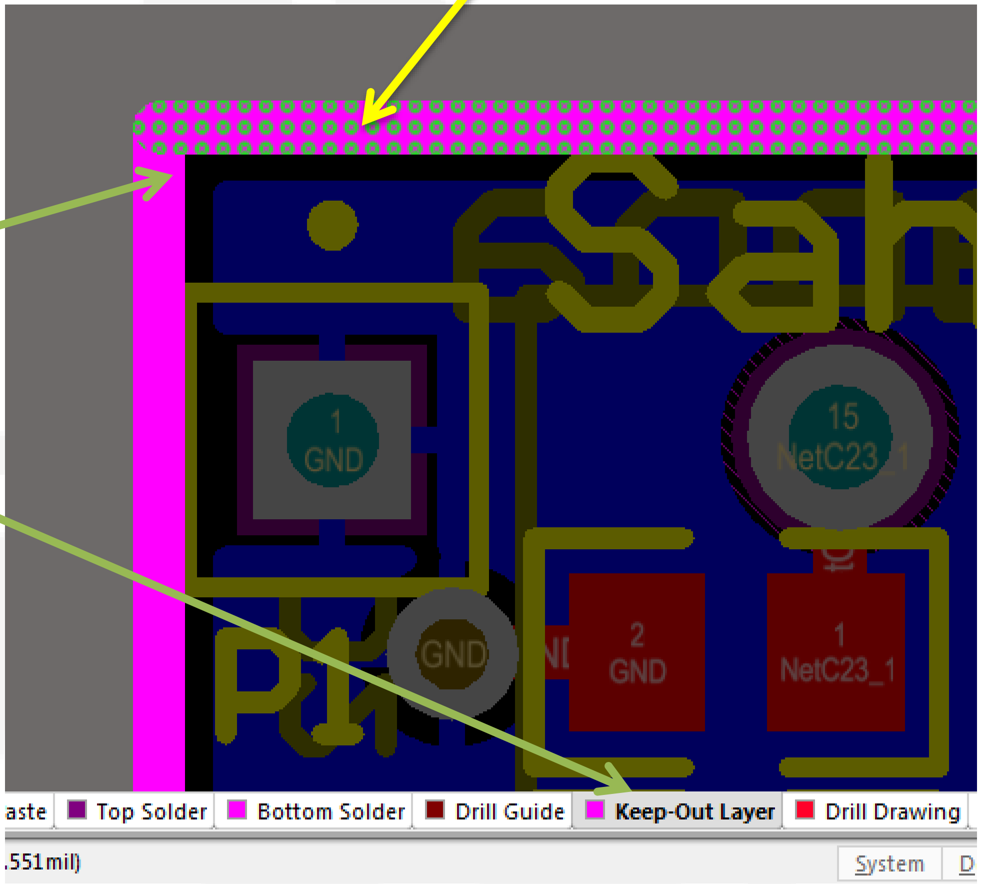


Set the value of 5 mil for the width

It will form an outline and will give an error if anything overlaps the layer

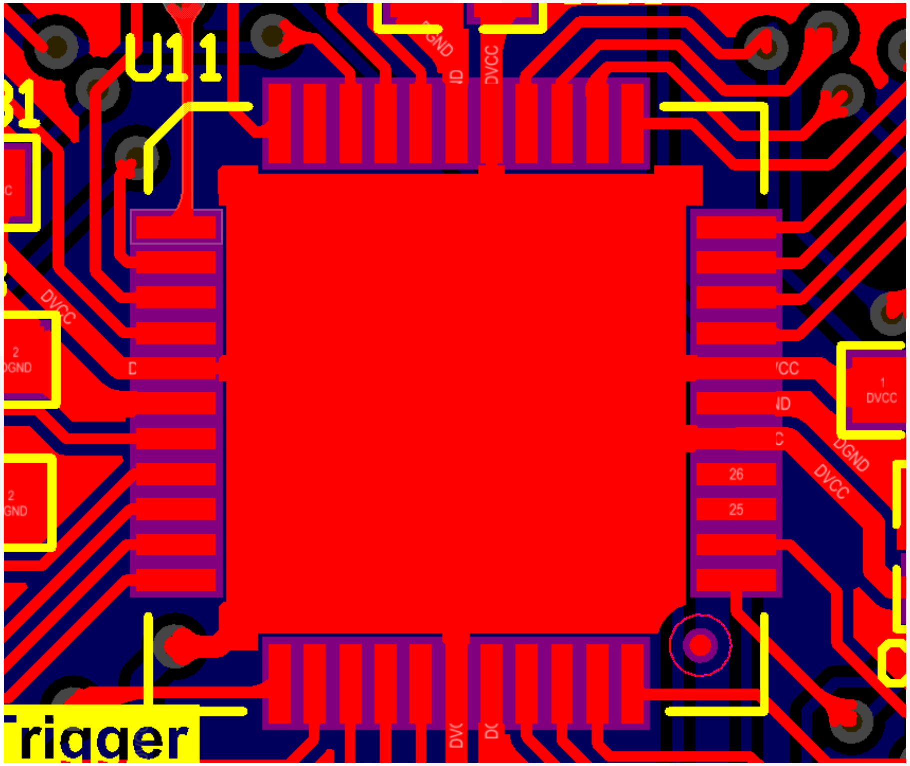
Violation

Keep-out Layer



Make sure

Good to have an uninterrupted VCC plane on the micro. For good distribution of current and in some cases can act as a good heat sink.



Shortcuts

- Shift + S
- Ctrl + click -> to mask
- Press “+” or “-” on num pad while routing to get place vias while changing the layer.
- Use PCB Inspector to change properties of anything! You don’t have to double click to change every single time.
- Frustrated with a particular net you made? Delete the whole track in one click! Tools>Un-route>NET and then select the net you want to get rid of.
- Right-click and select “Find Similar Objects” to select similar objects (-_-). Remember this is just to select similar object and not to set values. Use PCB Inspector to set values.