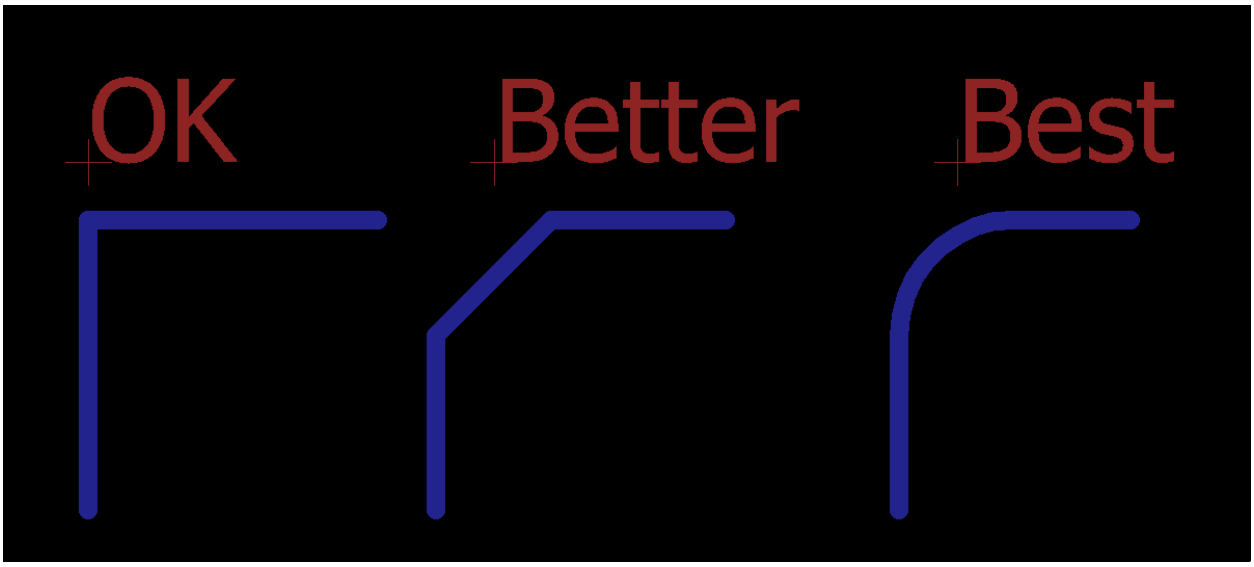
I- Important notes:

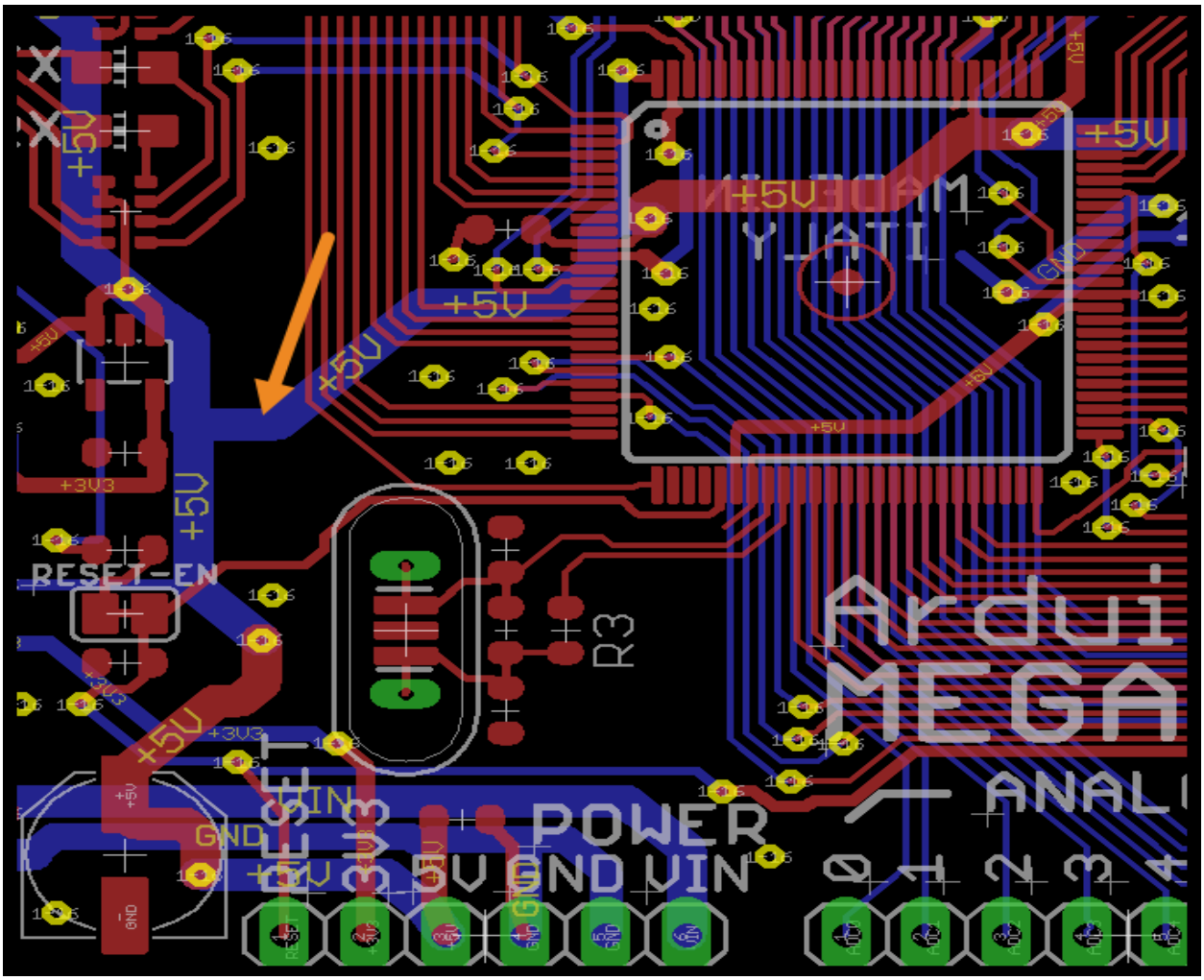
90-degree no-no. The first is always to avoid using 90-degree angle bends in your traces. Right angled traces can lead to signal reflections.



Differential pairs. You'll only get the benefits of electromagnetic field cancellation when both the signals in your differential pair have the same length and gap. This will likely require some length match tuning in your PCB design software.

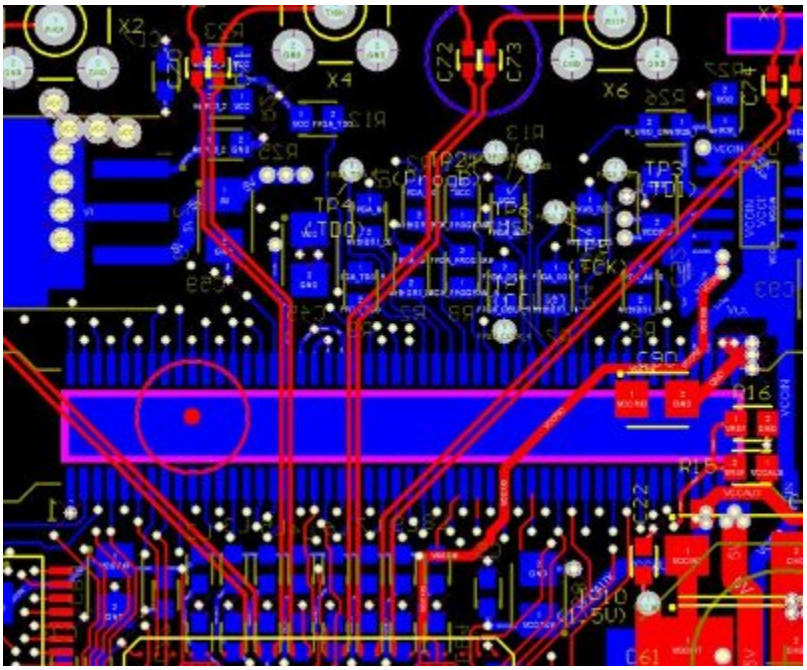
Leave Room Between Traces and Mounting Holes:

Make Your Power and Ground Traces Wider:



Route the differential signal traces in parallel:

Differential signals are often used to improve immunity to noise and amplify the dynamic range. This is only effective if the traces of both signals follow similar paths, so that the noise disturbs both paths equally. To that end, the two lines of a differential signal should be made parallel to each other and as close as possible.



Use multi-layer board:

Ground plane too small? Adding another layer can give you more options on how to handle high speed traces on your board. Differential pairs generating crosstalk? Route them in an inner layer where the noise is lowered.

Connect bypass or decoupling capacitors to ground plane:

If you have any of these in your design, you can reduce the return current path by connecting them to ground, which reduces the size of the loop, and therefore radiation. Just be sure not to connect a bypass capacitor between a power plane and an unrelated ground plane, which can cause capacitive coupling.

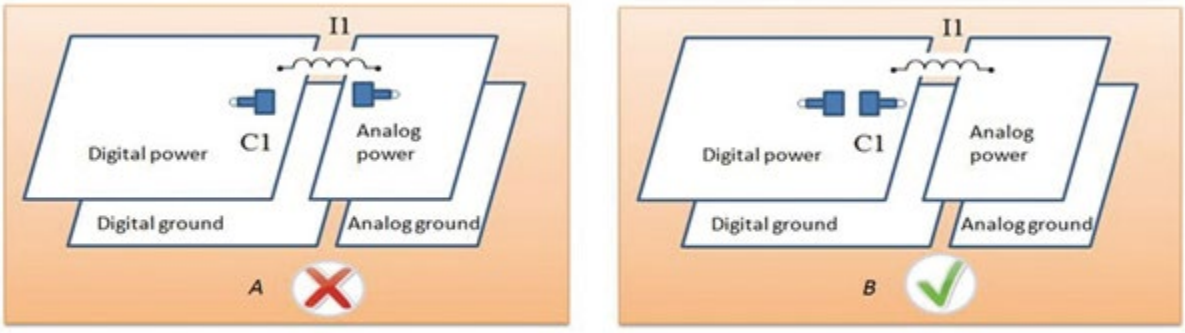
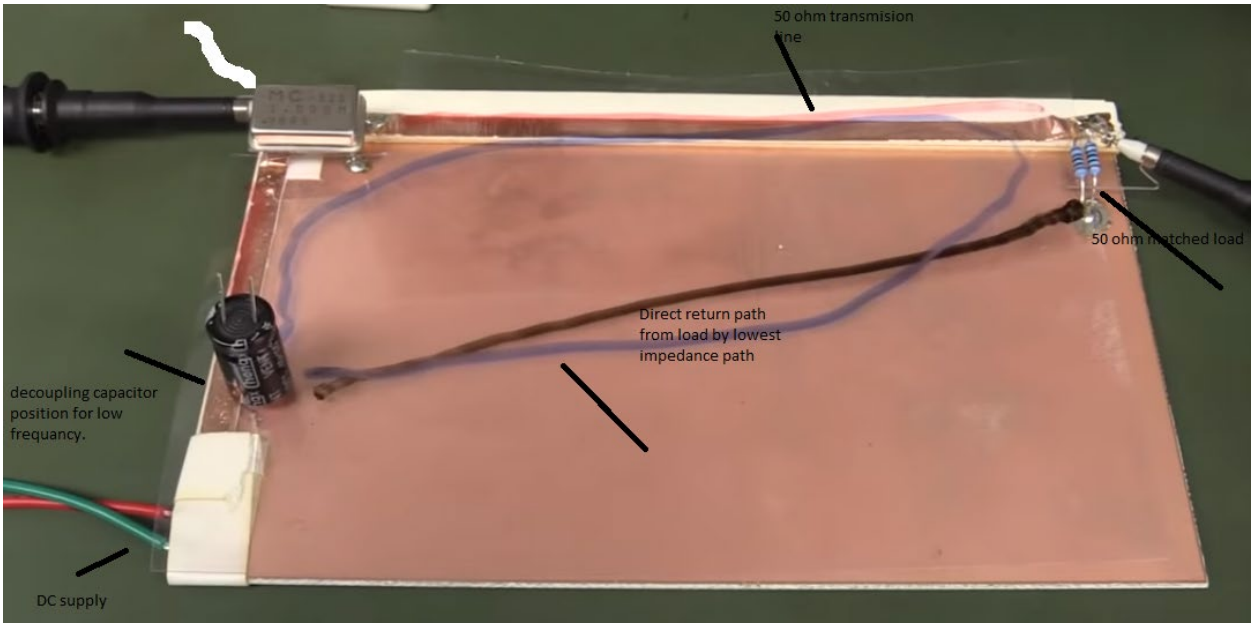
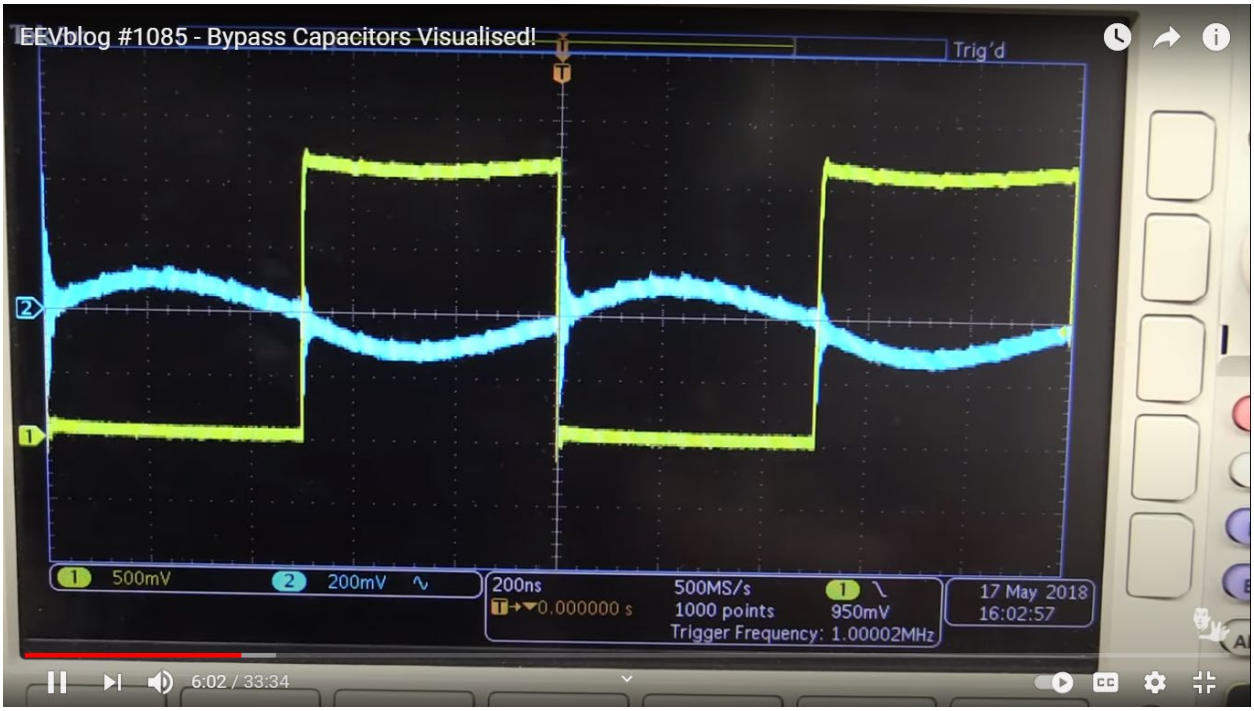
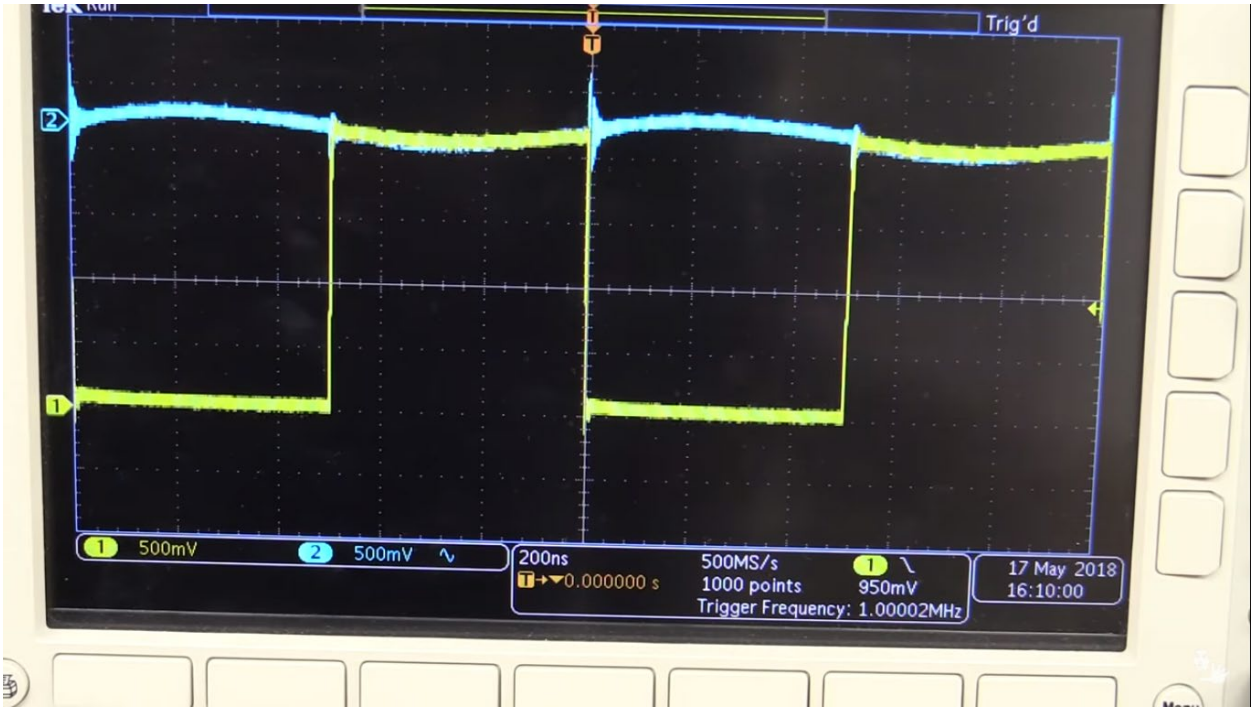


Figure 3 Capacitor Placement on Splitting Planes

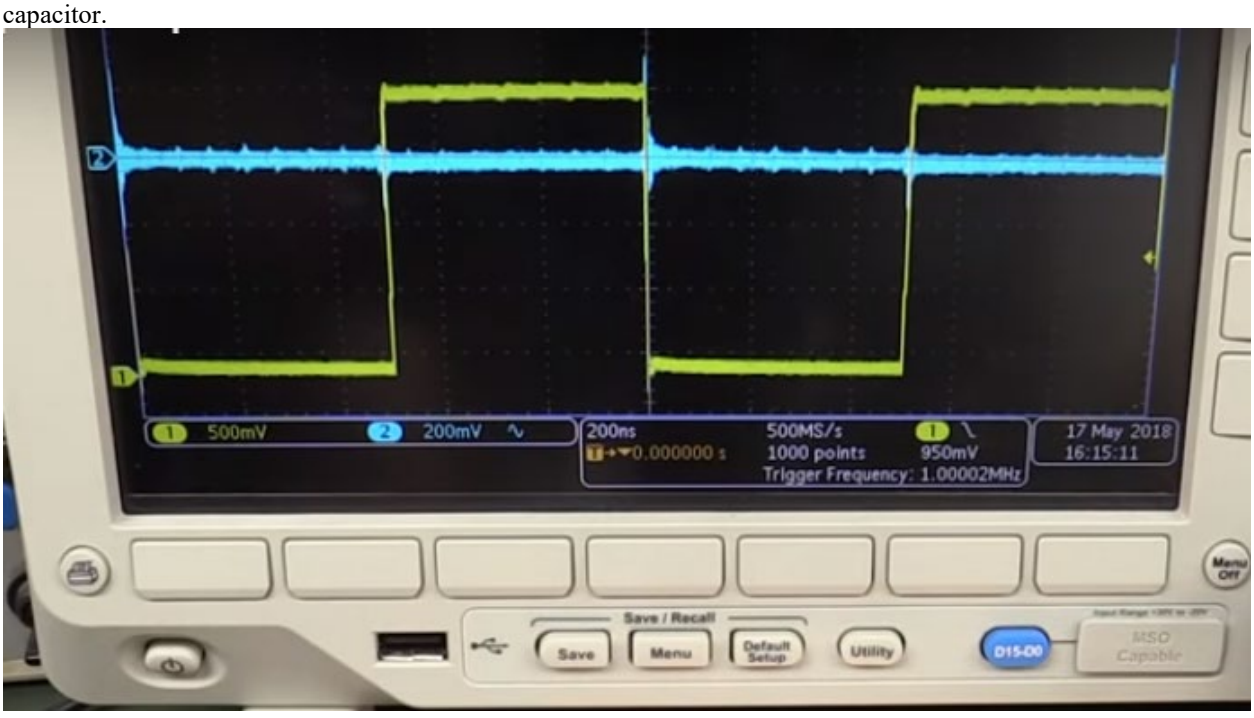


blue circle is loop area created from return current which we should reduce as much as possible to reduce EMI emission .

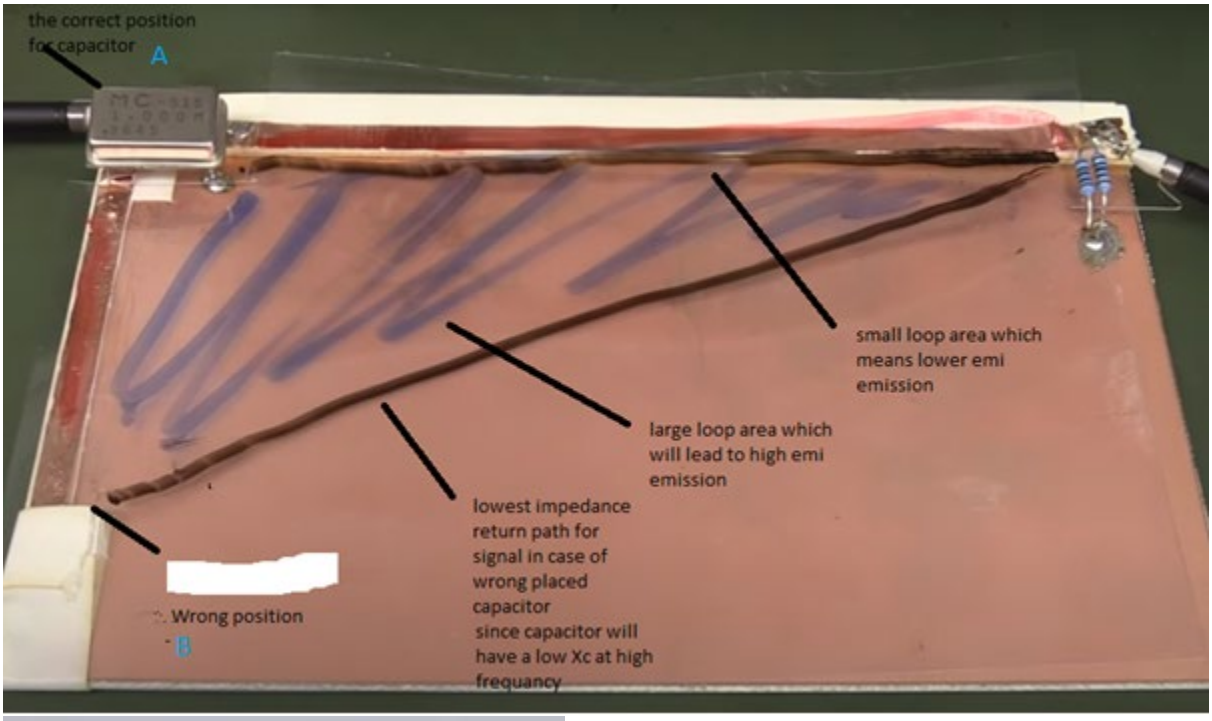


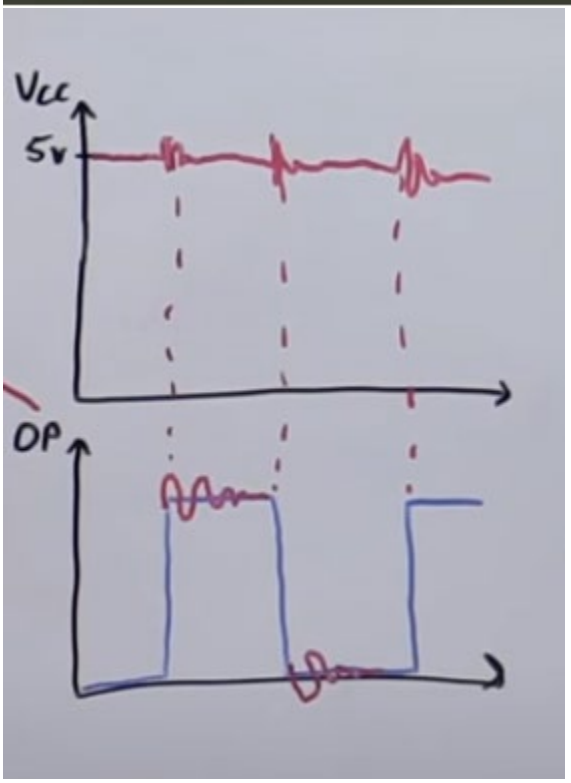


The yellow signal is at the load
And blue one is the supply input signal
So to remove low frequency ripple from input dc supply and output at load signal we use large value electrolytic capacitor.



For low frequency emission a bypass capacitor can be placed at such a distance as shown above and works properly since the ripple start from supply but with higher frequencies the situation differs the higher, we go up with frequency the more the loop area will act as an antenna and emit higher EMI as shown below and more susceptible to external noise:


the correct position for capacitor
A
small loop area which means lower emi emission
large loop area which will lead to high emi emission
lowest impedance return path for signal in case of wrong placed capacitor since capacitor will have a low Xc at high frequancy
Wrong position
B


Vcc
5v
OP

Final Outputs

Artwork File name generated by PCB Editor Designer	Associated PCB Layer
TOP.art	Top layer (copper)
BOTTOM.art	Bottom layer (copper)
Silkscreen_Top.art	Top silk screen
Soldermask_Top.art	Top solder mask
Soldermask_Bottom.art	Bottom solder mask
Solderpaste_Top.art	Top solder paste
Outline	Board outline
Project-name.drl	Drill File