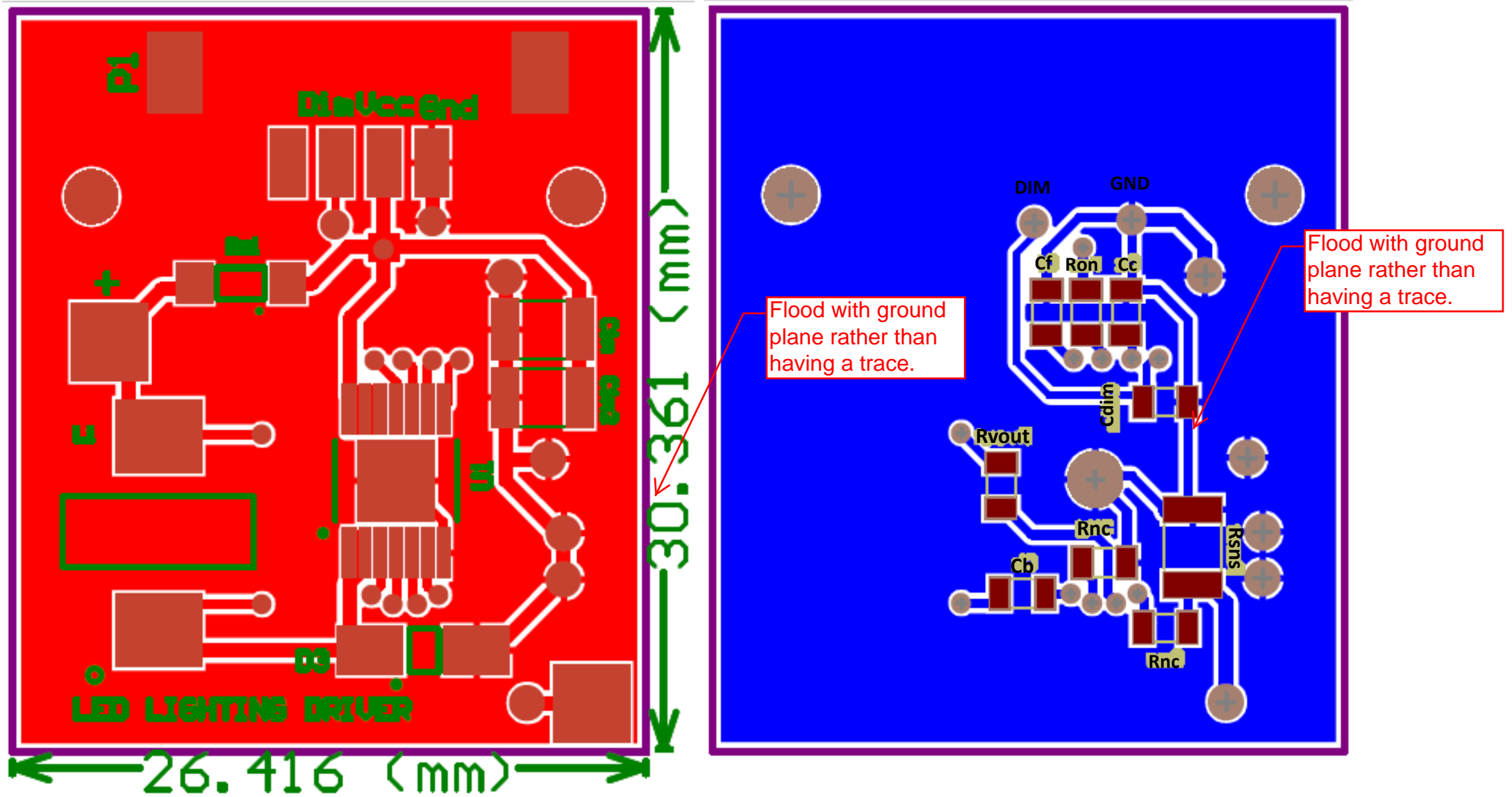
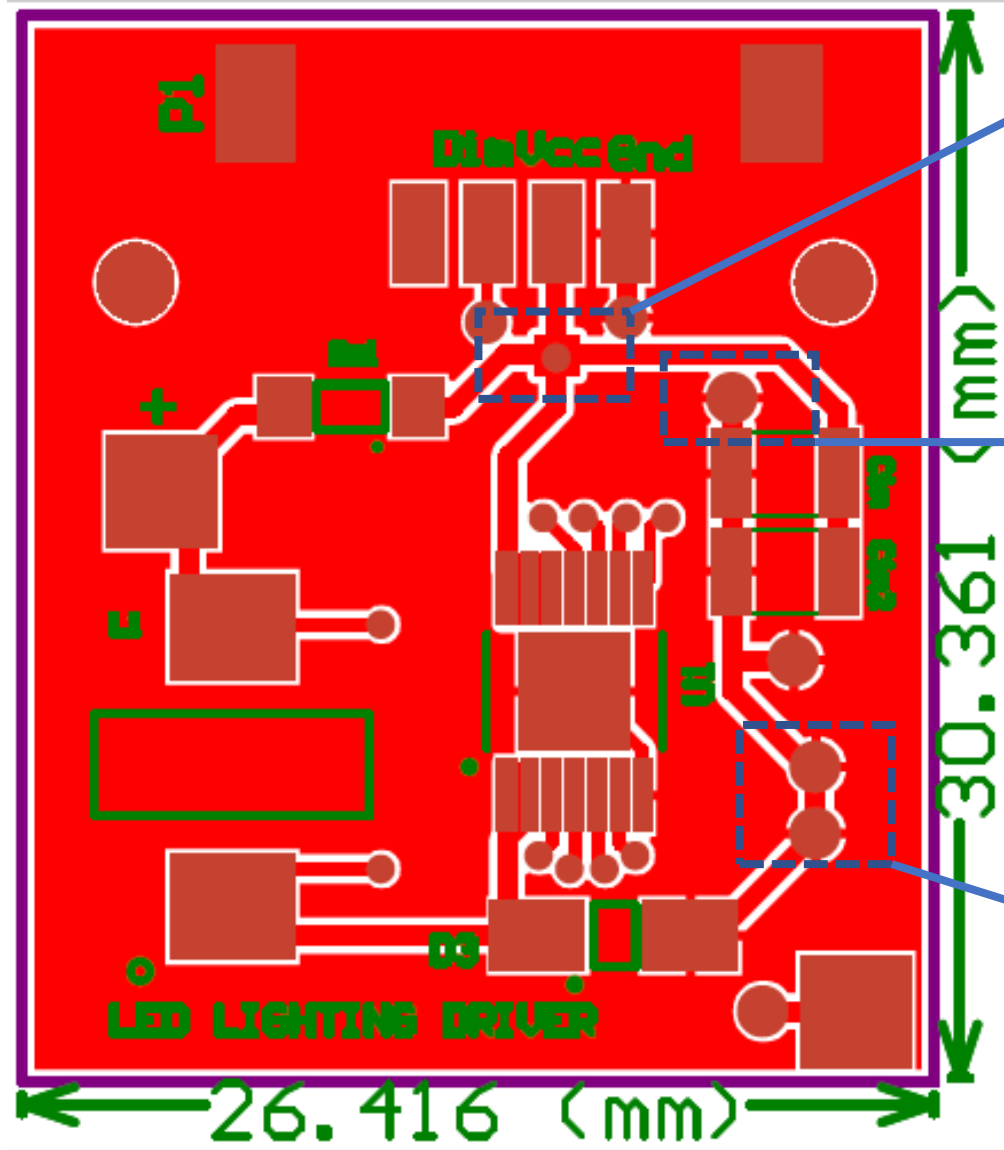


Design N° 2



Question 1



I don't know if this node is a good idea. I have read some stuff about PCB connections, but I did not find information that prohibit this. What do you think?

I don't see a big issue with this with the addition of the ground vias

I put an additional via next to the capacitor with the intention to connect through a direct wire the ground pad. What do you think of this?

That is what I was going for.

Is this path far enough from the other components? I notice that the connections of the vias from the top to the bottom layer are kind of thin (10 [mil] as maximum). Do you think a need to do the vias connections wider?

I would delete the ground paths you had initially on the top and bottom planes and just flood with ground so there is a completely unrestricted path. As it is you are still forcing a ground path in certain places. 10 mil should be ok.

Comment	Description	Designator	Footprint	LibRef	Quantity
Cap	Capacitor	Cb, Cc, Cf	0805	Cap	3
	Capacitor	Cdim	0805	Cap	1
Cap	Capacitor	Cin, Cin2	1210	Cap	2
LED2	Typical RED, GREEN, YELLOW, AMBER GaAs LED	D2	LED	LED2	1
D Schottky	Schottky Diode	D3	Diode_Schottky	D Schottky	1
Diode	Default Diode	Ds1	Diode	Diode	1
Inductor	Inductor	L1	L-Inductance	Inductor	1
Header 4	Header, 4-Pin	P1	Header2	Header 4	1
Res1	Resistor	Rcs, Rnc, Ron, Rvout	0805	Res1	4
Res1	Resistor	Rsns	1210	Res1	1
LM3406		U1	SOT14W	Component_1	1

PCB Considerations

Minimum Trace Width	10 [mil]
Maximum Trace Width	21 [mil]
Number of Layers	2
PCB Thickness	1.6 [mm]
Surface Finish PCB	HASL free
Min Tracking/Spacing	10/10 [mil]