



This thread has been locked.

If you have a related question, please click the "[Ask a related question](#)" button in the top right corner. The newly created question will be automatically linked to this question.

TPS5430: Ringing on the PH pin



TIMM Srl



Prodigy_20 points

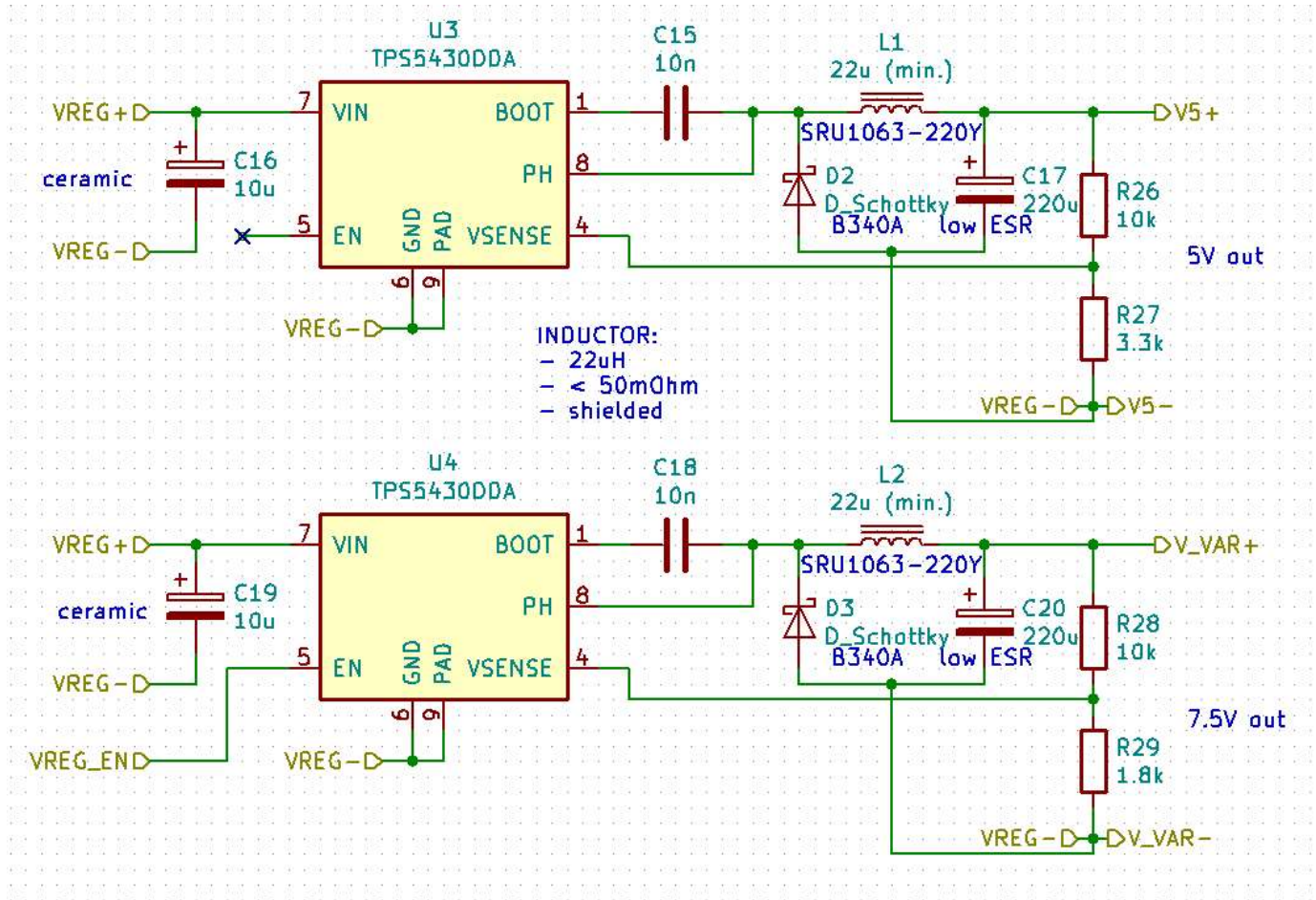
 Community Member

Part Number: [TPS5430](#)

Hello,

we are using two TPS5430 running from the same voltage source (12V), they are set to output 5V. We are experiencing lots of ringing on the PH pin waveforms which is making the regulators unusable (i.e. it's not switching at 500khz).

Here's the schematic:

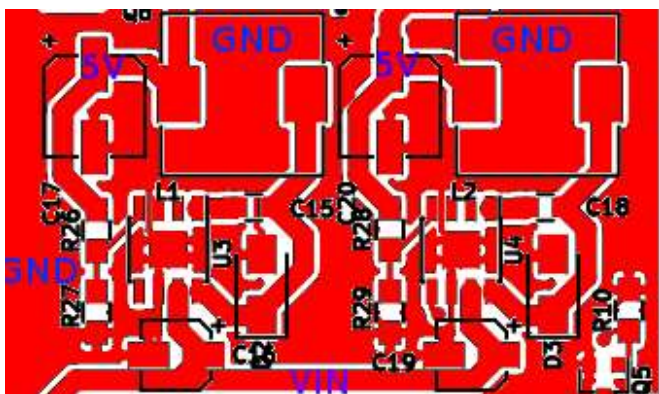


C16 and C19 are GRM31CR6YA106KA12L (CAP CER 10UF 35V X5R 1206)

C17 and C20 are TCJD227M010R0040 (Tant poly 220 μ F, 10 V, \pm 20%, D, 0.04 ohm)

L1 and L2 are SRU1063-220Y (22 μ H 3A 60 mOhm max)

Here's the PCB layout, red is the TOP copper, due to a production error there's no BOTTOM copper (e.g. GND)



This layout shows, as I said before, lots of ringing on the PH pin thus making the regulators unusable.

As we are making a second revision of the board, here's the updated layout, TOP and BOTTOM:

Frank De Stasi *over 4 years ago*



TI Mastermind 43175 points

I would try to get the ground of the input capacitor and the diode ground next to each other.

Also, a small 100nF or so, input bypass cap can sometimes be helpful. This should be close to the Vin and GND of the device.

It looks like your bottom ground plane is broken with small traces running through it; I would try to avoid this type of thing.



TIMM Srl *over 4 years ago in reply to Frank De Stasi*



Prodigy 20 points

Hello Frank,

thanks for your suggestions, I already modified the bottom ground plane to be a complete fill:



However when you mention the input cap-diode GND I followed the EVM layout:

Riccardo