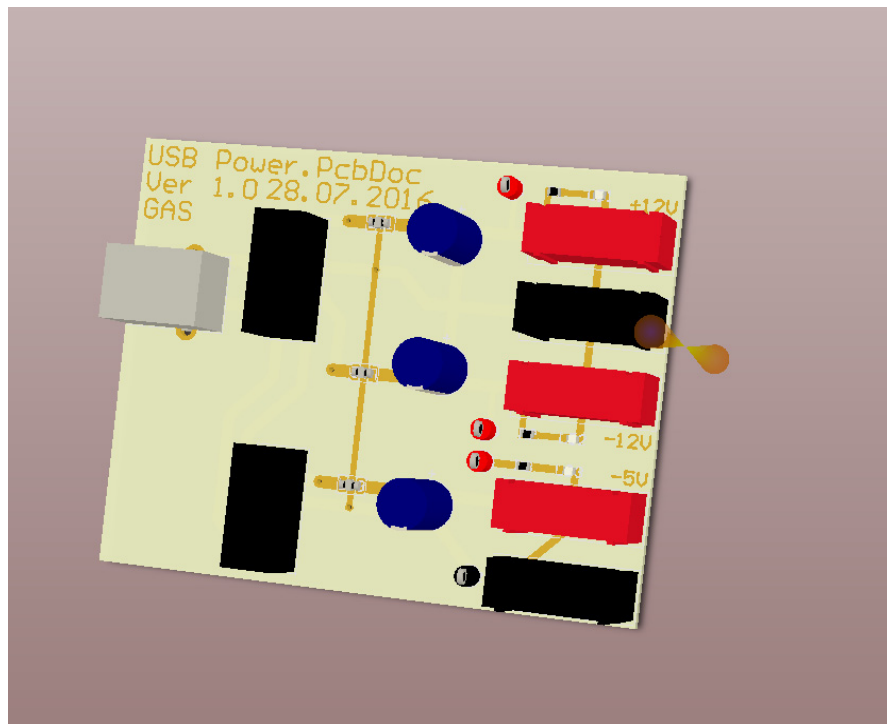


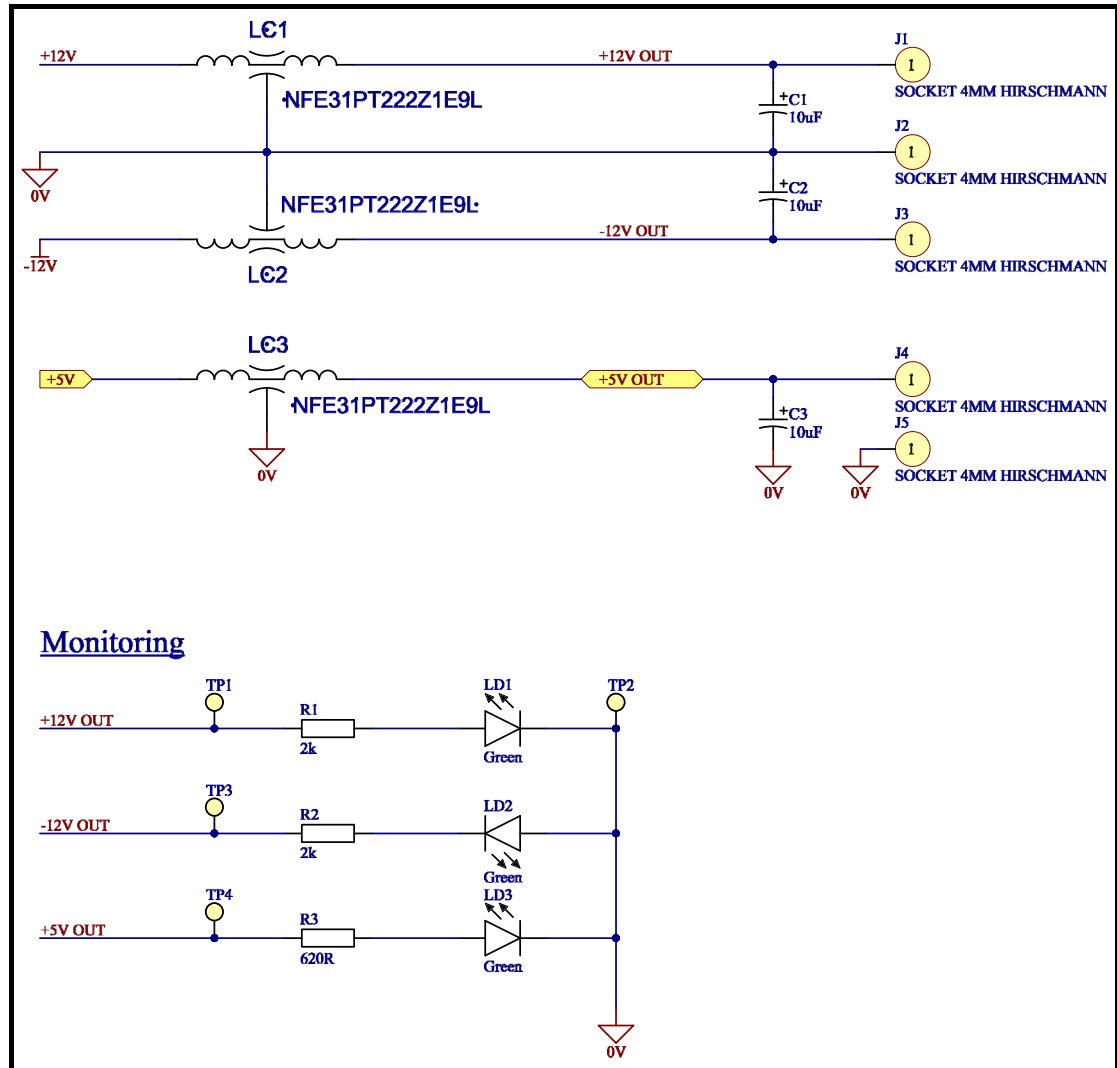
Exercice Altium Designer

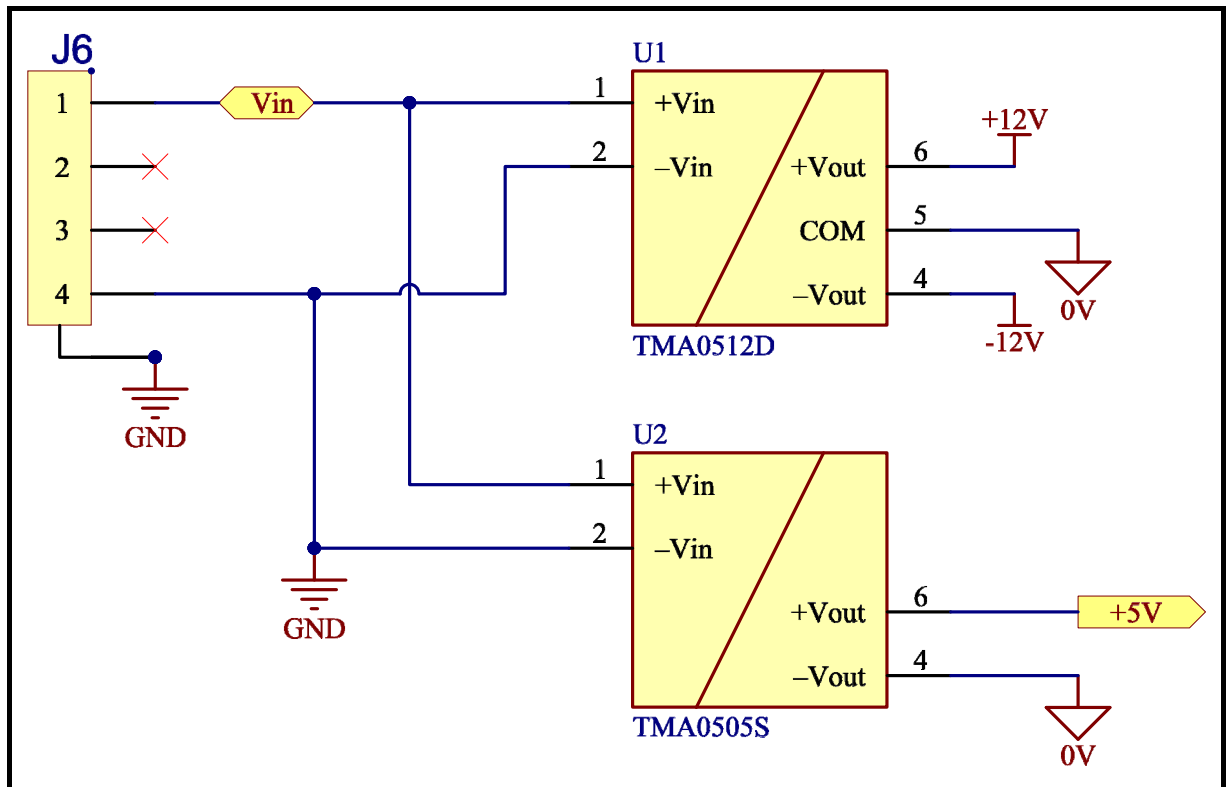


1. Schematic

Schematic

Redraw exactly this 2 sheets of schematic.

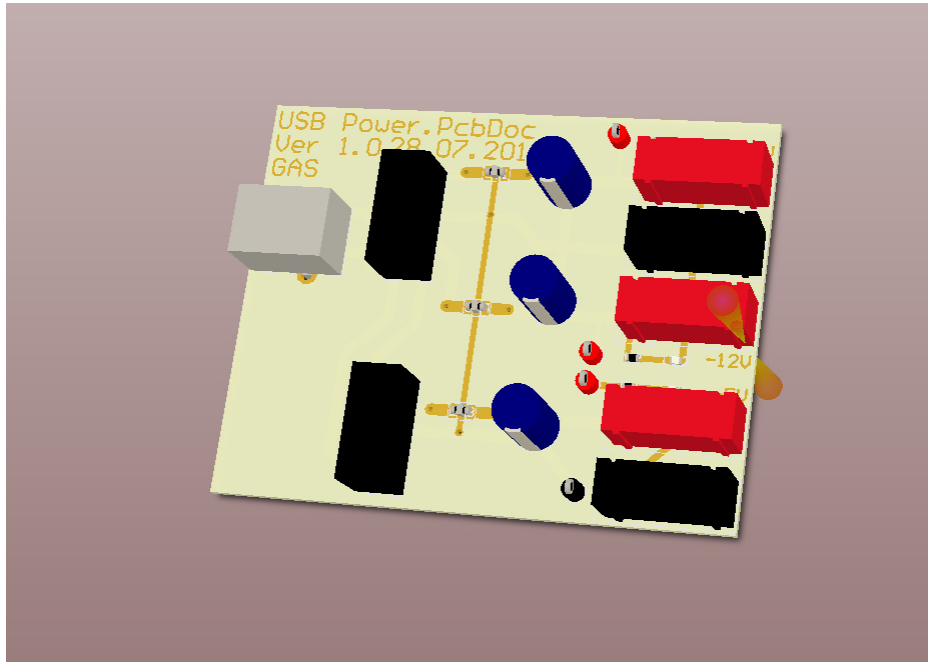




2. PCB

2.1 Routing

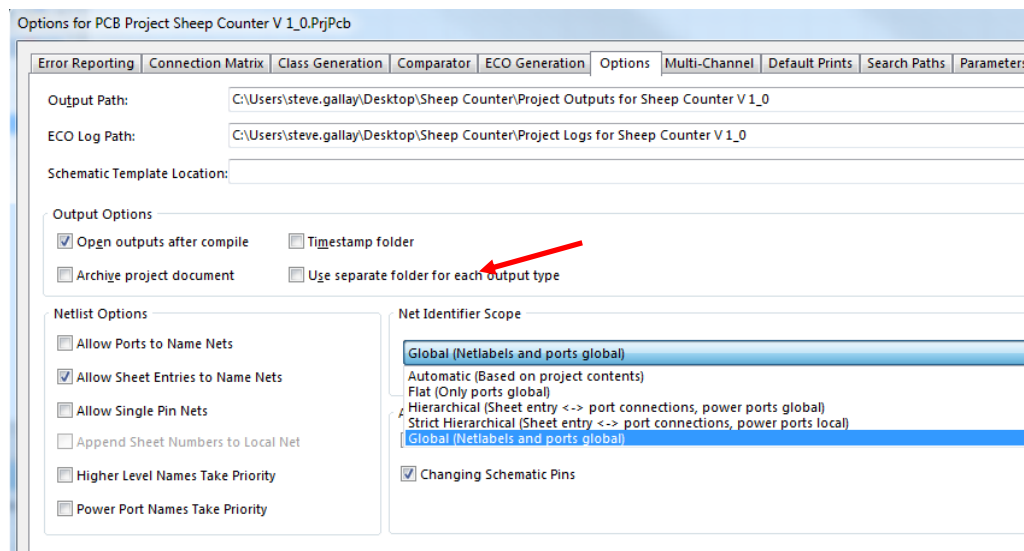
Route all the nets of the board, run Design rules check and observe the pcb.



3. Net identifier scope

3.1 Net identifier scope

Try the different connection method in *Project->Project Options->Options->Net Identifier Scope*



4. Hierarchical schematic

4.1 Hierarchical schematic

Add a new sheet and make a hierarchical schematic.

