PCB Design Course Work Supporting File

Shun YAO 20321234

Start from the template project on Moodle, main components were placed on the PCB board first: Converter Transformer (U26), Rectifier Bridge (U6) and two MOSFETs (U16, U20). The distance between components was designed bigger than 3mm for clearance and convenient for up coming soldering. Four heatsinks (20mm×17mm) were placed around two converter output side diodes and two MOSFETs, the heatsinks and components should be close enough, which here chosen 0.5mm for PCB manufacture.

To reduce the inductance and the clearance between components and connections, all tracks were designed as short as possible. The designed PCB board has two layers. The forward layer was used to track connections the back layer was used to place the copper fills. Two copper fills were used, one for ground pins connection (ground puddle) and another for PWM_COM_B pins (signal puddle).

Two decoupling capacitors (1uF, 0.1uF) were placed in parallel with the MOSFET Driver (U5) VDD pin, this input pin must be decoupled to ground with a local ceramic capacitor as it stated on MOSFET Driver datasheet [1]. This provides a low inductance source for high frequency current. And U11, U12 are the decoupling capacitors for input 30Vac, bigger capacitance would give smoother DC output voltage.

To design the track width and make it capable for certain current. Use the PCB calculator on KiCAD6.0, external track width would be calculated from a given current under designed temperature rise (set to 20℃). For the MOSFET driver whose typical high peak output current of is 4.5A and rated for up to 3A (datasheet) [1], so the track width calculated for it is 0.898mm, use 1mm in PCB editor.

Figure1 shows the basic schematic circuit of the main converter, the output is 8Vdc, at a maximum power of 50W, so the current flow in this side could be calculated by Equ1. The input is 30Vac, use Equ2, assume duty cycle is 0.4 here (<0.5) could give $N_2/N_1 = 2/3$. Eliminate the magnetizing current as 10% of the reflected load current (often much less than that), so the $i_{MAG} = 0.1 \times i_2 = 0.625A$. Hence, use Equ3, we can eliminate the input side current $i_1 = 4.792A$. The calculator gives the input, output side track width are 2.471mm and 1.713mm. Use 2.5mm and 2mm in PCB editor. The board passes the DRC without errors and the warnings are result in an update of the software.

All the external connections, test points and their polarity are labeled on Front Silkscreen.

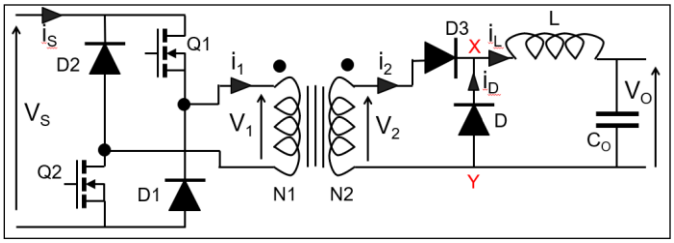

Figure1. Two transistor Isolated Forward Converter

$$i_2 = \frac{P}{U} \qquad Euq1$$

$$\frac{V_O}{V_S} = \frac{N_2}{N_1} d \qquad Equ2$$

$$i_1 = \frac{N_2}{N_1} i_2 + i_{MAG} \qquad Euq3$$

[1] Alldatasheet.com, “TC4424AVPA pdf, TC4424AVPA description, TC4424AVPA datasheet, TC4424AVPA view ::: Alldatasheet :::” *ALLDATASHEET*. [Online]. Available: https://pdf1.alldatasheet.com/datasheet-pdf/view/554300/MICROCHIP/TC4424AVPA.html. [Accessed: 05-Feb-2023].