Power Supply Design Assignment

Requirements

Parameter	Value
Input Voltage	12V
Output Voltage	3.3V
Output Current (max)	500mA
Allowed Ripple	<100mV
Transient voltage deviation	+/-150mV
PCB Dimensions	48mm x 48mm
I/O Connector	2.54mm pitch header pins

Regulator IC: MC34063ECN

This regulator was chosen from the available options due to its THT nature for ease of soldering and is a very common component that has been widely used. Parameters such as max switch current and input voltage range were largely identical across the options and thus, did not influence the selection.

Calculations & Component Justification

The following calculations are primarily based on the equations provided in the MC34603ECN datasheet.

Feeback Resistors:

V_out = 1.25*(1+R2/R1) = 3.3V

R2 = 16.2kohm

R1 = 10kohm

V_out = 3.275V

These resistor values are standard values which give an output voltage that is within reason to the desired output voltage.

f = 50kHz, T = 1/f = 20us

A switching frequency of 50kHz was chosen as the maximum frequency of the IC is 100kHz and this value is within the safe operating range. This value also provides a good balance between minimising output ripple and avoiding greater switching losses.

Duty Cycle: D = 3.3V/12V = 0.275

t_on = D*T = 5.5us

t_off = (1-D)*T = 14.5us

Timing capacitor: C_T = 4.5*10^-5*t_on = 247.5pF = ~240pF

The value of 240pF as this is a standard value and is reasonably close to the calculated value such that it won't affect the switching frequency of the IC.

$I_{PK(switch)} = 2*I_{out(max)} = 2*500mA = 1A$

$R_{SC} = 0.3/ I_{PK(switch)} = 0.3/1 = 0.3ohm$

Output capacitor: $C_{out} = (I_{PK(switch)}*T)/(8*V_{ripple}) = (1*20u)/(8*100m) = 25uF =\sim 22uF$

This output capacitor value was chosen as it is a standard value and close to the calculated value.

Inductor (min): $L_{min} = (V_{in(min)} – V_{sat} – V_{out})/I_{PK(switch)}*t_{on} = (12-1.3-3.3)/1*5.5*10^{-6} = 40.7uH = \sim 220uH$

The equation provides the minimum inductor value which is required for the peak current to not exceed its requirement. However, a larger inductor provides a safe margin, reduces the output voltage ripple and improves performance during transients on the load. Thus, although the 82uH inductor may be adequate, the 220uH is more suitable and thus, was selected. Furthermore, the THT component of the inductor was selected due to the ease of soldering and changing out to the 82uH inductor if necessary or to compare their effects.

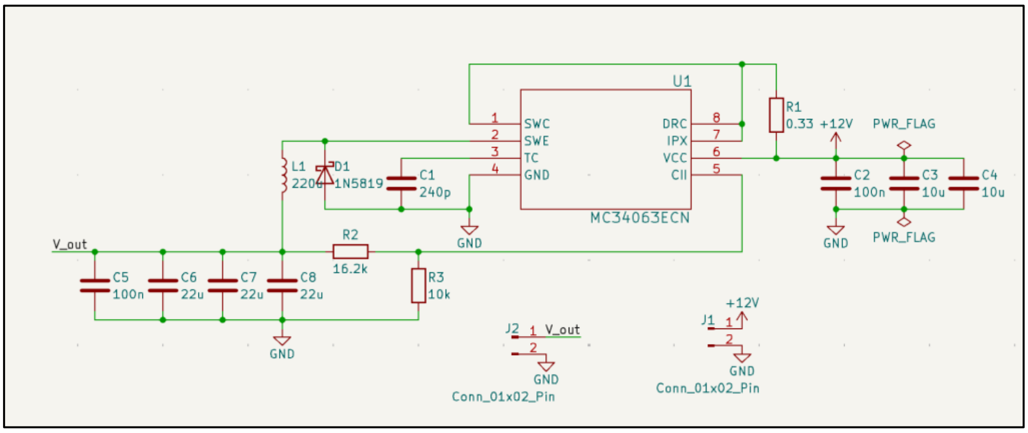

Figure 1: Schematic

Figure 1 above displays the schematic of the power supply design. It features all the components calculated earlier. The 1N5819 Schottky diode was chosen due to its low forward voltage drop, rating for 1A continuous and fast switching. This diode provides the discharge path for the inductor during the ‘OFF’ switching phase.

Input capacitor values of 100nF and 10uF were selected as the 100nF filters high frequency noise and the 10uF acts as a bulk energy storage. The output capacitor values of 100nF and 22uF were selected as the 100nF also filters out the switching and high frequency noise and the 22uF is from the calculations and is to supress the inductor output ripple. The schematic also features excess input and output capacitors which only the footprints will be placed to allow for optimisation during testing and provide no downside to the function of the design. These can be used to improve the output voltage ripple by increasing capacitance and reducing the equivalent series resistance (ESR), if needed.

All resistors and capacitors were chosen to be SMD, 0805 values with all capacitors being ceramic. The 0805 size was chosen as this is a common value that is large enough to hand solder if needed, particularly if capacitors need to be added or removed during testing. Furthermore, having all 0805 components allows for compatibility across different capacitor values.

Layout Justification

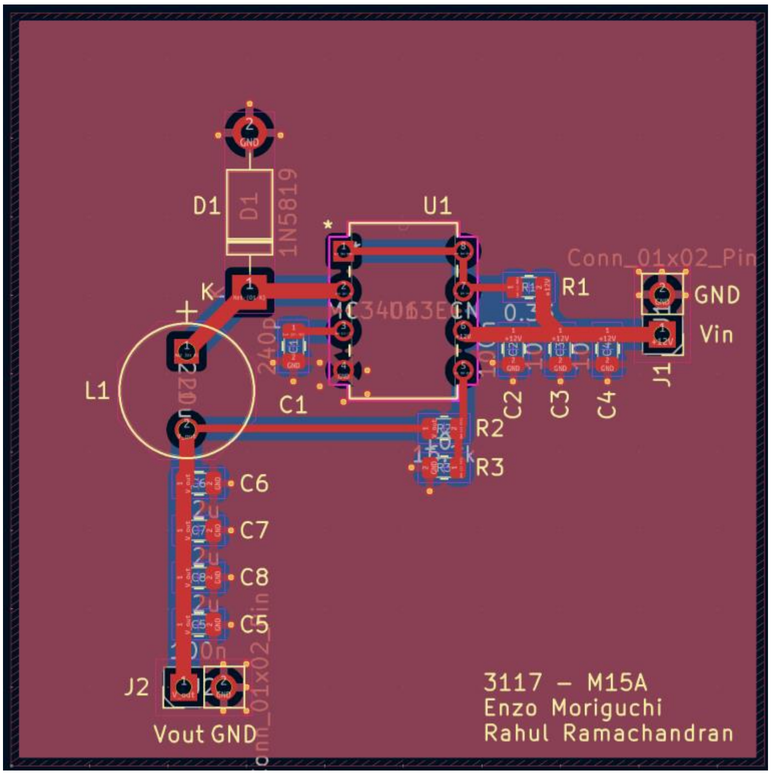

Figure 2: Layout

Figure 2 above displays the layout of the power supply, which features a 2-layer board with ground planes on both sides, stitched with vias close to all GND pads to provide low impedance return paths. All traces are kept short and direct, with 1mm width traces used for power traces to handle the higher currents and reduce resistive losses. 0.5mm traces were used for all other traces as they carry much less current.

On the input side, the 100nF input capacitor is placed closest to the IC to filter high frequencies with the larger 10uF capacitors placed close by for stabilising the input voltage. Similarly, the output capacitors are placed as close as possible to the output to filter out noise and smooth the voltage. The capacitors are placed slightly apart, not as close as possible, to allow for ease of modifications during testing as previously mentioned. This will allow for soldering or desoldering of single capacitors without affecting nearby capacitors.

The inductor and diodes are placed close to the output pin of the IC to minimise the loop area that can be formed from these components in conjunction with the output capacitors.

It should also be noted that although it is common practice to avoid 90-degree traces, the ones used here through the THT pins have little to no impact on the performance of the power supply, only to simplify routing, and given that this is not a high-speed design, with a low frequency (50kHz) used.