AND9133/D

Circuit Design and PCB Layout Guidelines for Designs Using the NCP108x

ON Semiconductor®

www.onsemi.com

APPLICATION NOTE

Introduction

The NCP108x integrates a Power-over-Ethernet interface with a DC-DC controller, reducing the BOM for a Power-over-Ethernet implementation. However, DC-DC convertor design is not always easy, especially considering the many EMC and EMI pitfalls. Regulatory testing often reveals that a convertor design radiates more than is allowed by regulations. In this document, we will give some guidelines on how to build a power supply that can pass these tests.

Schematic Design and Component Selection

Designing a PCB that will pass regulatory testing requires that attention for EMC issues is not an afterthought, but is present right from the beginning at the start of the circuit design. Often, not taking small measures early on in the design will result in big and costly patches later on, such as ferrite bead clamps on the cables which could have been avoided by placing much smaller ferrite beads on the PCB. Here are a couple of things to do while drawing the schematic and selecting the components:

- On the input connector, use a Bob-Smith Termination Network to properly terminate the Ethernet connection.
- Use input common-mode chokes.
- Use a shielded RJ-45 connector. It is possible to use an integrated connector, which can include common-mode chokes, the Ethernet transformer and the Bob-Smith network. This is preferable, as it will result in a more compact design.
- Use an LC filter on the input and output of the DC/DC convertor.
- Allow for snubbers on all switching components. When designing a flyback convertor this absolutely required, but even on other topologies this can be beneficial.
- Limit the switching edge rise time by inserting a small resistor before the gate of the switching transistor.
- Place a small (1 nF or 2.2 nF) bridge capacitor across the isolation boundary. This capacitor should be rated at least 2 kV.
- Use ferrite beads on input and output lines of the DC/DC convertor. When not required, these beads can be replaced by 0 Ω resistors.
- Foresee a possibility for a shielding box over the DC/DC convertor.

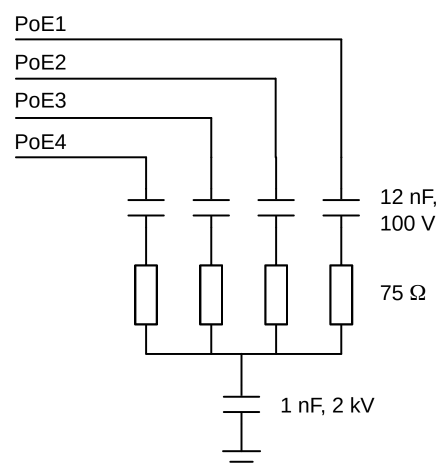

Figure 1. Bob Smith Termination for Power-over-Ethernet Circuits

PCB Layout and Component Placement

The most critical time for avoiding EMC issues in the product design is during the PCB layout. Great care must be taken to avoid current loops and radiating stubs. Below is a list of attention points that will minimize the risk of issues later on:

- If possible, use a 4 layer PCB. This will allow big ground and power planes, and will allow return currents to flow freely, resulting in compacter current loops.
- Use copper ground planes and surround the outside layers in a copper ground flood.
- Tie ground flood and planes together with vias at regular intervals.
- Ground and power planes should only be present near areas where these planes are used. Output ground and power layers should not be present under the Ethernet input or the converter primary side.
- At the input, use the Bob-Smith Termination virtual ground as the ground shield.

- Pair signal and return traces in paths that carry high-frequency or high-current signals. If possible, route the return path directly below the signal path. The loops for high dv/dt or di/dt signals should be as compact as possible.
- Keep the length of all traces minimal, with priority for the power and switching traces.
- Power traces should be made as wide as possible.
- Part placement should be determined by the power flow. Components belonging to the same function should be grouped together so as to minimize trace length and keep high-noise lines local. There should not be any crossovers of signals from one part of the flow to another.
- The NCP108x should be connected to local ground planes referenced to VPORTN for the Power-over-Ethernet side and to RTN for the converter side. The exposed pad should be connected to VPORTN.
- Snubbers should be placed as close to the switching component as possible. Similarly, TVS and decoupling capacitors should be placed as close to the input and output connectors as possible.
- The bulk input and output capacitors should be placed close to the switching components, so that the current loop created by capacitor, transistor and transformer is as compact as possible. The output filter should be located close to the bulk output capacitor.
- Keep a distance between the power and switching lines and sensitive low-voltage lines. If possible, foresee a section of ground plane in between. This will prevent noise from coupling over. Spacing should be consistent with the safety standards that apply to the product.
- The copper associated with the switching nodes should be shielded as much as possible, such as by a ground plane or by shielded components placed over it.
- When adding a heat sink to a component, preferably place the heat sink on the side of the component that sees the least amount of noise (such as the output pin of an inductor). Use the minimal amount of copper that is still adequate for cooling under all circumstances, so as to minimize the exposed radiating surface.

PUBLICATION ORDERING INFORMATION

LITERATURE FULFILLMENT:
Literature Distribution Center for ON Semiconductor
P.O. Box 5163, Denver, Colorado 80217 USA
Phone: 303−675−2175 or 800−344−3860 Toll Free USA/Canada
Fax: 303−675−2176 or 800−344−3867 Toll Free USA/Canada
Email: orderlit@onsemi.com

N. American Technical Support: 800−282−9855 Toll Free USA/Canada
Europe, Middle East and Africa Technical Support:
Phone: 421 33 790 2910
Japan Customer Focus Center
Phone: 81−3−5817−1050

ON Semiconductor Website: **www.onsemi.com**

Order Literature: http://www.onsemi.com/orderlit

For additional information, please contact your local Sales Representative