

DM9620/9621 Demo Board Layout Guideline

Davicom Semiconductor Inc.
05/04/2010

Reference circuits: DM9620/9621 Demo Board V1.01

Impedance requirements:

1. USB trace pairs to be 90 ohm differential and Ethernet trace pairs to be 50 ohm single-end and 100 ohm differential
 - 1.1 USB traces are DP and DM signal pair.
 - 1.2 Ethernet traces are TX+ and TX- signal pair and RX+ and RX- signal pair.

Placement requirements:

1. Place C16 and C19 as a group and close to DM9620 pin 10 or DM9621 pin 10.
2. Place C5, C87 and C88 as a group and close to DM9620 pin 58 or DM9621 pin 44.
3. Place bypass capacitors close to associated power pins.
 - 3.1 Place C89 and C90 as a group and close to DM9620 pin 32 or DM9621 pin 24
 - 3.2 Place C50 and C51 as a group and close to DM9620 pin 59 or DM9621 pin 45.
 - 3.3 Place C56 and C57 as a group and close to DM9620 pin 51 or DM9621 pin 37.
 - 3.4 Place C58 and C59 as a group and close to DM9620 pin 56 or DM9621 pin 42.
4. Place J2, DM9620/21, T1 and JP1 as close as possible.
5. Do not place any power or ground planes in the area between T1 and JP1.
6. Place C9 and C10 in between Y1 and DM96/2021.
7. Place R28 and R34 as close to DM9620/21 as possible.

Routing requirements:

1. Route power nets, VCC33, AVCC33 and VCC33_PLL, as star type distribution with a common connection point only.
2. High speed signals, DP/DM pair, TX+/TX- pair and RX+/RX- pair, are to be route in pair and with as short as possible distance and minimize number of vias.
 - 2.1 In order to maintain consistent impedance, place a solid ground plane under these trace pairs.
3. Signals do not cross over DP/DM pair, TX+/TX- pair and RX+/RX- pair.
4. Do not cross power input on pin 3 of U4 with output power on pin 2 of U4.

Other requirements:

要解決 QFN 焊接等品質問題，應注意以下幾點：

1. 鋼網開孔：中間焊盤 50%開孔，形狀為“田”字形，管腳焊盤長度向外擴 0.1~0.2mm,以增加錫量。
2. 貼裝位置要十分準確，建議用顯微鏡觀察貼片狀態，多數虛焊問題都是由於貼片位置偏造成的。
3. PCB 板厚度在 1mm 以上，由於 PCB 不易變形，不容易產生虛焊現象。如 PCB 板厚度在 1mm 以下並為連板時，要注意後製成拆板方法，避免 PCB 板受力產生變形引發的虛焊問題