

Re: decoupling caps placement

Source: <http://sci.tech-archive.net/Archive/sci.electronics.basics/2008-12/msg00379.html>

- *From:* IanM <Invalid@xxxxxxxxxxxxxxxxxx>
 - *Date:* Mon, 22 Dec 2008 16:19:42 +0000
-

Tim Wescott wrote:

On Sun, 21 Dec 2008 16:33:47 -0500, tempus fugit wrote:

Hey all;

I've got a circuit that uses 3 4049 inverters. On this IC, the V+ is on pin 8 and the ground is on pin 1. I know that the decoupling caps need to be as close to the IC as possible, but how can I connect 1 end of the cap to V+ and the other to ground when the pins are so far away? Is it sufficient to connect 1 end of the cap to V+ and the other to a nearby ground node, or should the cap be connected close to the actual ground pin of the IC? Also, do I use 1 cap for each IC? If so, (the ICs are fairly close together) wouldn't the IC "see" the caps as being the paralleled value of the 3 caps, thus reducing the available capacitance? I was going to use 0.1uF for the value of each decoupling cap. Would it also be wise to use a larger (1uF or higher) cap in parallel?

Thanks

For that chip a 100nF (0.1u) ceramic up by pin 1, with a lead straight to pin 1 and another one straight to pin 8 will be more than sufficient.

You can think about reducing them later --- but why? Unless you're trying to shave deci-pennies off of the board cost, it's better to have too many bypass caps than just enough.

Some general comments on effective grounding and decoupling pitched at hobbyists (pros will know how much they can save by cutting back on both, but for the rest of us, the small additional cost is easily repaid by time saved debugging a glitchy circuit):

By ground, I and most other dabblers in low voltage electronics mean chassis, common rail, 0V, battery negative, Logic negative supply etc, NOT a hard wired connection to a copper stake in the earth. Electricians are different - when they say ground, they mean ground!

If building on veroboard or solderless breadboard, make sure you have reliable power and ground rails. Except in exceptional circumstances, DONT wire them point to point. (low level audio is an exceptional

Re: decoupling caps placement

circumstance with special layout and decoupling requirements as are high gain RF amplifiers and high power circuits in general.)

Most solderless breadboards have long rails of contact positions on each edge, USE THEM (but beware of boards with a split between two halves of the rail half way down it, they need a link inserted). Local decoupling can be put over the top of critical ICs. Have an electrolytic capacitor across the power rails where the power comes onto the board. Somewhere between 10 and 100uF axial type is a good choice for most circuits on breadboard.

If building digital or RF circuits on veroboard, you need a *solid* ground rail. 2 or 3 tracks tied together at intervals along the board is about right, or use the board crossways and run a heavy bare solid copper buss wire along the top tacked down at intervals to any tracks you want grounded by a little loop of bare wire over it, soldered through two adjacent holes and to the copper buss wire.

Power rails can be done the same way but are usually less critical.

On veroboard, if you are using thin kynar wirewrap wire or similar for your signal wires, DONT use it for power and ground.

All decoupling caps should lead as directly as possible to the ground buss and to the + supply pin of the chip in question. 0.022uF to 0.1uF disk or resin dipped ceramic capacitors are generally suitable for individual chips. The chip should be located for the most direct ground connection possible. If there is more length of wire than the width of the board between the power supply and the board, put an electrolytic (typically around 100x the individual ceramic capacitors) across power and ground where the supply wires go. Add an additional electrolytic decoupling cap for every 10 chips.

The aim is to keep everything happily stable with respect to the ground bus with any bounce from one IC switching *NOT* getting into other ones.

With some care with layout, bread boards are good for circuits operating at up to a couple of MHz and veroboard up to a few tens of Mhz.

In the case of the OP's 4049 logic circuit, *ALL* 4000 series logic is slow and low power so needs minimal decoupling. If there is no other power consuming circuit on the board, a single 0.1uf ceramic located centrally, WITH THE POWER SUPPLY WIRES CONNECTED DIRECTLY TO IT and as short as possible supply and ground wires radiating to the three chips will almost certainly be fine. If there are LEDs, a speaker or relay or other high current loads, add an electrolytic capacitor as well. However, I wouldn't build it that way unless I needed minimum size/weight.

If one is etching double sided PCBs or building complicated processor boards etc., one is out of the 'dabbler' category, even if still an amateur, so had better have a PROFESSIONAL understanding of layout grounding and decoupling, otherwise you will get PRO sised grief as alluded to by Graham (Eeyore) in his first reply.