

DESIGNER'S NOTEBOOK



Guidelines for Using Decoupling Capacitors on DSP Designs

Contributed by Ralph Weir and Gene Frantz

Design Problem

What are some guidelines for selecting decoupling capacitors for TMS320C3x devices?

Solution

On page 13-14 of the TMS320C3x Users Guide, there is a note that recommends using 0.1 μF decoupling capacitors on the V_{dd} pins of the TMS320C31. Here we will provide tips on the number and types of capacitors you should use.

- 1) On a multilayer PCB with separate $V_{\text{cc}}/V_{\text{dd}}$ planes, half a dozen 0.1 μF will be ample. This assumes that other devices (e.g., memories) are decoupled adequately. Ceramics are the right choice. The real rule is to find capacitors with no inductive component. Electrolytics, for example, have a significant amount of inductance due to their manufacturing method. That is why in many switching regulators you find a large electrolytic in parallel with a small ceramic capacitor.

Note: You can use capacitors in the range of 0.01 μF to 0.2 μF . The purpose is to eliminate noise spikes. You may need to tune to "circuit" for a specific noise component.

- 2) Put the "holes" necessary to bypass every V_{dd} pin on the board such that the capacitors can be auto attached. This not only provides the places for the capacitors, but allows for repair procedures in the future when a noise problem occurs. Don't populate all of the locations for cost reasons, but have the vacant locations there if needed. A corollary is to put the pads down for by-pass capacitors on other signals that might need them. Actually, if well thought out, other components besides capacitors can be placed in these locations. For example "by-pass capacitors" on address, data, and control lines. Later these could be used for many other things (e.g., pull-up/pull-down resistors). The simple rule is that PCBs are expensive to modify or repair.
- 3) If a board has DRAM on it, make sure that you have slightly more 0.1 μF ceramics, but add a couple of bigger tanks – e.g., 10 μF . This is because a DRAM refresh cycle eats power and can cause the whole board's power rails to drop.
- 4) Make sure that the power tracks are wide, low-impedance vias.
- 5) If a board has problems after following these guidelines, try decoupling with additional mylar capacitors. These have a great frequency response but will add to the power consumption.