Application Note AN-6009

Components Calculations and Simulation Tools for FAN6520A

Background/Overview

To simplify designs using the FAN6520A PWM modulator, Fairchild has developed:

- An Excel spreadsheet to calculate recommended component values. There are two separate spreadsheets, each one for Type 2 and Type 3 Compensation.
- Phase Gain Bode Plot inside the Excel spreadsheet.
- Two Mathcad documents for users who would like to use Mathcad for calculations.

  and

- A continuous time behavioral model of the modulator that runs in PSPICE A/D. Provides transient analysis and Bode plot outputs. The model is sufficiently simple that it will run under Cadence's Orcad Lite Edition (includes Orcad Capture and PSPICE A/D, orderable on CD) or PSpice 9.1 Student Version (downloadable) which are available from Orcad at: http://www.orcad.com/downloads/demo/default.asp

This package of design aids (including this document) is contained in AN-6009.ZIP which can be found on: http://www.fairchildsemi.com/collateral/AN-6009.zip.

To install, copy AN-6009.ZIP to an empty folder (e.g. FAN6520 Design). Then unzip AN-6009.zip into that folder.

Recommended Design Procedure

1. Please go through the "General Instructions" tab to understand the color coding in the spreadsheet.

2. Use the "Main" tab of the spreadsheet to define the Input and output parameters and calculate the component values for output L and C , the Current limit resistor and the feed back bias resistors.

3. Type*_Calc" sheet calculates the recommended values for R1, R2 & R3 (For type 3) and C1, C2 & C3 (For Type 3). These values can be modified based on the designer's choice.

4. View results on the Bode plot inside "Type*_Graph" spreadsheet. (* is for either 2 or 3)

5. Input the values you have selected into the PSPICE model (see Figure 2). Generate a Bode plot by simulating with the "application circuit-ac sweep" simulation profile. Make sure to simulate over the corners of VIN and IOUT for your application.

6. Once you are satisfied with the small signal stability, you can view the transient response by simulating using the "application circuit-transient response" simulation profile.

Design Calculation Spreadsheet

FAN6520 Design Calculation Aid with Type * Comp.xls

This spreadsheet calculates external components and provides a bode plot for stability analysis. Instructions can be found in the "General Instructions" tab of the spreadsheet.

There are two separate files for caculating component values based on type 2 and type 3 compensations.

PSPICE Simulation Model

The simulation model is a sampled data continuous time model, which is adapted from Ray Ridley and Dennis Feucht's modeling work for current mode controllers.[1,2,3] It is set up to provide a bode plot where the red trace is Phase Margin (in degrees) and the green trace is gain (in dB). For stable response, we recommend at least 45° of phase margin when the gain crosses 0 dB. The model also provides transient response using a pulsed current source (I1) as the load. The IC's error-amp behavioral model is based on Ray Kendall's Macromodelling article in EDN.[4]

To run the model start Capture (9.1 or higher), open **FAN6520A.opj** (this is the "project" file for Capture CIS). Double click on Page 1 Under .\fan6520A.dsn\ Application Circuit (See Figure 1).

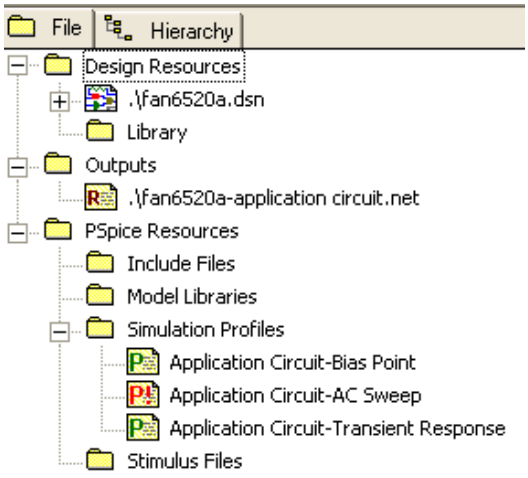

Figure 1. FAN6520A.OPJ Project

The parameters for this model are entered in the "Parameters" block on the lower right-hand corner of the schematic. Double click on any parameter in that block to set the values in the schematic. Once the schematic is set up, F11 (function key) will display the Bode plot.

This model will also allow for transient response simulation. To choose between Bode plot (AC small signal) and Transient response, pick the simulation that you need from the drop down menu on the left hand corner in the application circuit window. If you are simulating for transient response, be sure to set up RLOAD. You will know if you forgot this step, as the inductor current [I(L1)] trace will be much higher (probably beyond the Y axis limit) than the pulse load current [I(I1)].

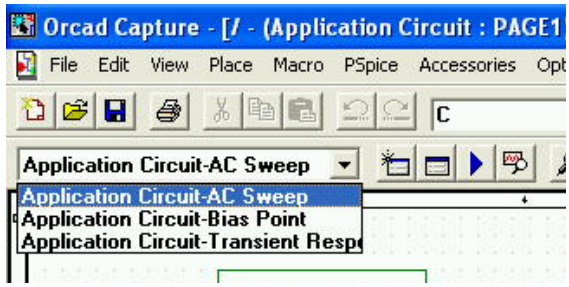

Figure 3. Selecting Between Simulations.

Troubleshooting the Plot Window

Some older versions of PSPICE may not automatically load the probe settings (which are contained in the *.prb files). These settings define the XY axis settings, trace colors, and which signals are displayed. If you run a simulation and the probe window has no trace, then add a trace, and input the expressions for the signals to plot:

Gain: DB(V(Out)/V(Sig)) **Phase**: P(V(Out)/V(Sig))

Copy and paste these expressions into the add trace window as shown in Figure 4. The also schematic contains the expressions for the BODE plot in the lower right hand corner.

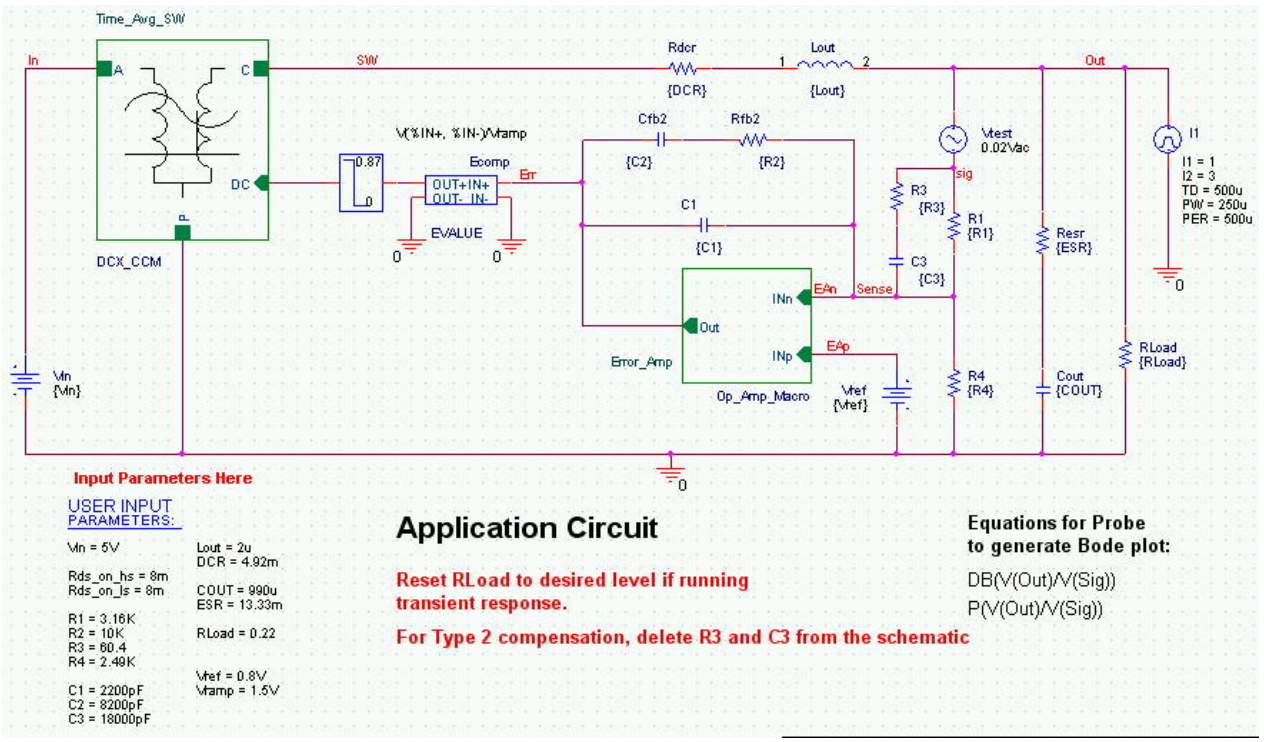

Figure 2. PSPICE Schematic

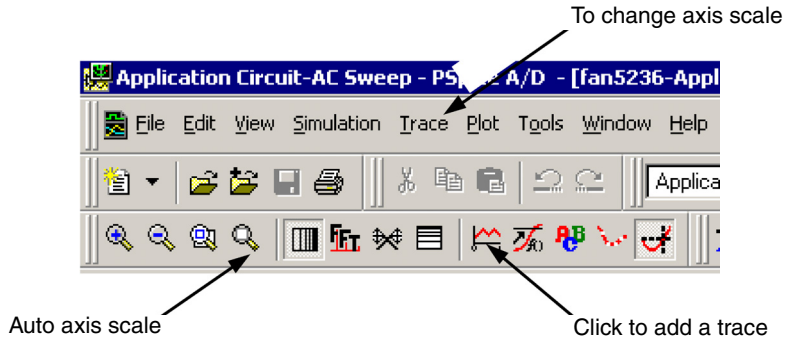

Figure 4. Tips for Adjusting Probe Window Settings and Adding Traces

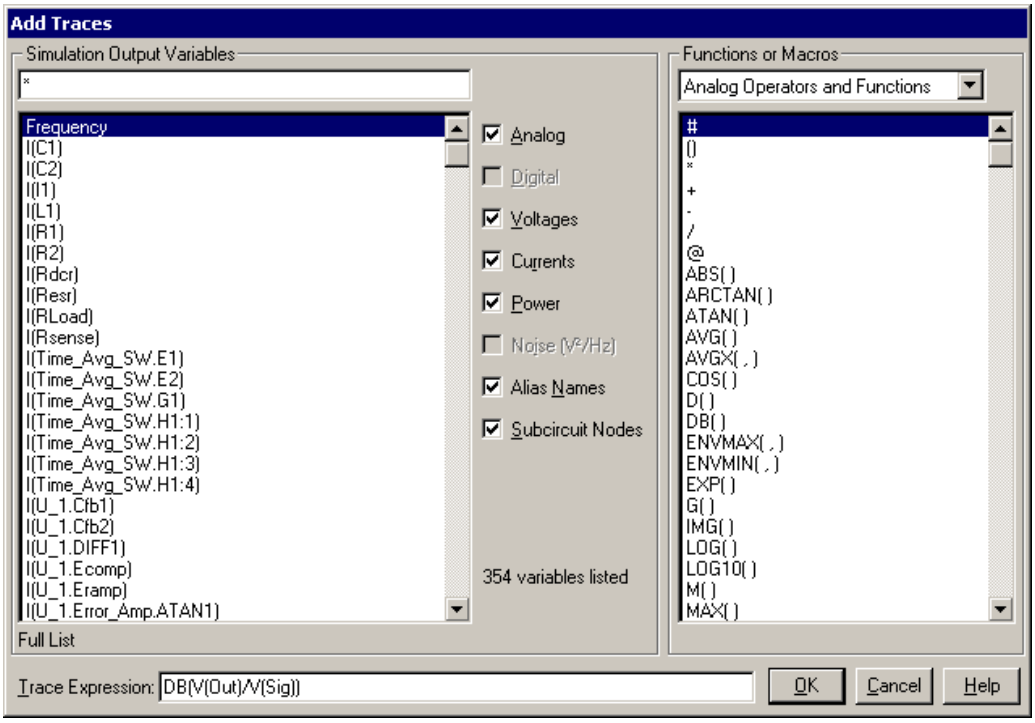

Figure 5. Adding the Trace (Gain Example)

References

1. Ray Ridley, *An Accurate and Practical Small-Signal Model for Current-Mode Control,* 1999, http://www.ridleyengineering.com/downloads/curr.pdf

2. Dennis Feucht, The Tymerski Switch Model, http://www.chipcenter.com/eexpert/dfeucht/dfeucht036.html

3. Dennis Feucht, Basic Power Converter Configurations, http://www.chipcenter.com/eexpert/dfeucht/dfeucht037.html

4. Ray Kendall, Modular macromodeling techniques for Spice simulators, EDN, March 7, 2002 http://www.reed-electronics.com/ednmag/contents/images/198891.pdf