Is Now Part of

ON Semiconductor®

To learn more about ON Semiconductor, please visit our website at
www.onsemi.com

AN-6002

Component calculations and simulation tools for FAN5234 or FAN5236

Background / Overview

To simplify designs using the FAN5234 and FAN5236 PWM modulators, Fairchild has developed:

- A spreadsheet to calculate recommended component values
- NEW Capability added 6/19/04: Phase Gain Bode Plot inside the spreadsheet.
  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 small enough to run under Cadence's Orcad Lite Edition (includes Orcad Capture and PSPICE A/D), which can be ordered on CD at
  http://www.ema-eda.com
  or downloaded from:
  http://www.orcad.com
  These links have been verified as of this publication date but may change over time.

This package of design aids (including this document) is contained in AN-6002.ZIP which can be requested on: http://www.fairchildsemi.com/.

To install, copy AN-6002.ZIP to an empty folder (e.g. FAN5236Design). Then unzip AN-6002.ZIP into that folder.

Bode PlotsA new feature has been added to the Spreadsheet tool

Recommended design procedure:

1. Use the spreadsheet (FAN5236 Design calculation aid.xls) to calculate the output filter using the “Output Filter” tab.
2. Use the “Main Sheet” tab of the spreadsheet to calculate the other component values once you have selected the output L and C. Most applications will not require C2. Try your first simulation with this value set to 1pF.
3. View results on the Bode plot inside the spreadsheet.
4. Use the “input Filter” tab to determine the ripple current rating required for the input filter capacitor.
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: FAN5236 Design calculation aid.xls

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

Macro Security Note:
FAN5236 Design calculation aid.xls uses macros extensively. For the spreadsheet to operate properly, check the “Always trust macros from this source” box if a security warning appears, then click the “Enable Macros” button..

This is only required the first time you run a Fairchild spreadsheet tool with macros.

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 0dB. 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 **FAN5236.opj** (this is the "project" file for Capture CIS). Double click on Page 1 Under .\fan5236.dsn\Application Circuit.

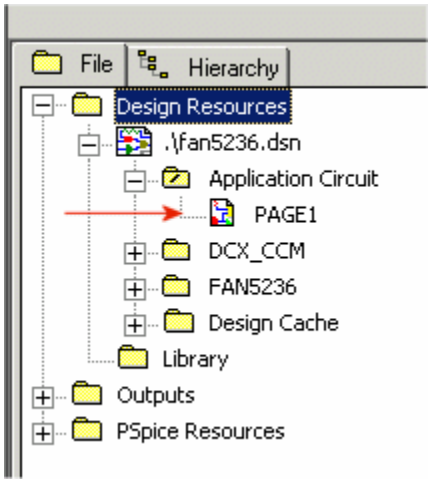

Figure 1. FAN5236.OPJ project

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.

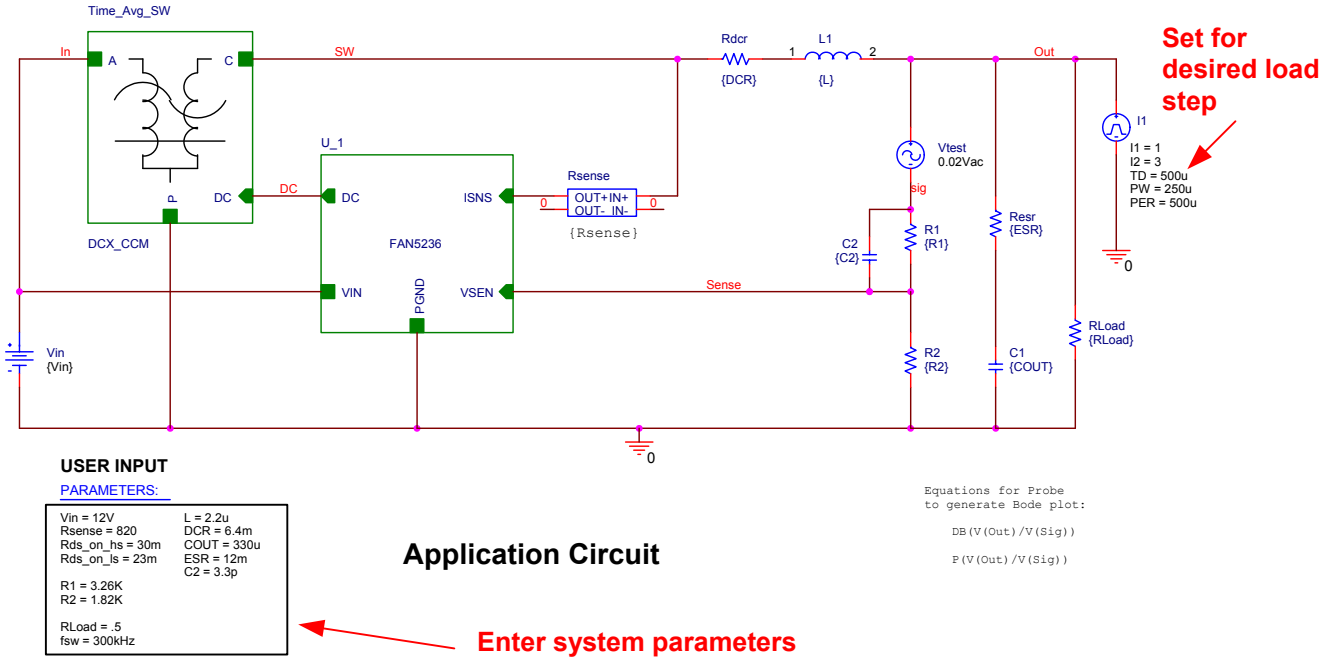

Figure 2. PSPICE schematic

This model will also allow for transient response simulation. To choose between Bode plot (AC small signal) and Transient response, go to the project window and right click the appropriate model. 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)].

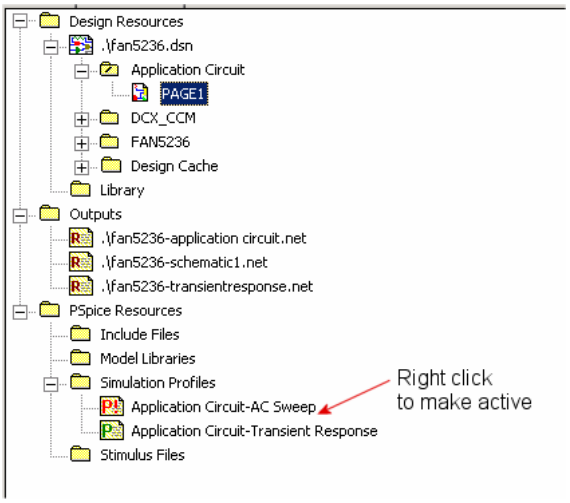

Figure 3. Selecting between Bode plot and Transient simulations.

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 below. The also schematic contains the expressions for the BODE plot in the lower right hand corner

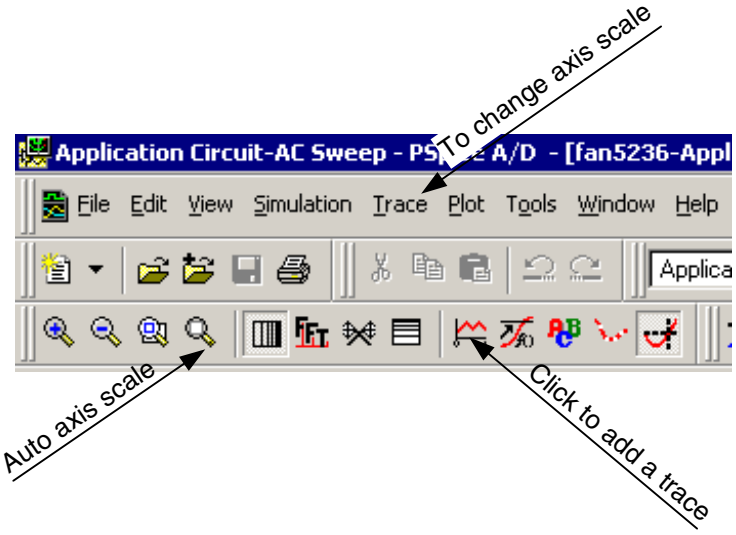

Figure 4. Tips for adjusting probe window settings and adding traces

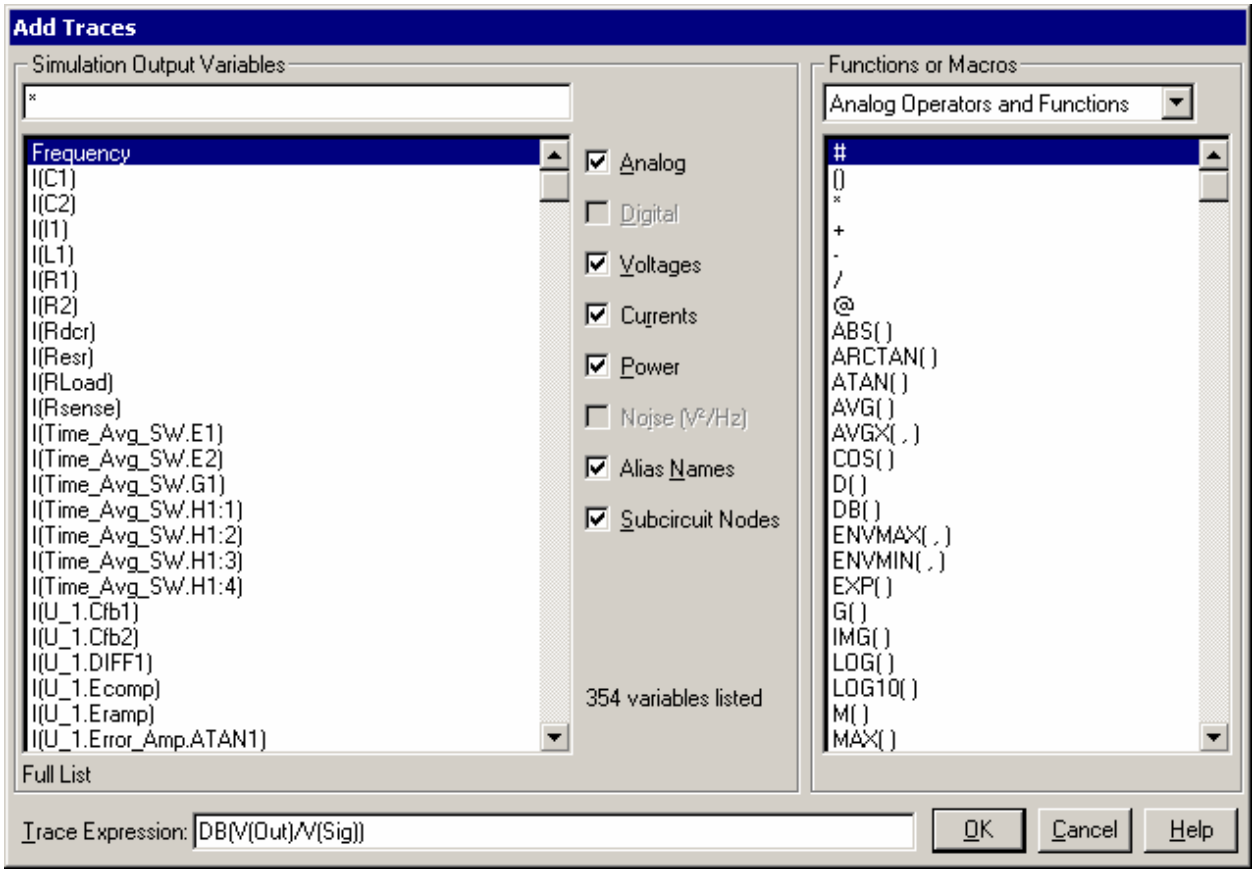

Figure 5. Adding the trace (Gain example)

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

PUBLICATION ORDERING INFORMATION

LITERATURE FULFILLMENT:
Literature Distribution Center for ON Semiconductor
19521 E. 32nd Pkwy, Aurora, Colorado 80011 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