AB-19

FAN5230, FAN5233, FAN5235
Component Calculation Spreadsheet and
PSPICE model Guide

Background / Overview

To simplify designs using the FAN5230 family of PWM system regulators, Fairchild has developed

- A spreadsheet to calculate recommended component values
                            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/products/Default.aspx
  or downloaded from:
  http://www.orcad.com/download.orcaddemo.aspx.
  These links have been verified as of this publication date but may change over time.

This package of design aids (including this document) can be found at :
 http://www.fairchildsemi.com/collateral/ab-19.zip .

To install, copy AB-19.ZIP to an empty folder (e.g. FAN5235Design). Then unzip AB-19.ZIP into that folder.

The schematic for these calculations is in the FAN5230, FAN5233 or FAN5235 datasheets, figure 8.

Design Calculation Spreadsheet: FAN5235 Design calculation aid.xls

This spreadsheet contains macros, which are largely used for navigation. You will need to turn macro security off to use them. Depending on your version of Excel, this is either located in Tools|Options "General" tab checkbox, or Tools|Options, Security tab, "Macro Security" button.

An AutoExec macro takes you to the "Instructions" tab of the workbook, which outlines the functions of each tab in the workbook.

Macro Security Note:
FAN5235 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[i][ii][iii]. 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. [iv]

To run the model start Capture (9.1 or higher), open **FAN5235.opj** (this is the "project" file for Capture CIS). Double click on Page 1 Under .\fan5235.dsn\Application Circuit.
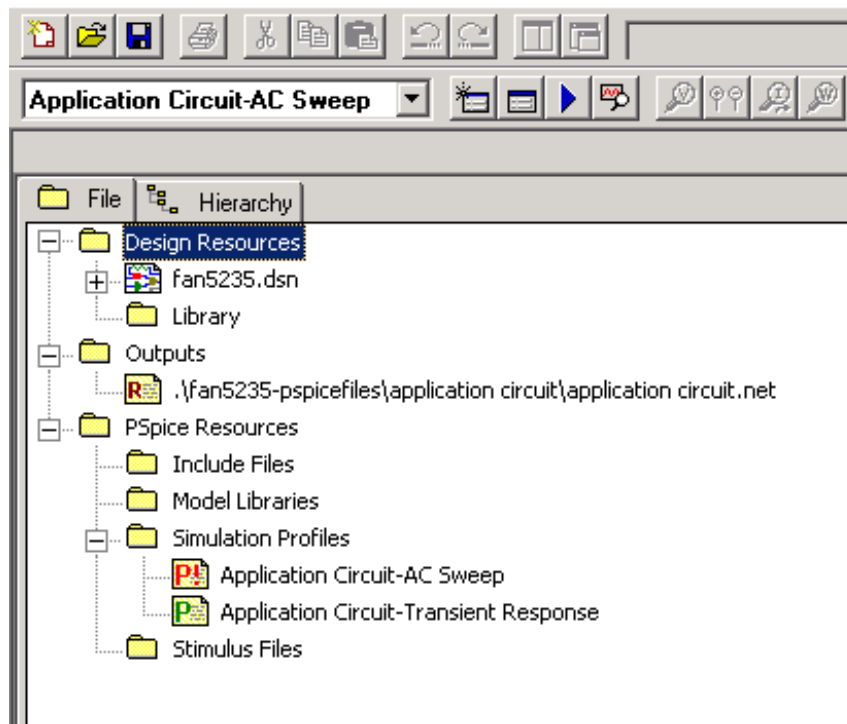
Figure 1. FAN5235.OPJ project

The parameters for this model are entered in the "Parameters" block on the lower left-hand corner of the schematic. As such, there should be no need to edit the schematic itself.

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.

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)].

Alternately, from the schematic view, select the type of simulation profile from the drop-down box as shown below.

Choosing between 5V regulator and 3.3V regulator

The model is set up as a general purpose regulator with a 2.5V reference. To model the FAN5235 5V regulator, edit
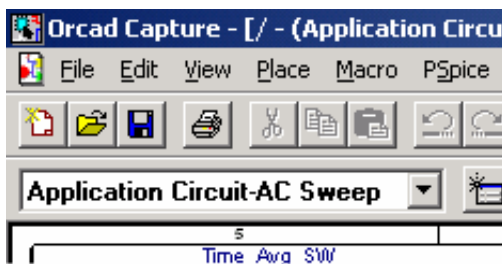

Figure 2. Simulation profile select drop-down

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
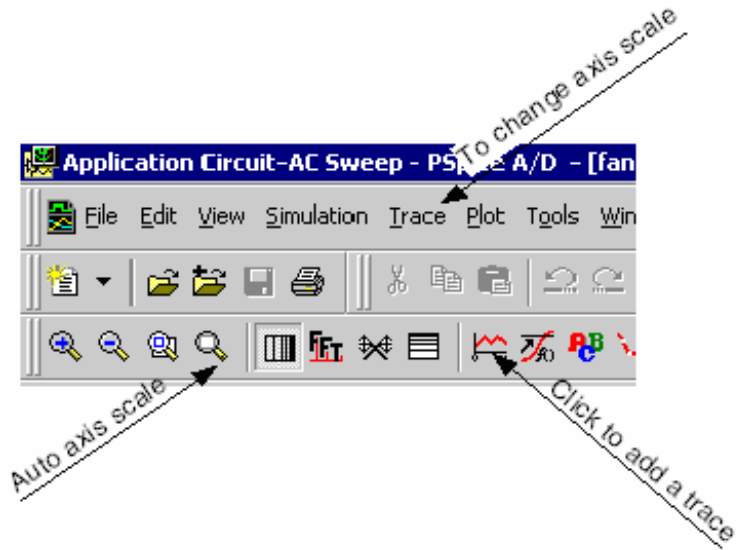

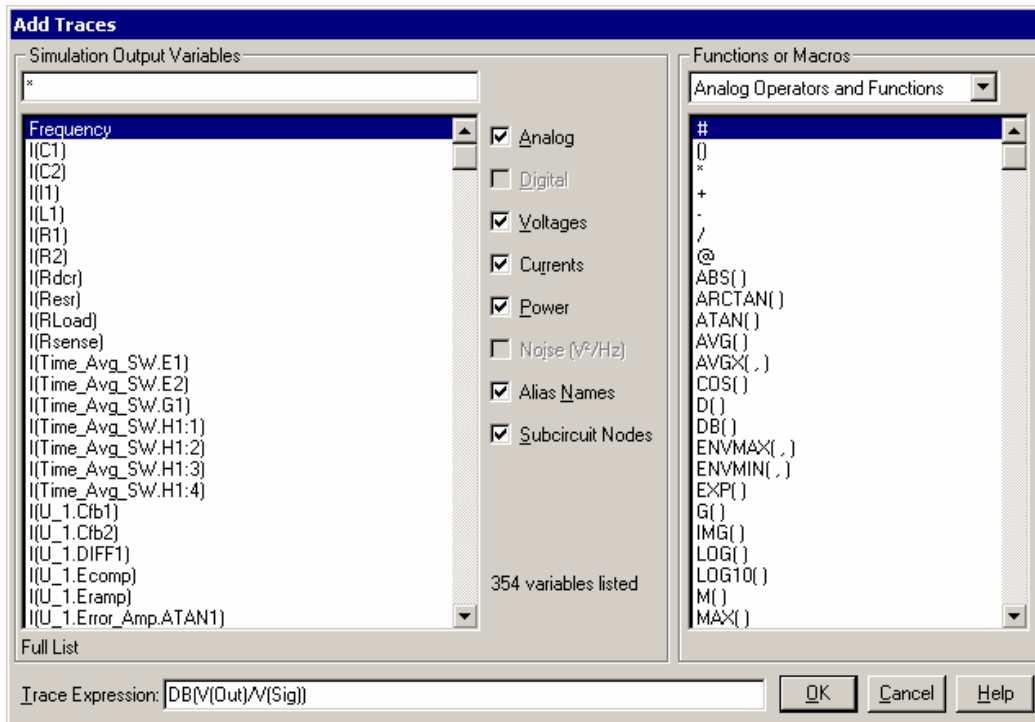**Figure 3. Tips for adjusting probe window settings and adding traces**

Figure 4. Adding the trace (Gain example)

---

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

[ii] Dennis Feucht, *The Tymerski Switch Model,* http://www.chipcenter.com/eexpert/dfeucht/dfeucht036.html

[iii] Dennis Feucht, *Basic Power Converter Configurations,*
http://www.chipcenter.com/eexpert/dfeucht/dfeucht037.html

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