www.fairchildsemi.com

AN-6033
Component Calculations & Simulation Tools for FAN2106

Summary

Fairchild Semiconductor has developed application tools to minimize the design time for the FAN2106 IC. The following (independent) tools are available:

- A spreadsheet calculator to calculate all the component values and estimate losses

  The Excel-based spreadsheet accepts a few design variables and returns recommendations on the inductor, capacitor, and other components, then provides the Bode Plots along with an estimate on the losses based on the components chosen.

- An average simulation model that can be run on the full version of Pspice/AD

- An average simulation model that can be run on the evaluation version of Pspice/AD

  These simulation models are average models optimized to provide Bode plots and run transient analysis. These models can be used to run iterative simulations over line / load conditions.

These tools are available on Fairchild's website in a zip file (AN-6033.zip) and can be downloaded by clicking on the link below.

http://www.fairchildsemi.com/collateral/AN-6033.zip.

Spreadsheet Calculator

Requirements:

- Microsoft® Excel 2002 or later
- Analysis ToolPak Add-In
- Analysis ToolPak – VBA Add-In

Before Running the Calculator

1. Copy AN-6033.ZIP to an empty folder (e.g. FAN2106).
2. Extract the calculator from AN-6033.ZIP into that folder.
3. Run Microsoft Excel and choose **File… New…** to open a blank worksheet (or in Windows, run **Start… Programs… Microsoft Excel**).
4. From the Excel menu, select **Tools… Add-Ins** to open the following window.

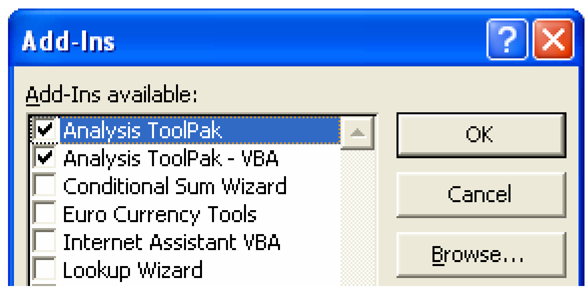

Figure 1. Add-Ins Dialog Box

5. Select (check) **Analysis ToolPak** and **Analysis ToolPak – VBA** and click **OK**.
6. Close the spreadsheet.

Running the Calculator

1. Choose **File…Open…**
2. Locate and select the FAN2106 calculator (in the folder where you extracted the ZIP file.)
3. When prompted to enable macros, choose **Yes** (macros are required for the calculator).
4. Start entering values for your design!

PSPICE Simulation Model

To work with the simulation tool, copy AN-6033.ZIP to an empty folder (e.g. FAN2106), then unzip AN-6033.ZIP into that folder to get all the simulation files into that folder.

Evaluation versions of PSpice can be ordered from Cadence™ at:

http://www.cadence.com/downloads/orcad/requestform.aspx?dl=orcadDemo

The following descriptions are applicable to both evaluation and full version of PSpice-A/D. These PSpice models are configured to run AC (Bode plots) and Transient response analyses.

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. To ensure a stable system over production variations, Fairchild recommends at least 60°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 error-amp behavioral model is based on Ray Kendall's Macromodelling article in EDN[iv].

1. Start the program Capture CIS.
2. Open **FAN2106.opj** (this is the "project" file).
3. Double-click on **Page 1** under .\fan2106 pspice avg model.dsn\Application Circuit.

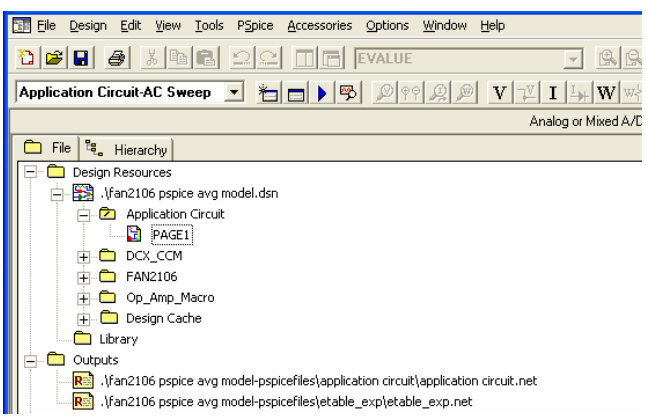

Figure 2. FAN2106.OPJ project

4. Select the type of simulation in the drop-down in the left-hand top corner. Select "Application Circuit-AC sweep" for steady-state analysis or "Application Circuit – Transient" for load transient analysis.

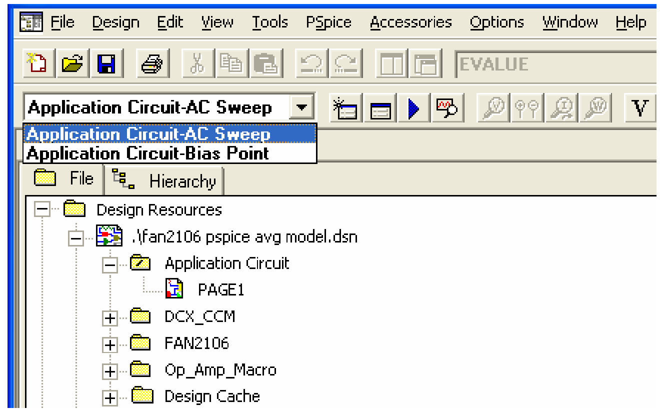

Figure 3. Selecting the Simulation Type

5. Enter the parameters for this model in the "Parameters" block in the lower left-hand corner of the schematic.
6. Double-click on each parameter to set the value in the schematic *(see Figure 4).*

Once the schematic is set up, the F11 (function key) runs the simulation and displays the Bode plots.

To change how the "probe" window is displayed, click on the Edit Simulation Settings button

In the "Probe" window, edit the settings.

Note: When simulating for transient response, set up R_{LOAD} to desired value. If this step is skipped, the inductor current [I(L1)] trace is much higher (probably beyond the Y axis limit) than the pulse load current [I(I1)].

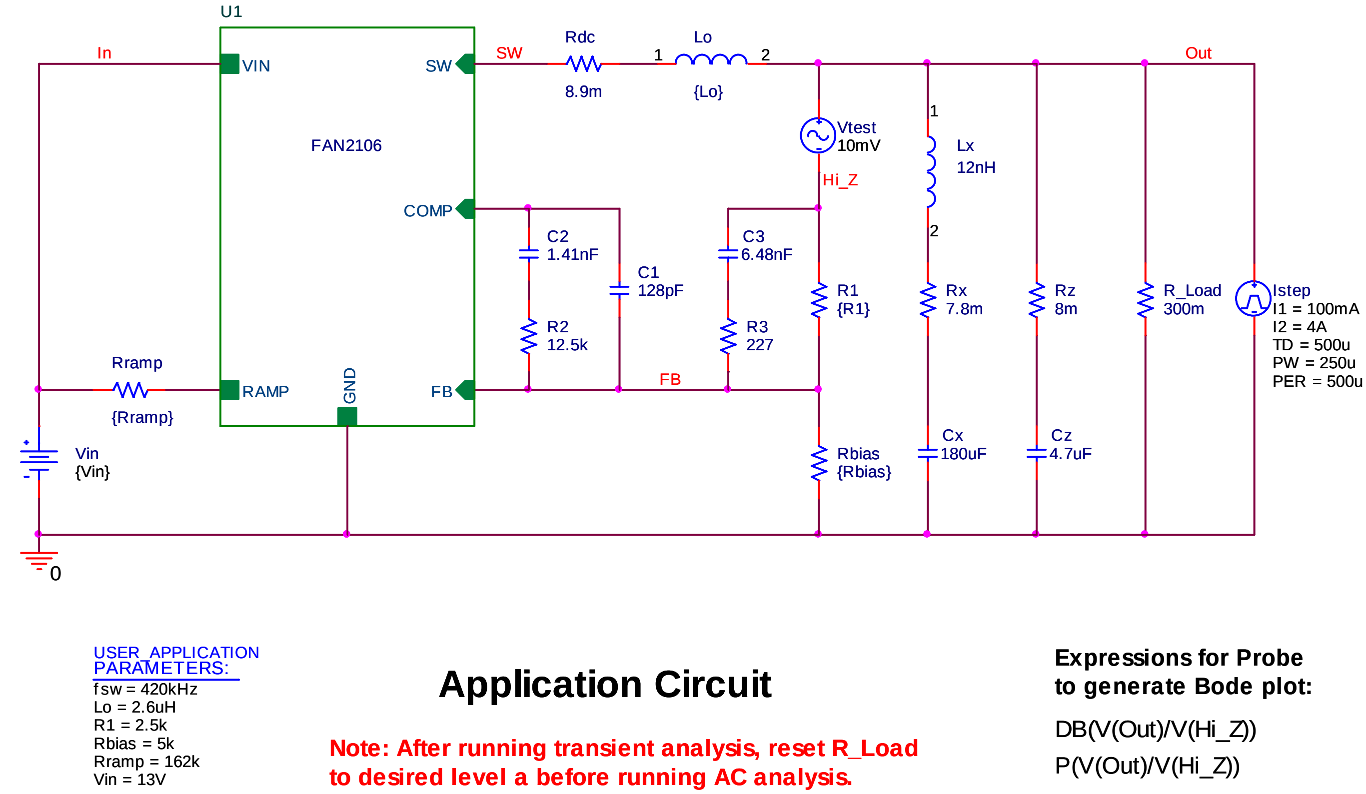

Figure 4. PSPICE schematic

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 signals displayed. If a simulation probe window has no trace, add a trace and input the expressions for the signals:

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 6.

The schematic contains the expressions for the Bode plot in the lower right-hand corner.

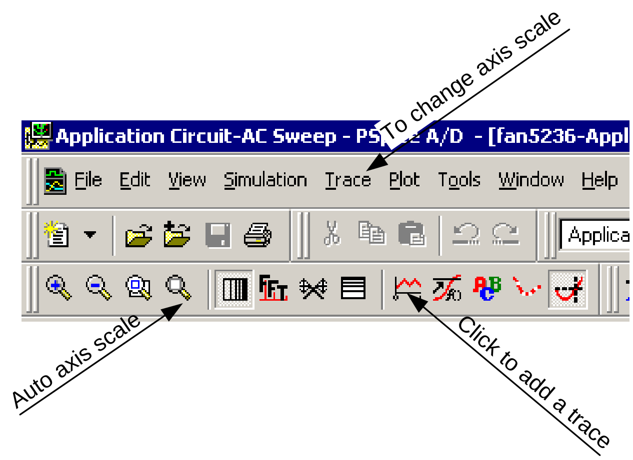

Figure 5. Adjusting the Probe Window Settings and Adding Traces

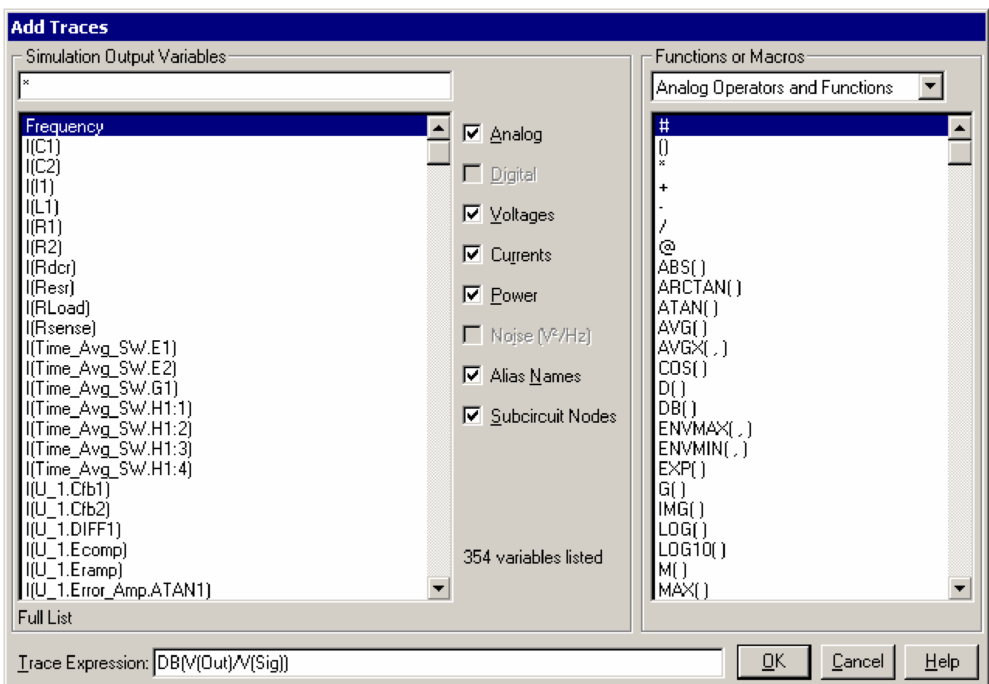

Figure 6. Adding a Trace (Gain Example)

References:

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