ON Semiconductor

Is Now

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

Is Now Part of

ON Semiconductor®

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

AN-6010
FAN5069 Component Calculation and Simulation Tools

Background / Overview

This app note describes design tools for FAN5069, which include:

- An Excel workbook to calculate recommended external component values
- A continuous-time behavioral model of the modulator that runs in PSPICE A/D v 9.1 or above. 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
  Note: These links have been verified as of this publication date, but may change over time.

These design aids can be requested from: https://www.fairchildsemi.com/design/design-tools/pwm-uldo-controller-combo/.

To install, copy AN-6010.ZIP to an empty folder (e.g. "FAN5069Design"), then unzip / extract AN-6010.ZIP into that folder.

This tool set applies to the following products:

FAN5069
FAN5069 Product Folder

Design Procedure

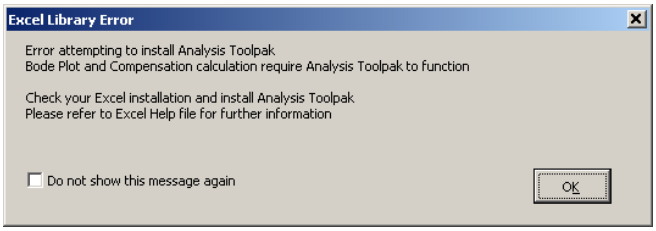
1. Use the spreadsheet (FAN5069 Design calculation aid.xls). The "**Instructions**" tab provides detailed instructions for the spreadsheet.
2. After completing the "**Component Selection**" tab, the regulator's small signal response can be viewed in the "Bode Plot" tab.
3. Verify transient and small signal stability using the PSPICE Simulation Model provided, using the components chosen in the "Component Selection" tab.

Note: There will be minor differences between the PSPICE and Spreadsheet bode plots. The PSPICE model has a more accurate representation of the IC.

Spreadsheet Start-up

The spreadsheet uses functions in Excel's Analysis ToolPak add-in to optimize compensation and installs it automatically if it is not already installed.

Some Excel installations may not have Analysis ToolPak included. If the following message appears, run Microsoft Office installation to make the Analysis ToolPak add-in available.



Consult your Microsoft Excel documentation for information should this error occur.

Macro Security Note

FAN5069 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]. The model provides a bode plot where the red trace is phase margin (in degrees) and the green trace is gain (in dB). The model also provides transient response using a pulsed current source (Istep) as the load. The IC's error-amp behavioral model is based on Ray Kendall's Macromodelling article in EDN.[iv]

1. To run the model, start Capture (9.1 or higher).
2. Open FAN5069 PSpice Avg Model.opj.
3. Double-click on Page 1 under: \FAN5069 pspice avg model.dsn\Application.

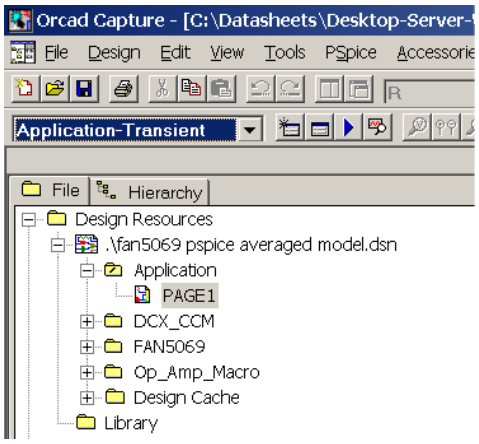


Figure 1. Project View

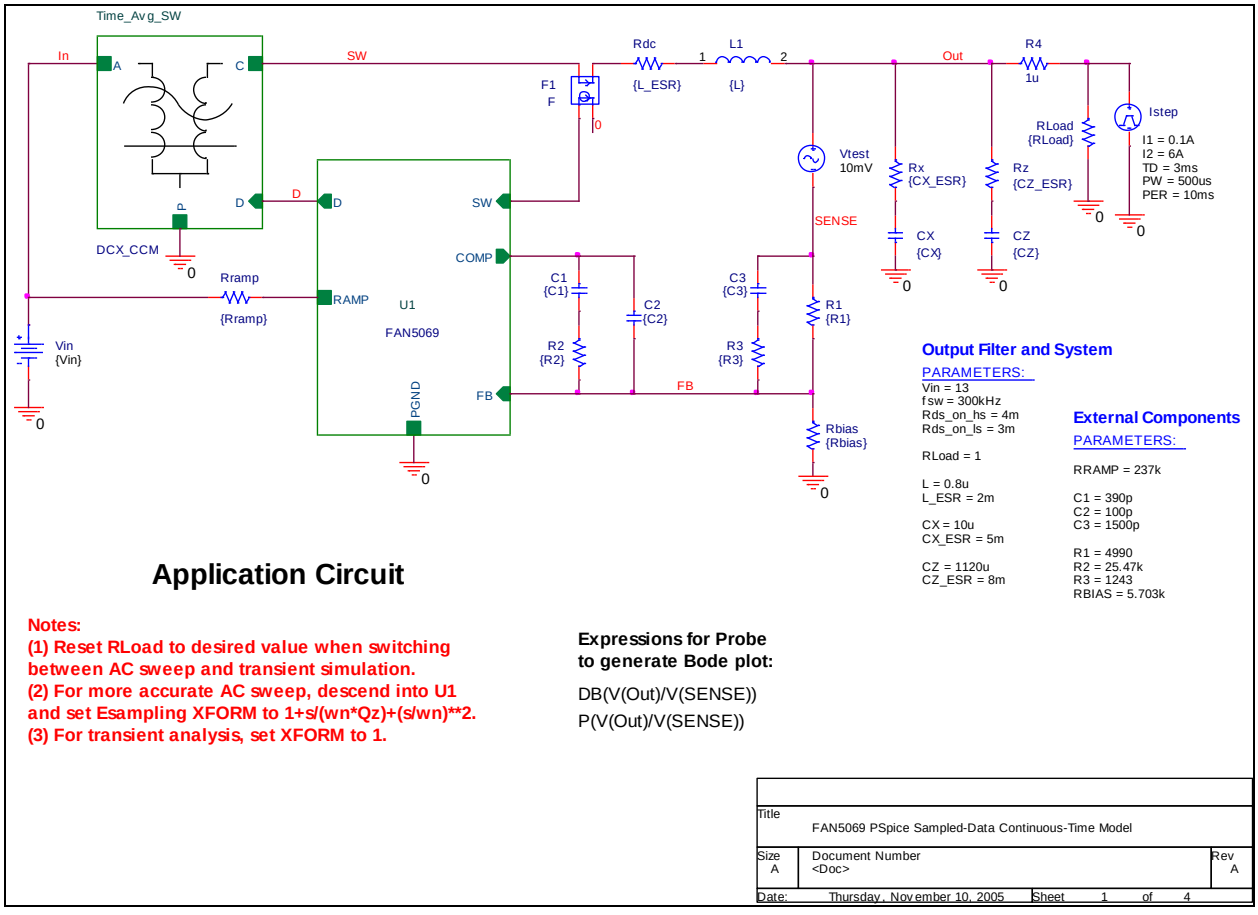


Figure 2. PSPICE behavioral model schematic for 1.5V output @ 300Khz

Note: The parameters for this model (from the results of the spreadsheet) are entered in the "Output Filter and System" and "External Components" parameter blocks on the lower left-hand corner of the schematic; there should be no need to edit the schematic itself.

4. Double click on any parameter in that block to set the values in the schematic.
5. Once the schematic is set up, press the F11 (function key) to display the bode plot.
6. To choose between bode plot (AC small signal) and transient response, select the simulation profile type from the drop-down box in the upper left corner.

Figure 3. Simulation profile select drop-down

Note: If you are simulating for transient response, be sure to set RLOAD appropriately. If you skip this step, the output and inductor current [I(L1)] traces may exceed the current limit of the IC. Example simulation outputs begin with Figure 8.

Note: When running transient response, be sure that the XFORM value in the ELAPLACE block inside U1 (the FAN5069 block) is set to 1. See detailed instructions below for setting this value.

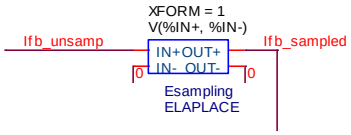


Figure 4. ELAPLACE block, XFORM parameter settings for Transient response

AC Sweep (Bode Plot) Tips

For maximum accuracy, define the Laplace block inside the FAN5069 sub-circuit.

1. Right click on U1 (FAN5069) and choose "Descend hierarchy."
2. Double-click the text block above the ELAPLACE block whose value is: 1+s/(wn*Qz)+(s/wn)**2.
3. Copy the entire text (CTRL-C) to the clipboard.
4. Close the text window, double-click the "XFORM" value, and paste the text from the clipboard into the value for XFORM (CTRL-V).

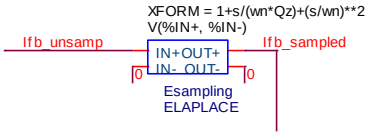
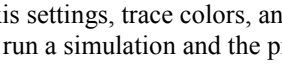
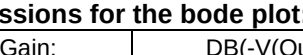


Figure 5. ELAPLACE block, XFORM parameter settings for Transient response

To obtain cross-over frequency and phase margin from the AC sweep plot (see Figure 8), turn on cursors (see Figure 6) then left-click the gain plot marker 

in the lower left corner of the graph) and right-click the phase plot marker 

, which establishes gain and phase as A1 and A2 respectively in the cursor window. Left-click, then right-click where gain crosses 0. A1 and A2 X values show the gain cross-over frequency and the A2 Y value displays phase margin.

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 you run a simulation and the probe window displays no traces, then click the "Add Traces" button as shown in Figure 6, and paste the following expressions for the signals.

Expressions for the bode plot:

Gain:	DB(-V(Out)/V(Sense))
Phase:	P(V(Out)/V(Sense))

The schematic also contains the expressions for the Bode plot in the "Bode Plot Instructions" in the schematic window, as shown in Figure 2.

Expressions for Transient response:

Output Voltage:	V(Out)
Load Current:	I(R4)
Inductor Current:	I(L1)

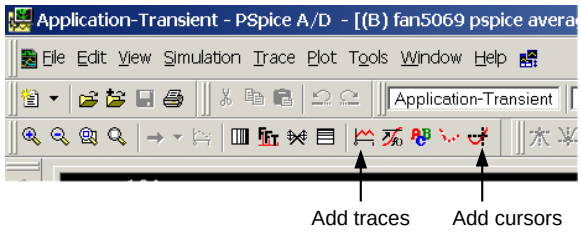


Figure 6. Probe window tips

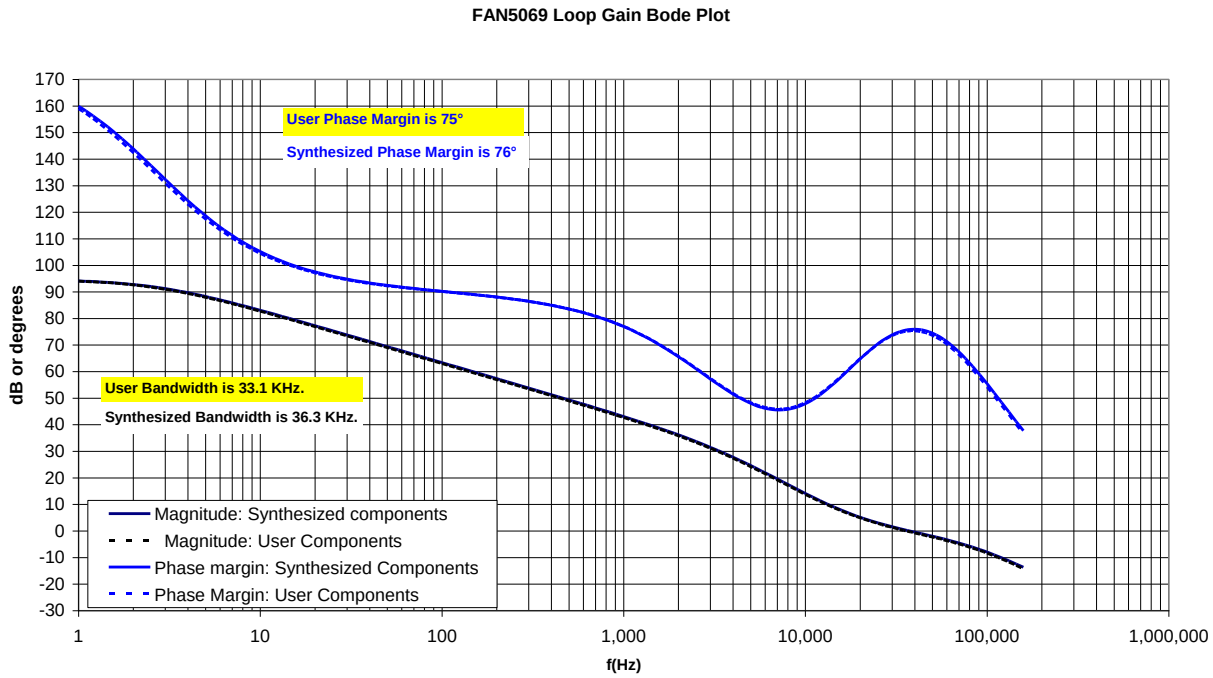


Figure 7. Bode plot from spreadsheet using Ckt values per Figure 2

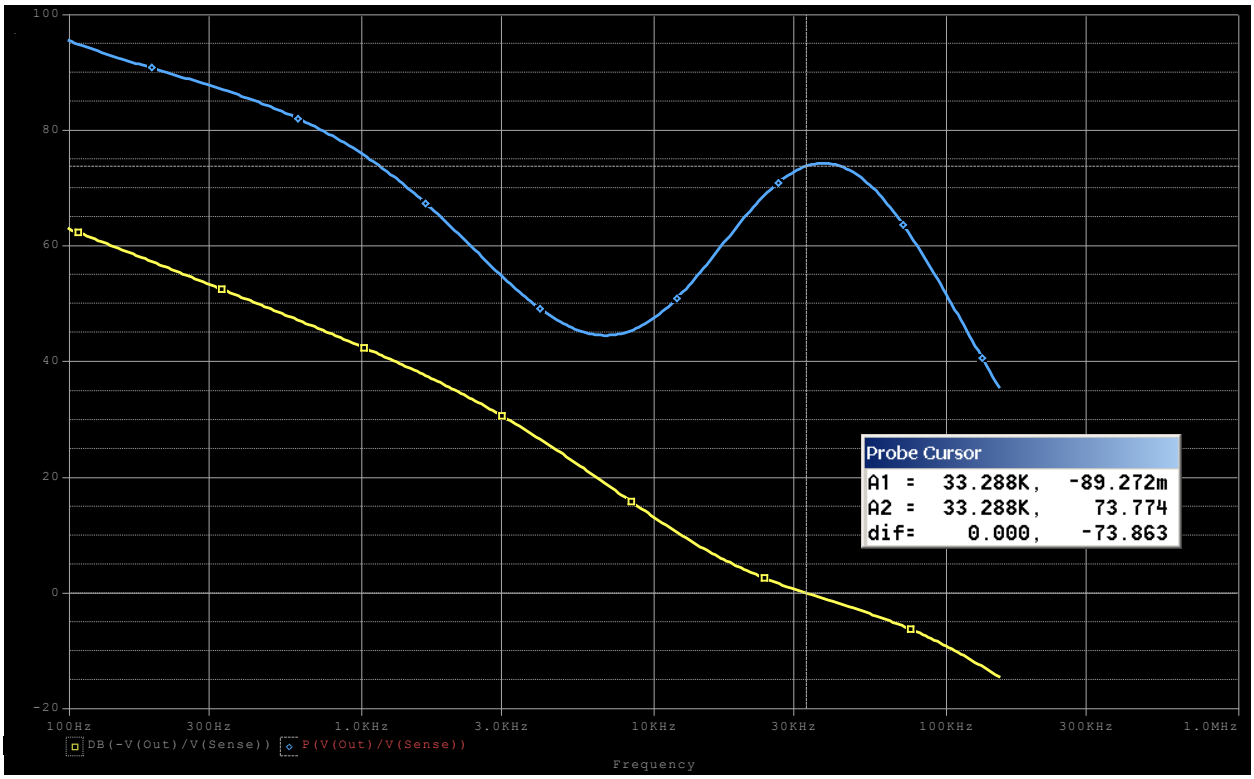


Figure 8. Bode plot response simulation result using Ckt values per Figure 2
For probe window, A1 X value is the cross-over frequency and A2 Y value is phase margin

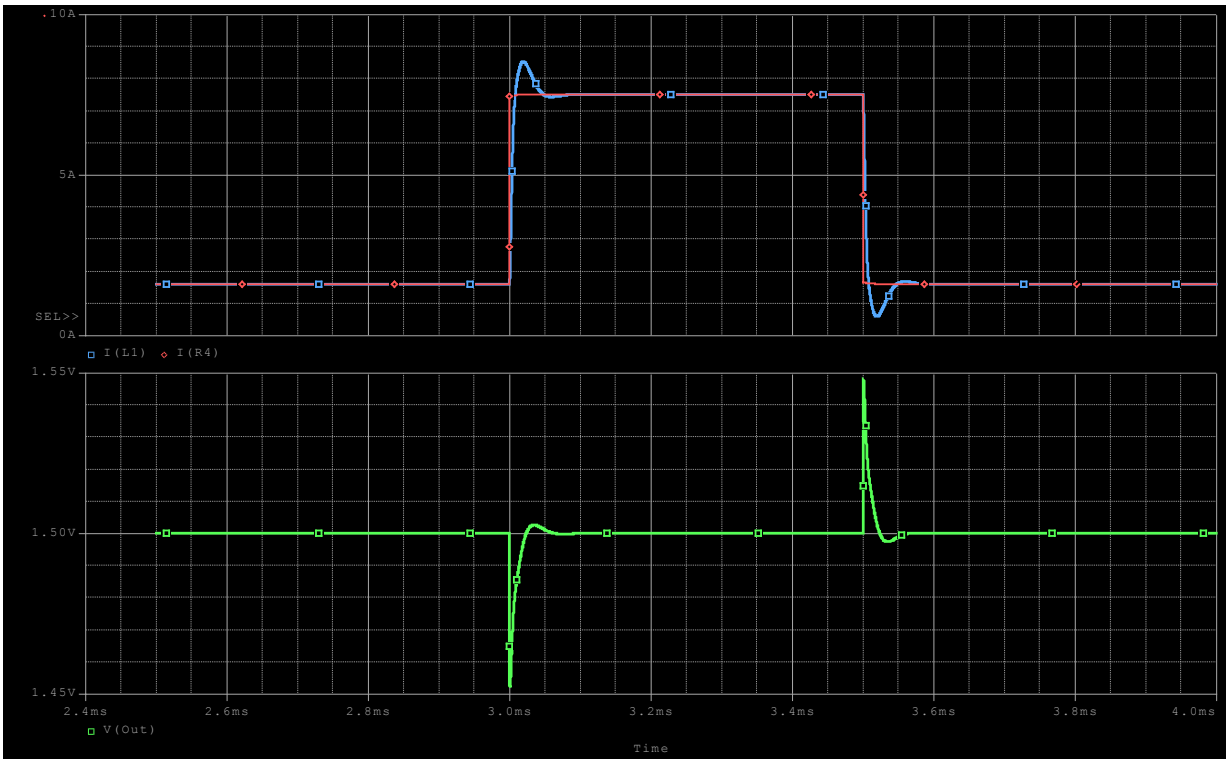


Figure 9. Transient response simulation result using Ckt values per Figure 2

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

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