Working with Power Supply Templates

www.intusoft.com

Getting Started: After installing the ICAP/4 demo software, you'll find a directory called "Circuits" in the folder you chose for your installation target. Within that folder; the following structure for power electronics circuits can be found:

- Circuits
  - Power
    - Models
      - FLYBACK_ID.DWG
      - FORWARD_ID.DWG
  - PwrDemo
    - ForwardTemplates
      - FwdTemplateDemo.dwg
    - FlybackTemplates
      - FlyBkTemplateDemo.dwg
    - PushPullTemplate
      - PushPullTemplateDemo.dwg

FLYBACK_ID.dwg and FORWARD_ID.dwg are the source for the Z Transform Average models. You can't run these standalone using the demo. With non-demo software you can access them as hierarchical drawings using the Avg configuration in the PwrDemo folder drawings.

We currently provide you with 3 working power supply templates. Forward and Flyback templates converts 28 VDC to 5VDC; one uses a transformer coupled Flyback topology and the other is a transformer coupled Forward topology. Each of these drawings used the average models in FLYBACK_ID.dwg and FORWARD_ID.dwg. The push pull template converts 42 volts to 12 volts at 100 Amps. These models are collapsed into subcircuits for use in the template drawings.

The template drawings make heavy use of subcircuits in order to extend the size of the circuit that can be run using our demo software. You can vary parameters in the subcircuits; however, these circuits are near the capacity of the demo version so that you are limited in making additions and changes. We will bring you additional topologies and vendor models that will work within the demo limits. Remember, Intusoft is providing the demo free in order to interest you in purchasing our software programs. The templates are quite useful in demo form; but the real thing is so much better! You can simulate both average and switched transient subcircuit (avgSubckt and TranSubckt) configurations using the demo software.

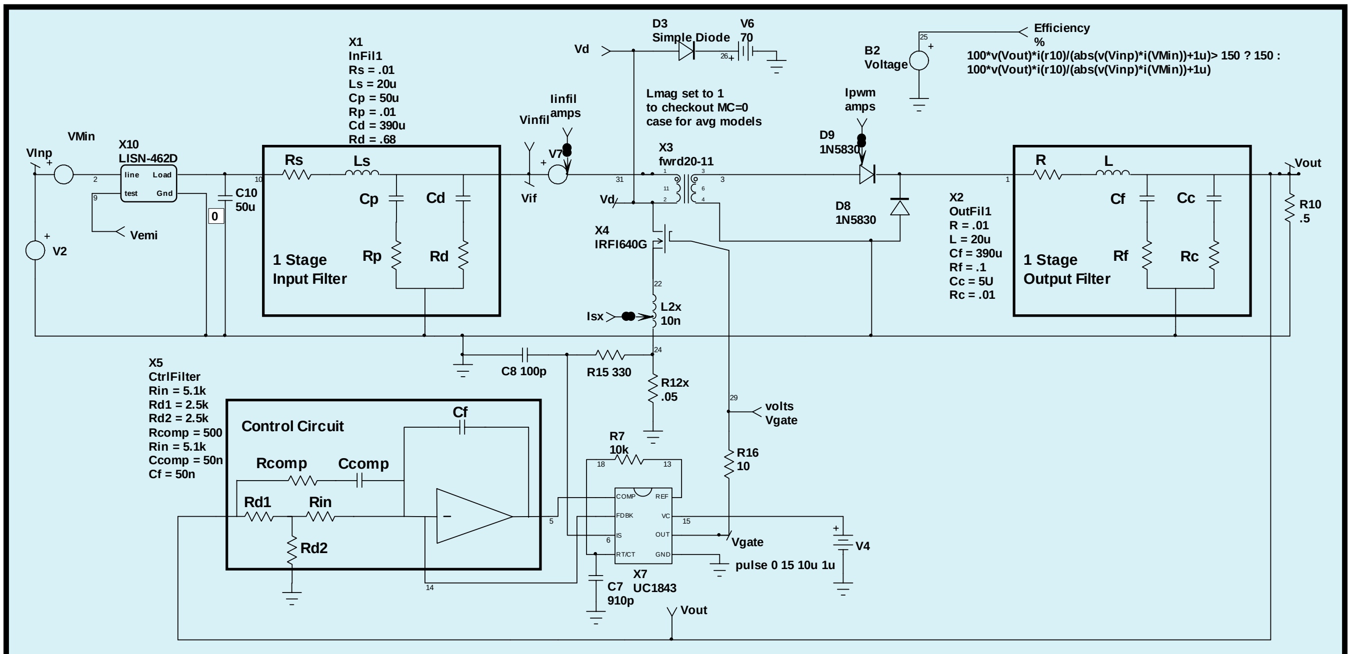

Figure 1: FwdTemplateDemo.dwg, TranSubckt Configuration

Open FwdTemplateDemo.dwg by navigating to it and double clicking. Once the drawing is opened, there are 2 menu boxes in your toolbar. The left shows the drawing configuration and the right is the simulation setup. Use the arrow on the right to open the menu and choose an item. The button to the left lets you edit the contents of the configuration or setup respectively. This combination of configuration and setup defines the circuit and method for its simulation. Choose <AvgSubck> and <TranAvg 2ms> and press the running man icon in the toolbar to start the simulation. This drawing has 6 layers that define 3 configurations. The Subckts and common layers are shared. tran, avg, avgS and avgH are used to differentiate the functionality. The TranSubckt uses switching models to capture the cycle-by-cycle switching behavior. It's used to measure stress on switching elements. The AvgSubckt configuration contains the average PWM models and is used for Bode plots and transient optimization. The Avg configuration is used for model development, letting you cross probe into the model.

Layers/ Configurations	Subckts	tran	common	avg	avgS	avgH
TranSubckt	x	x	x			
AvgSubckt	x		x	x	x	
Avg	x		x	x		x

Table 1: Illustration of combining layers into configurations

Running the Optimizer

Optimizing control components in the time domain: The goal of today's control loop design for power supplies is just to keep them from oscillating. With a reasonably good AC model, getting ample gain and phase margins is the goal. It's like being happy that you're car doesn't tip over going around a corner. But with the ICAP/4 optimizer (included in the demo), you can get around the corner really fast and stay on the road. Here's how it's done using the demo you just opened. Double click on the Control Circuit block. Choose the Tolerance/Sweep/Optimize tab. In the Optimize column, scroll to the bottom and enter 80% in the Ccomp and Cf column. That sets the boundary for the optimizer to +- 80% of the value set in the Label tab.

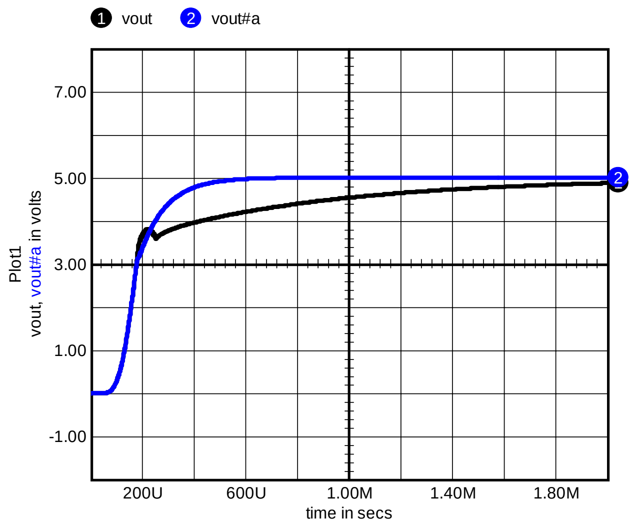

Figure 2: Waveform 1, before optimization, 2 afterward

The objective function sets the goal: Optimizers work by minimizing an objective function. The objective function is usually made into the weighted sum of squares of different criteria. Open the Actions\ICAPs menu to get the Simulation Control dialog. Select the Measurements tab and choose AvgSubckt-tranAvg 2m-TRAN-ofunc. Press the <Edit Test Group> to see how we made the objective function. There are 2 parts to the objective function. First is to get to an accurate final condition after 400 usec; tending to increase bandwidth. Next is to penalize overshoot after 200 usec; which will tend to lower bandwidth in favor of stability.

Running the simulation: Close the "Edit Test Group" dialog and select the "Main" tab of the Simulation Control Dialog. Then select the "AvgSubckt + tranAvg 2m" Test configuration. Then choose OPTIMIZE in the Mode group. The "Data Reduction" group should already be correctly set to "Interactive", "Script". Then press "Simulate Selections". IsSpice will then process the optimization. The algorithm used in the OPTIMIZE script varies each parameter over its range, and measures its objective function. The parameter and the values of the objective function are shown during the simulations. You can check and make sure a minimum occurs somewhere in between the end points.

You can read about how it works by pressing the "?" icon in the Simulation Control Dialog.

Take a look at the before and after results: Before performing the optimization, a reference simulation was run. The results were stored in the "ref" plot. After the optimization finished, a final run was made in the last tran plot. You can see before and after results by plotting the ref.vout and tran18.vout vectors.

Gain and Phase Margins: To see what happened to gain and phase margins, first record the optimizer results, then run an "AvgSubckt + ac" analysis in the Standard mode. Turn on Scope5, open a new graph and press "b" to get a Bode plot (Hot keys are case sensitive). Then copy the optimizer values into the "Control Circuit" block and run the ac analysis again. Go to Scope and open a new graph and press "b" again to get the new Bode plot. You can also go back the original bode plot and press <Ctrl + u> to do an update. Note: the Bode, "b", script doesn't run when updating, it just draws the plots.

The transient response is better and the gain and phase margins improved, and that's not bad for a few minutes work! Now for a real design, you need to consider all of the environments for line and load and test configurations. You can see

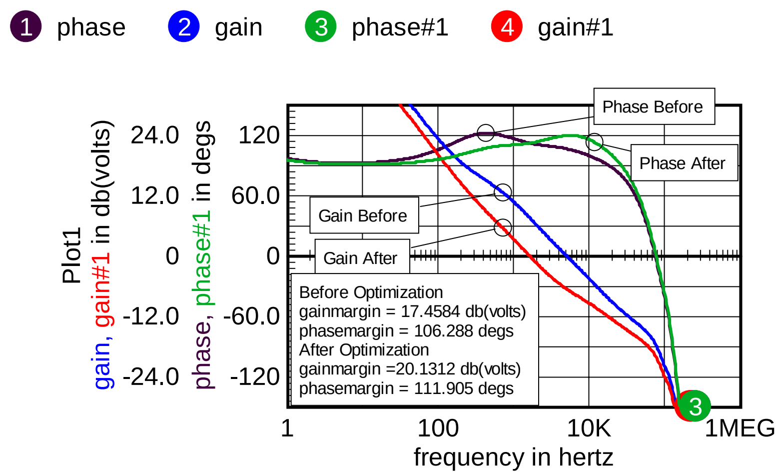

Figure 3: Bode plot shows improved margins

here, that we optimized with the LISN in place because we don't want to blow out the loads for an embedded power supply while running an EMI test. You may also want to run with different input conditions as well as different loads to pick the "best" design.

EMI compliance testing: Next, run the "TranSubckt - tran emi" to see how well the design complies with the conducted emission specification. Run the !User\emi script to see the following result.

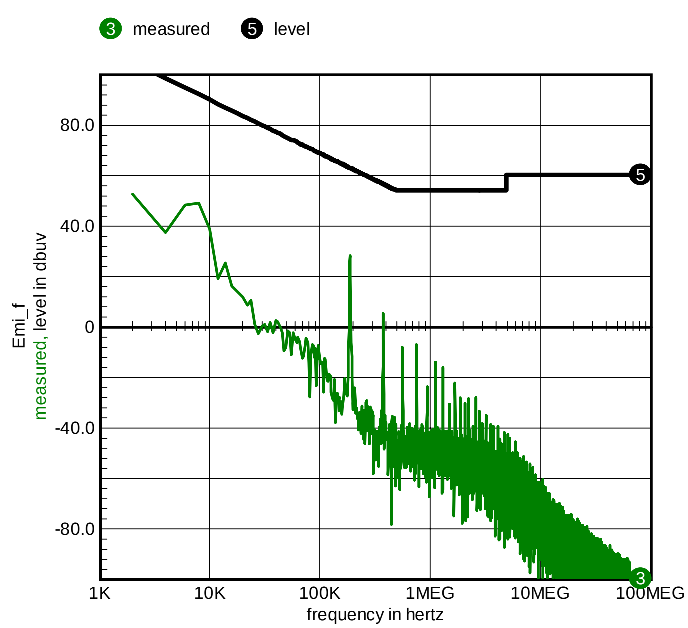

Figure 4: Conducted emissions are well under control