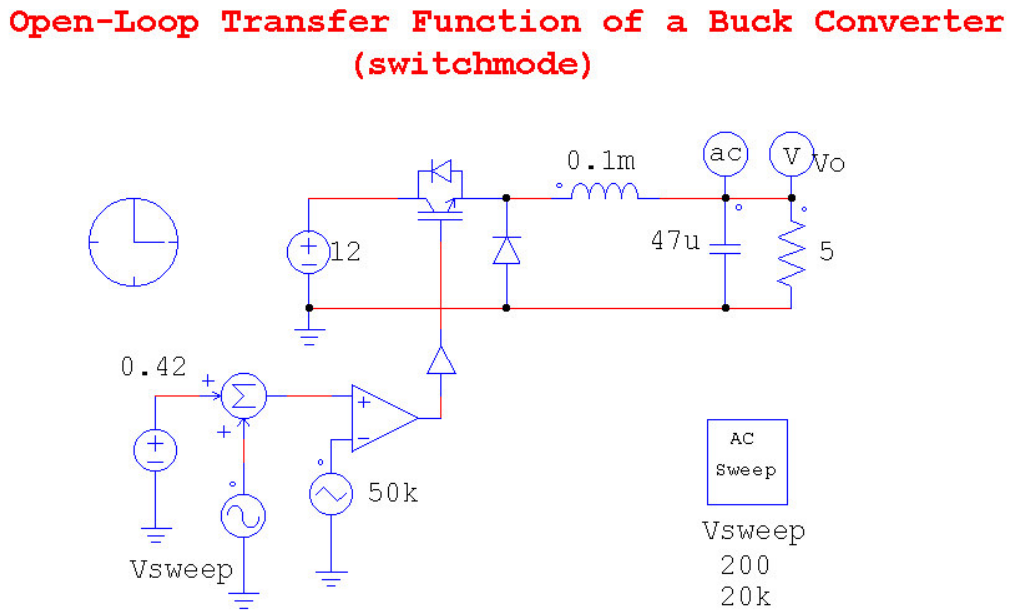
PSIM SIMULATION SOFTWARE

TUTORIAL
How to use the AC SWEEP

POWERSYS – Les Grandes Terres – 13650 Meyrargues -Tel : +33 (0)4 42 63 60 88 – Fax : +33 (0)4 42 63 61 19
www.powersys.fr - info@powersys.fr

The objective of this tutorial is to help you use the AC Sweep function to realise a frequency response analysis.

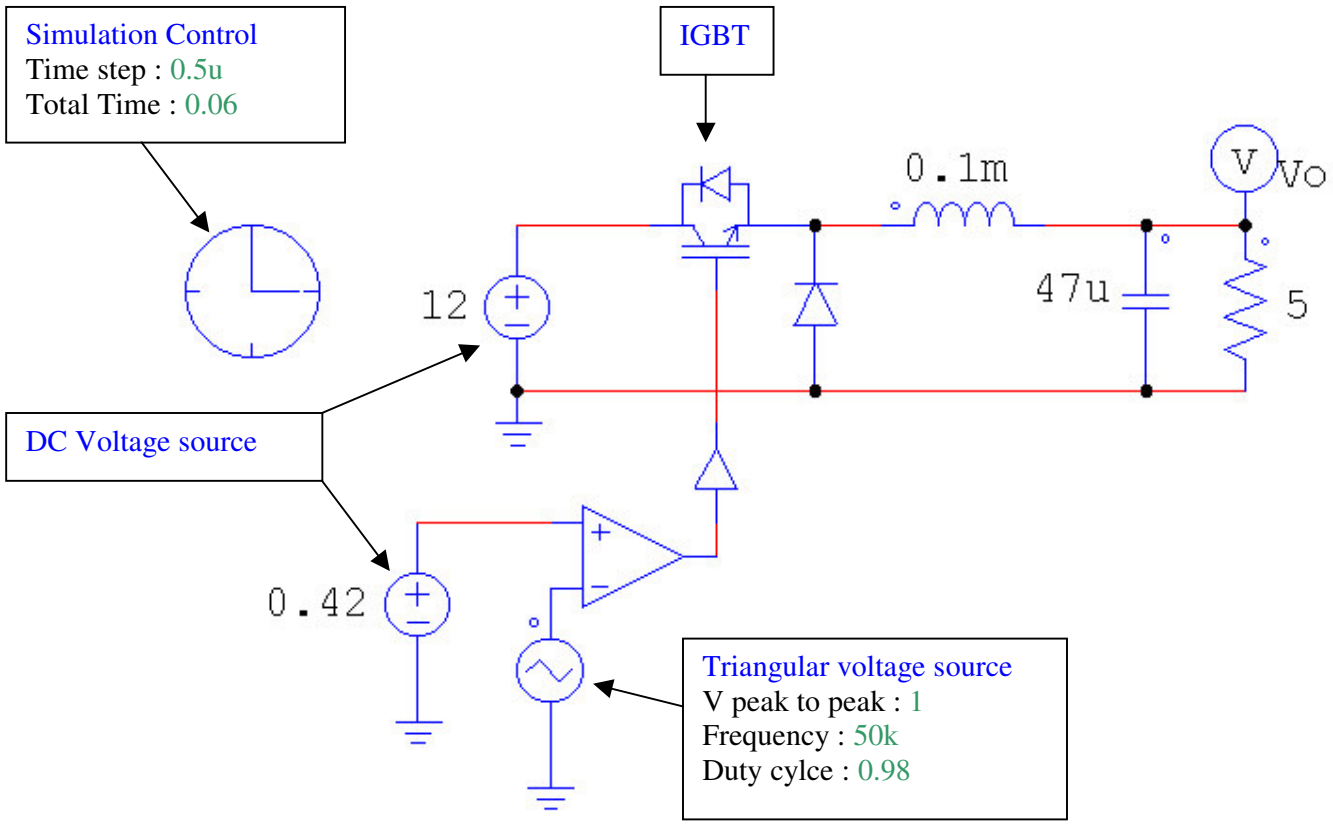
The example circuit is the following (this example can be run with the demo version) :



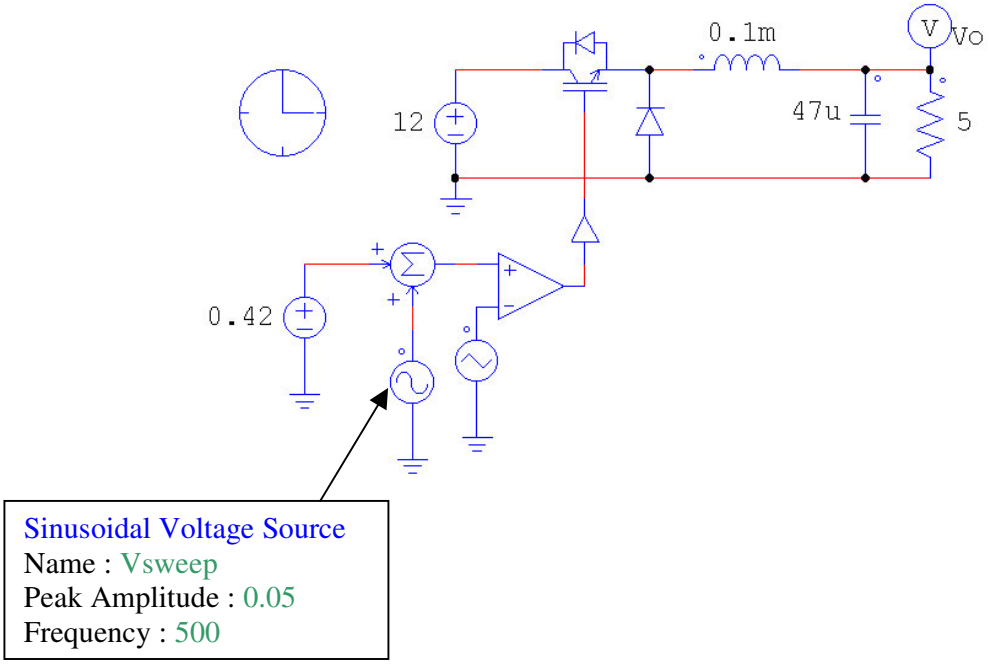
The principle of the AC analysis is that a small AC excitation signal is injected into the system as perturbation and the signal, at the same frequency, is extracted at the output.

Hereafter are the different steps to use the AC Sweep :

1. Open your PSIM version.
2. Open a new file
3. Save the file with the name “Tutorial AC Sweep.sch”
4. Create the following circuit :

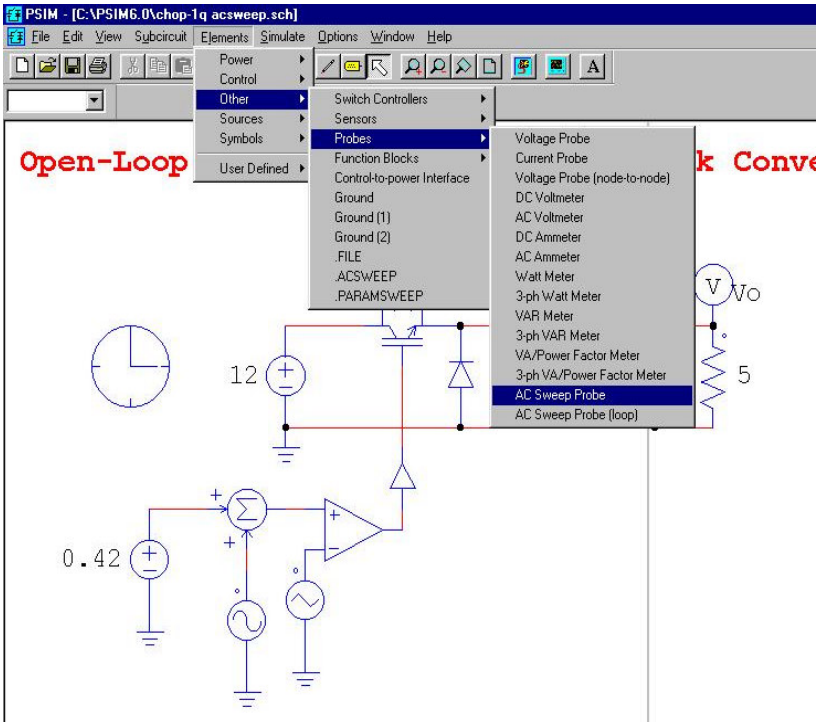


5. Place a sinusoidal source (VSIN) as the excitation source for the AC sweep after the DC voltage source of 0.42V.

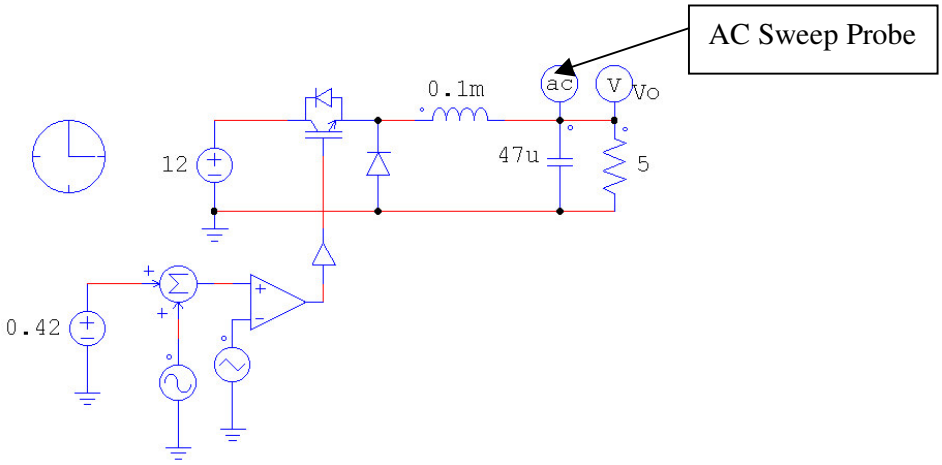


Sinusoidal Voltage Source
Name : Vsweep
Peak Amplitude : 0.05
Frequency : 500

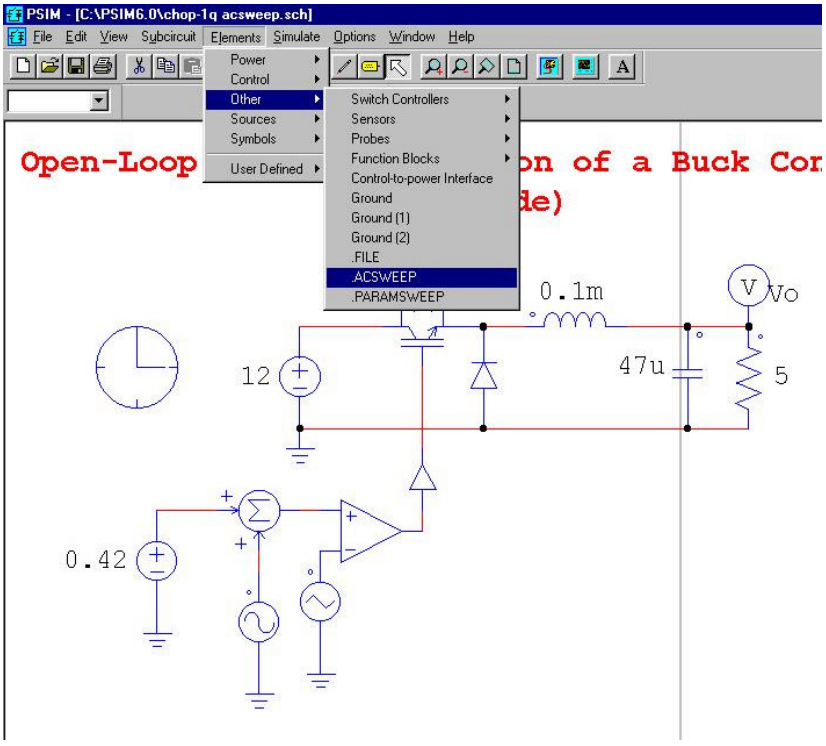
6. Go to the Menu "Elements/Other/Probes/"



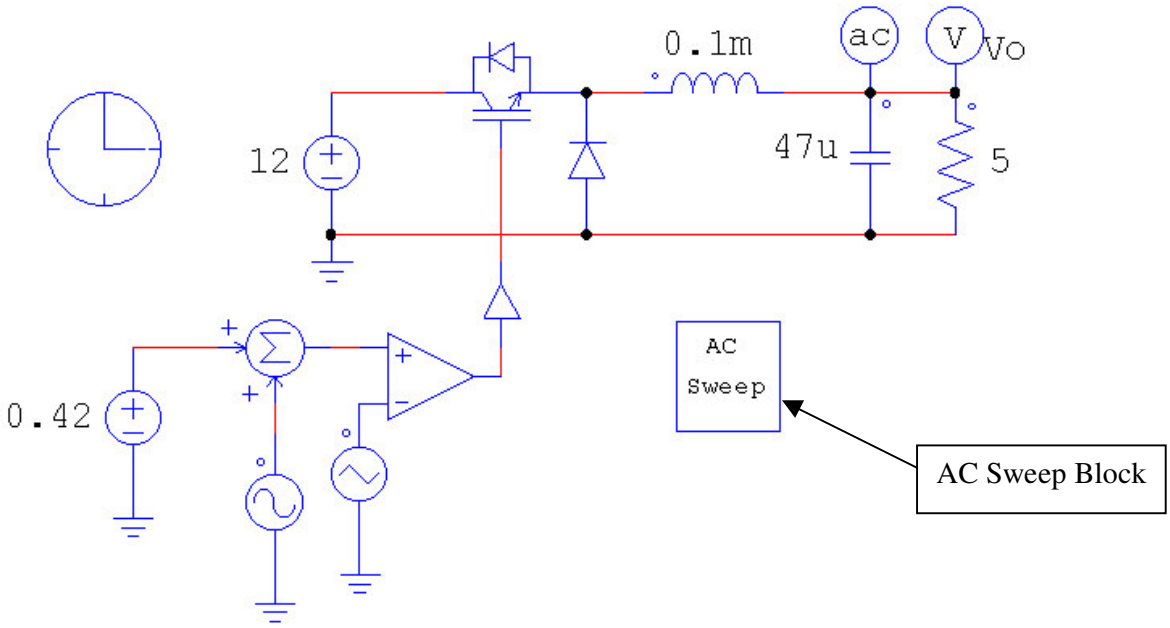
Place the AC Sweep Probe (ACSWEEP_OUT) at the desired output location :



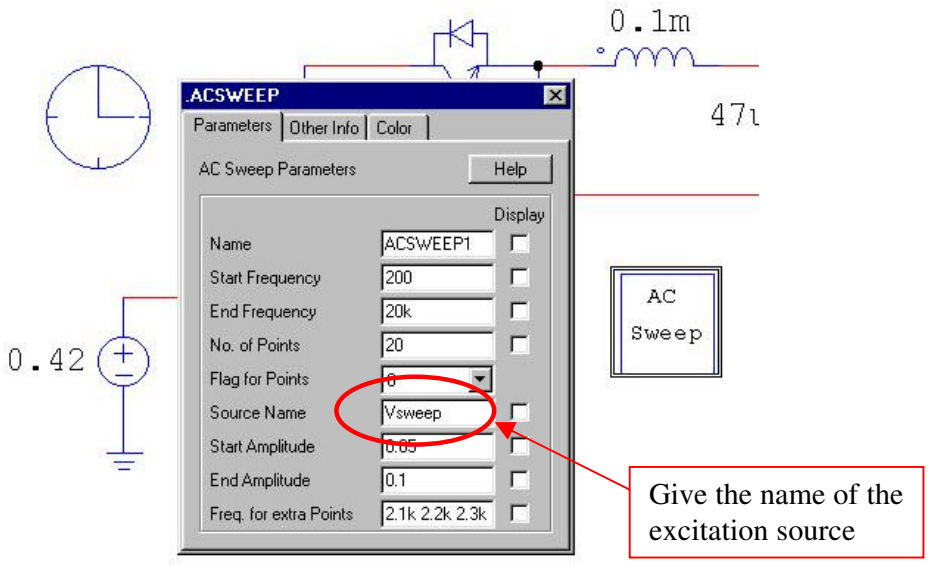
7. Go to the Menu “Elements/Other/”



Place the AC Sweep block on the circuit :



8. Parameter the AC Sweep block :



The parameters are defined hereunder :

- Start Frequency : Start frequency of the AC sweep, in Hz
- End Frequency : End frequency of the AC sweep, in Hz
- No. of Points : Number of data points
- Flag for Points : Flag to define how the data point is generated. Flag = 0: Points are distributed linearly in LOG10 scale. Flag = 1: Points are distributed linearly in linear scale
- Source Name : Name of the excitation source
- Start Amplitude : Excitation source amplitude at the start frequency
- End Amplitude : Excitation source amplitude at the end frequency
- Freq. for extra Points : Frequencies of additional data points. If the frequency-domain characteristics change rapidly at a certain frequency range, one can add extra points in this region to obtain better data resolution.

9. The setup to use the AC Sweep is now complete. You can run the simulation.

Additional remarks :

- With the AC analysis, the frequency response of a circuit or a control loop can be obtained. A key feature of the AC analysis in PSIM is that, if a circuit is in a switchmode, it can be in its original switchmode form, and no average model is required. Nevertheless, with the average model, the time it takes to perform the AC analysis will be shorter.

- To obtain accurate AC analysis results, the excitation source amplitude must be set properly. The amplitude must be small enough so that the perturbation stays in the linear region. On the other hand, the excitation source amplitude must be large enough so that the output signal is not affected by numerical errors. In general, a physical system has low attenuation in the low frequency range and high attenuation in the high frequency range. A good selection of the excitation source amplitude would be to have a relatively small amplitude at the start frequency, and a relatively large amplitude at the end frequency.

- Sometimes, after AC analysis is complete, a warning message is displayed as follows: *"Warning: The program did not reach the steady state after 60 cycles. See File "message.doc" for more details."*. This message occurs when the software fails to detect the steady state at the AC sweep output after 60 cycles. To address this problem, one may increase damping in the circuit (by including parasitic resistances, for example), or adjust the excitation source amplitude, or reduce the simulation time step. The file "message.doc" gives the information on the frequency at which this occurs and the relative error. The relative error will indicate how far the data point is from reaching the steady state