Title: Using AutoConvergence

Product: OrCAD PSpice A/D, OrCAD PSpice AA and AMS Simulator

Summary: The convergence problem will be described briefly in this application note and the AutoConvergence feature of PSpice will be introduced.

Author/Date: Wei Ling / 24.8.2010
Update/Date: Pascal Willems / 20.9.2013

Table of Contents

1 Introduction

In order to calculate the bias point, DC sweep and transient analysis for analog devices, PSpice must solve a set of nonlinear equations which describe the circuit's behaviour. This is accomplished by using an iterative technique, the Newton-Raphson algorithm, which starts by having an initial approximation to the solution and iteratively improves it until successive voltages and currents converge to the same result.

The Newton-Raphson method can be described as follows.
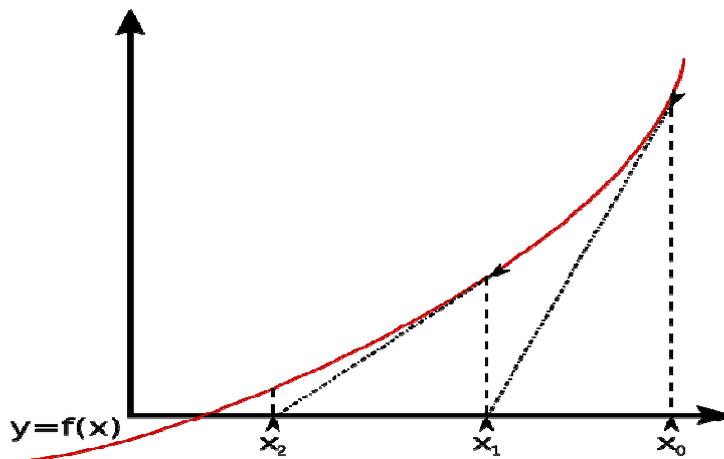
We want to find successively better approximations to the roots of a function $f(x)$. With the first guess x_0 we begin to find a better approximation x_1

$$x_1 = x_0 - \frac{f(x_0)}{f'(x_0)} \qquad (1)$$

Where $f'(x_0)$ is the derivative of the function at x_0

Further we have

$$x_{n+1} = x_n - \frac{f(x_n)}{f'(x_n)} \qquad (2)$$

In order to guarantee the convergence of a solution, some conditions must be considered.

- The nonlinear equations must have a solution
- The equations must be continuous
- The algorithm needs the equations' derivatives
- The initial approximation must be close enough to the solution.

Each of these can be taken in order. We should be aware that the PSpice algorithms are used in computer hardware that has finite precision and finite dynamic range with these limits:

- Voltages and currents in PSpice are limited to +/-1e10 volts and amps
- Derivatives in PSpice are limited to 1e14
- The arithmetic used in PSpice is double precision and has 15 digits of accuracy

In a few cases PSpice cannot find a solution to the nonlinear circuit equations. This is generally called a "convergence problem" because the symptom is that the Newton-Raphson repeating series cannot converge onto a consistent set of voltages and currents.

If you use the transient analysis, and it is the case that the voltage or the current in the circuit moves too fast, it may be unable to continue because the time step required becomes too small.

Normally if you face the convergence problem in PSpice, you have to change the runtime parameters to relax the limits. With the help of the following demo-circuit we will discuss how you can try to solve the convergence problem manually and how to use the AutoConvergence feature of PSpice to converge the simulation automatically.

2 Demo Project

The example circuit we would like to build up is a DC-DC converter. This circuit in the project AutoConverge.opj is created as follows:
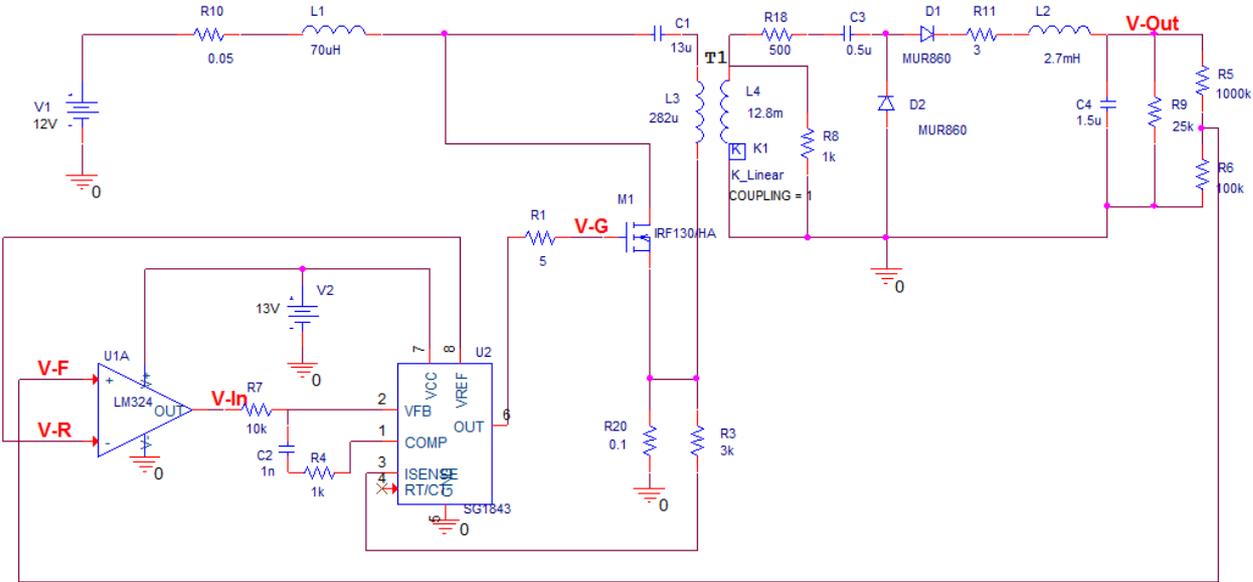
The N-Channel Power MOSFET M1 switches on and off, the energy from DC source V1 can be saved in capacitors and inductors. The AC voltage of L3 will be transformed to the secondary side of the transformer T1. The diodes D1 and D2 rectify the AC voltage and the capacitor C4 will be charged with the rectified voltage. The amplifier LM324 and the PWM controller SG1843 are responsible for the control of M1. The feedback voltage V-F will be compared with the reference voltage V-R and the result V-In will determine the gate voltage V-G of M1.

3 Convergence Problem

We can, for example, simply run the transient analysis for 10ms. However, the simulation will abort and the **PSpice Runtime Settings** window comes up.
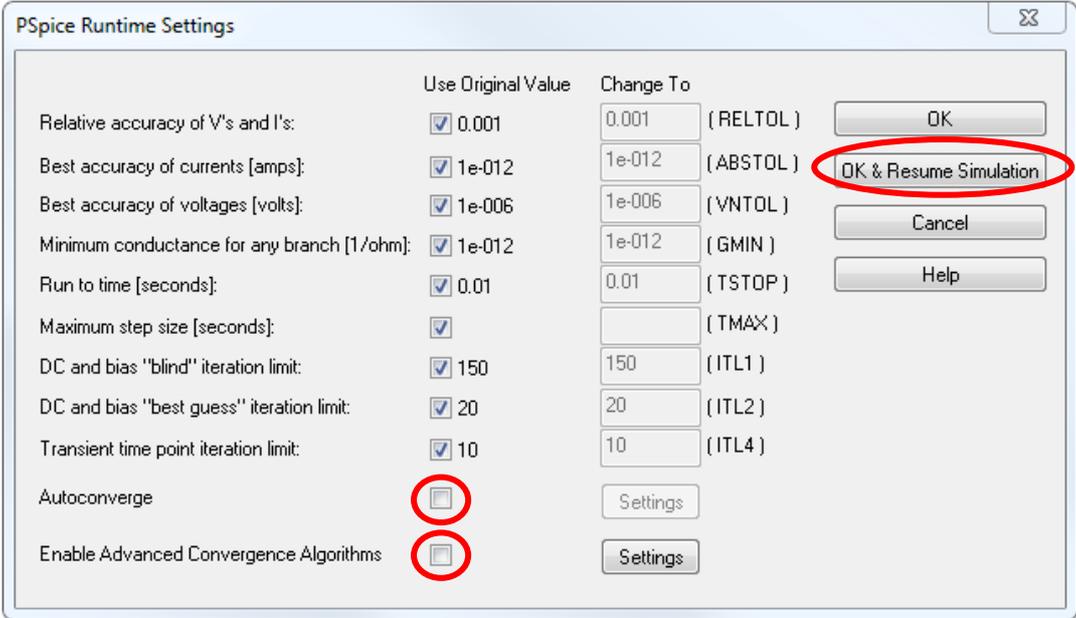
If you look at the PSpice output file, you can see an error is reported as follows.

ERROR(ORPSIM-15138): Convergence problem in transient analysis at Time = 128.3E-15.
Time step = 39.71E-21, minimum allowable step size = 1.000E-18

You can check **Autoconvergence** and **Enable Advanced Convergence Algorithms** and click on **[OK & Resume Simulation]**
The simulation will continue where it stopped and readjust automatically the Runtime settings.

If you don't want to use the Autoconvergence, you can readjust the Runtime Settings manually. PSpice uses by default the original value of the runtime parameter. The following table describes these parameters.

RELTOL	Relative tolerance of voltage and current
ABSTOL	Absolute current tolerance
VNTOL	Absolute voltage tolerance
GMIN	Minimum conductance for any branch
TSTOP	Run to time, for the demo circuit it is set to 10ms
TMAX	Maximum step size, if you don't define it in Simulation Profile, it is by default set to: TSTOP/50

ITL1	Limit of iterations of convergence calculation during bias point analysis
ITL2	Limit of iterations during DC analysis
ITL4	Limit of iterations at any repeating point in transient analysis

The hardest part of the whole process is getting started, that is, finding the bias point.
Therefore we will then run the bias point simulation. In the PSpice Run Time Setting click
[OK] or **[Cancel]** to close this window.
In **Simulation Settings** window select the **Bias Point** as Analysis type.
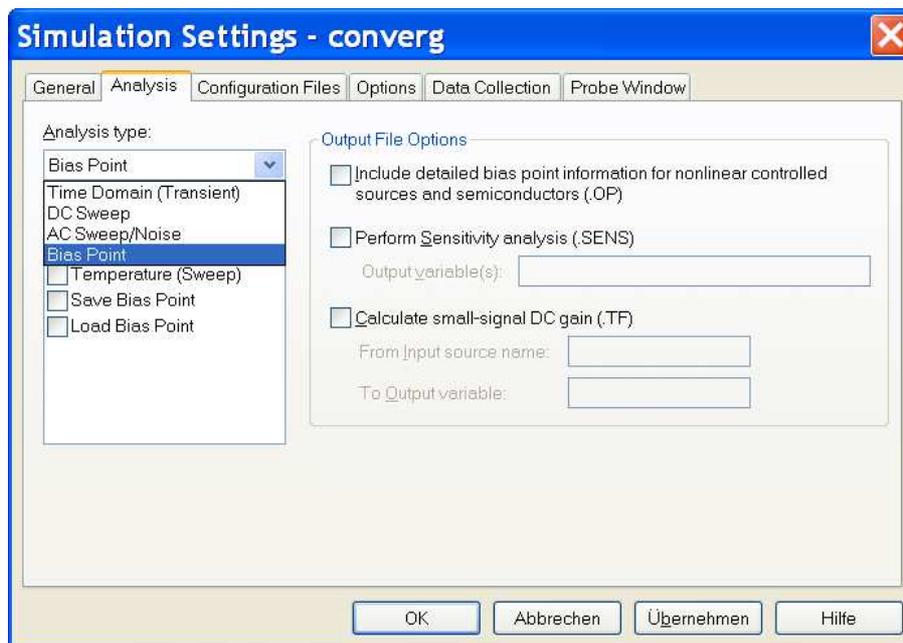
Click **OK** and run the bias point simulation.

The bias point simulation runs without any convergence problems. Once a bias point is
found, it moves on to run the transient analysis. It starts from a know solution (bias point) and
steps forward in time.

Changing back to run the transient simulation and the simulation will abort again. In order to
resolve the convergence problem, we will try to relax the limits for some runtime parameters.
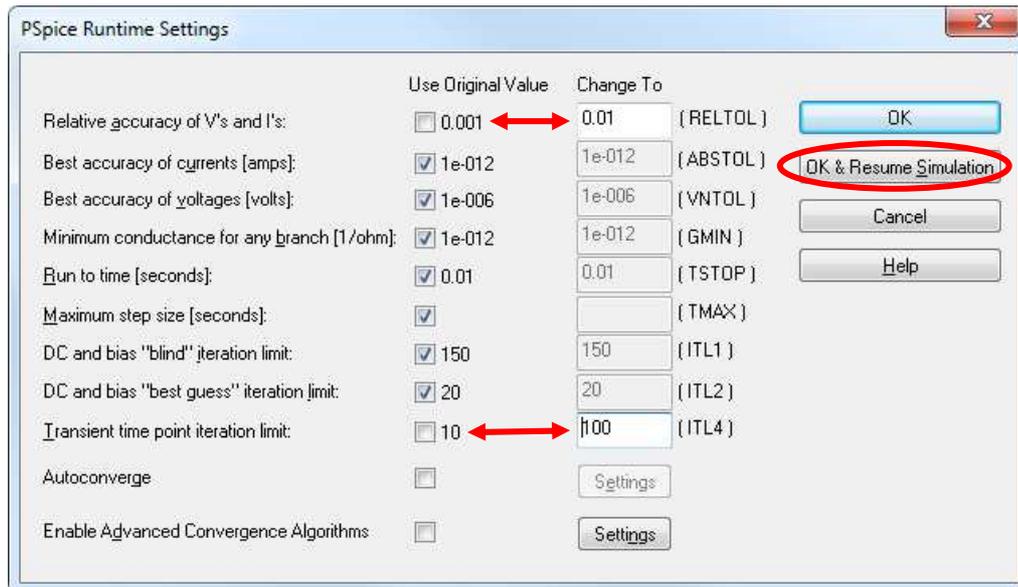
In the **PSpice Runtime Settings** dialogue window, uncheck the **Use Original Value** check box for **RELTOL** and set its value to 0.01.

Uncheck the **Use Original Value** check box for **ITL4** and set its value to 100. This increase the number of transient iterations that PSpice will attempt at each time point before it gives up.

Note:
You cannot specify a value that is less relaxed than the normal limit. For example, if the limit for ITL1 is set to 150, you cannot specify 120 as the relaxed limit.

Click **OK & Resume Simulation** to continue the simulation from where it left off. However we still have the convergence error. Change the runtime parameters with the relaxed limits as follows:
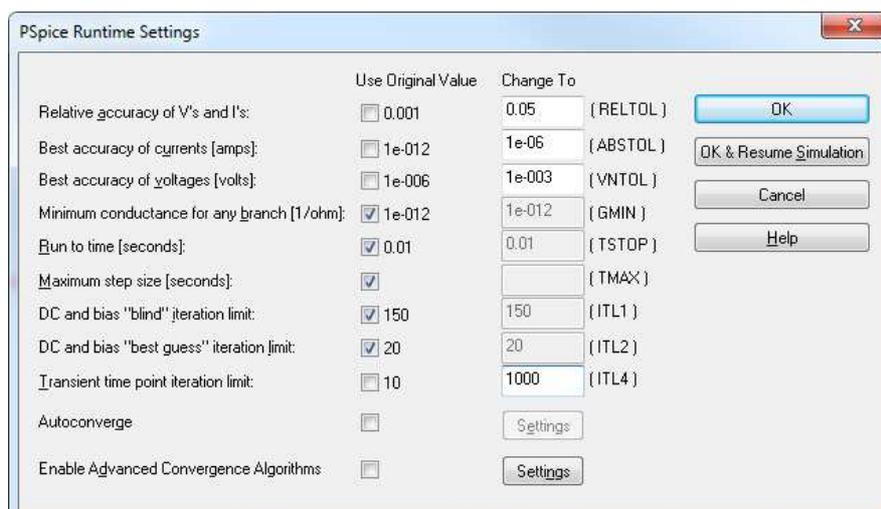
Click **OK & Resume Simulation** to continue the simulation. This time PSpice runs without any convergence problems.

Although the convergence problem is resolved, sometimes it could be very time consuming and you need some understanding of the runtime parameters. Next you will see how you can set Autoconvergence feature of PSpice before you start the simulation.

4 Using AutoConvergence

When you run the simulation with AutoConvergence, PSpice initially runs using the original values for the specified simulation time. However, if the simulation does not converge, PSpice changes the values within the relaxed limit for the run time parameters selected in the **AutoConvergence Options** dialogue box.

Open the simulation profile and click the **Options** tab.
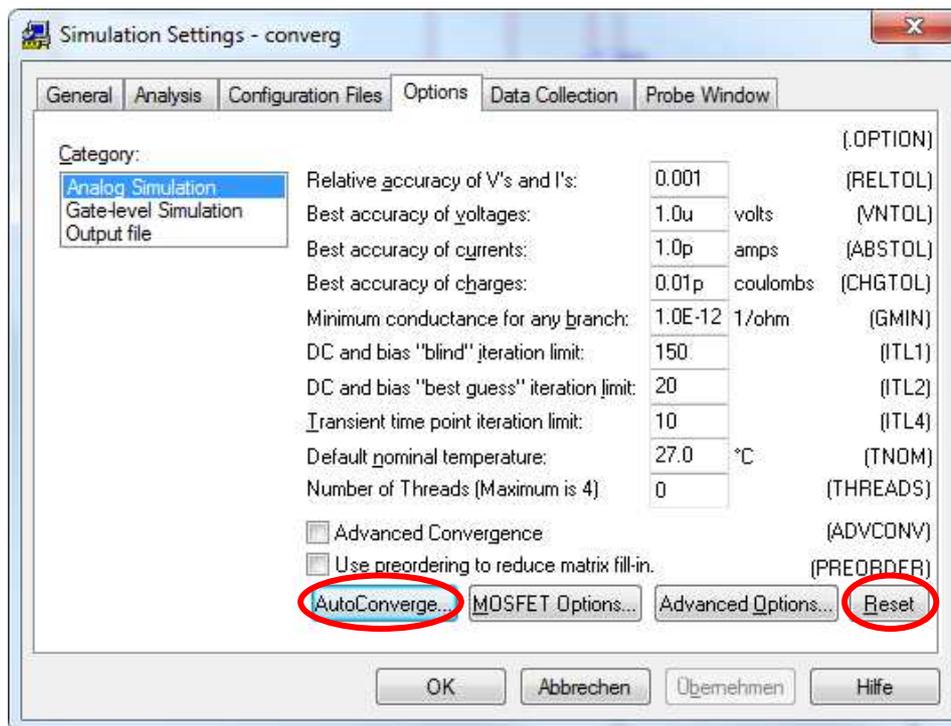
Click on **[Reset]** to bring back all of the original values of the parameters. Then click the **[AutoConverge..]** button, the **AutoConverge Options** dialogue box pops up. Check the **AotoConverge** check box to active the relaxed limits.
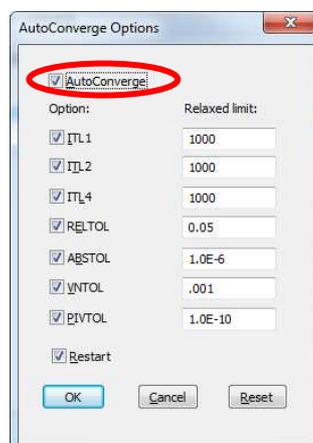
Click **[OK]** to accept the new relaxed limits and run the simulation. PSpice will run to completion without reporting any convergence problems.

In the Probe window add traces for V-Out, V-In and V-G. The curves are displayed as follows:
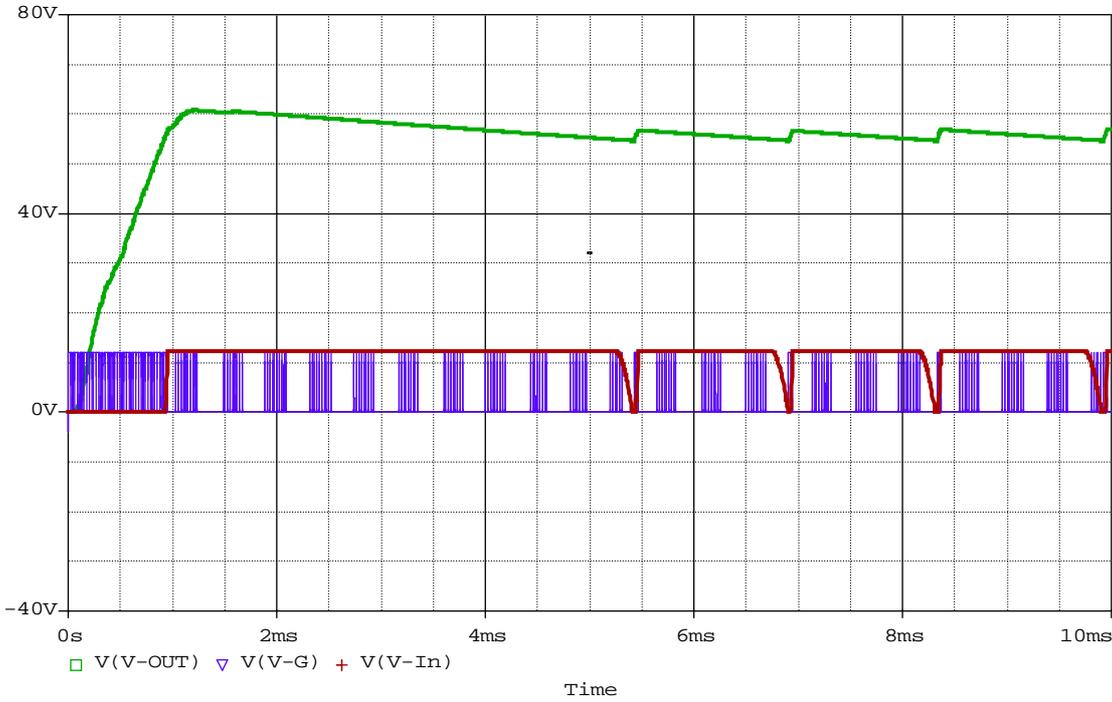
In the above diagram you will see, the output voltage V-Out, which is the voltage that crosses the capacitor C4, increases because of the charge of C4. The feedback voltage V-F increases but it is still smaller than the reference voltage V-R. As long as V-F < V-R, V-In stays at a low voltage level and the gate voltage V-G turns M1 on more periodically.

When V-F > V-R, V-In changes to have the high voltage level, the Power MOSFET M1 switched periodically off more and the output voltage V-Out decreases because of the discharge of C4. V-F decreases also. As long as V-F > V-R, V-In stays at a high voltage level and the capacitor C4 will continue to be discharged through R9. When V-F < V-R, M1 switches on to start the charge process for C4 and the output voltage V-Out increases.

The capacitor C4 will be periodically charged and discharged and the output voltage V-Out will increase and decrease accordingly. However, the output voltage will be kept at a certain DC voltage level.

We can add the current trace I(C4) for the capacitor and zoom in to have a better observation.
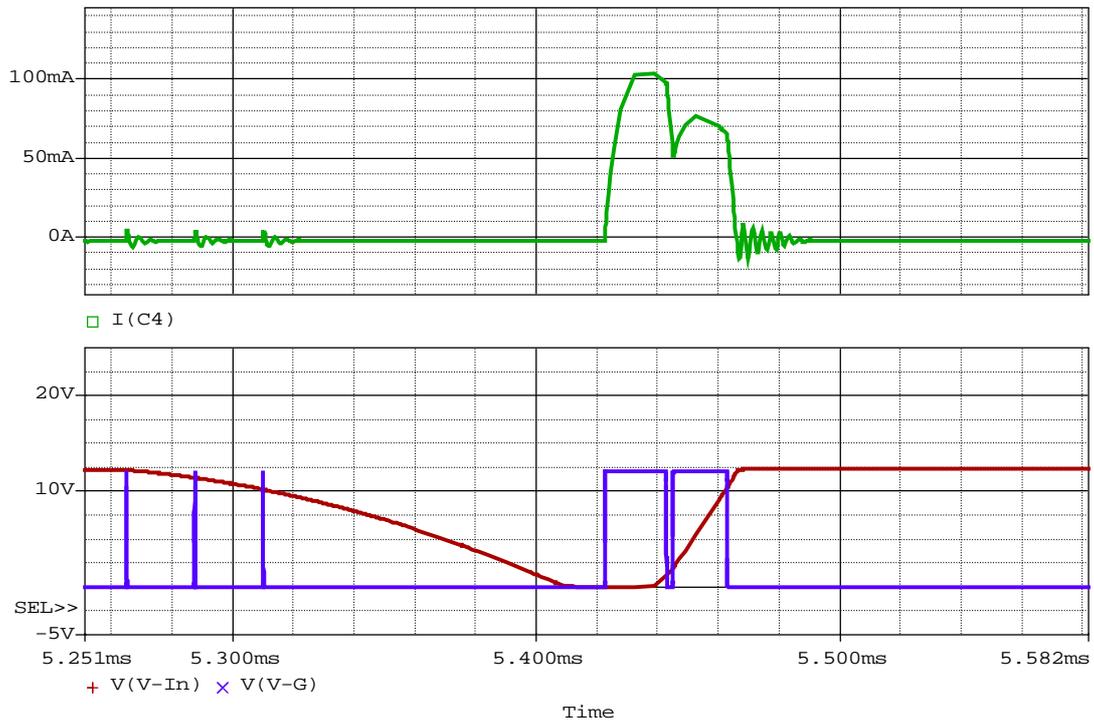
In the above diagram you can see how C4 will be charged while M1 switches on.

5 Bibliography

[1]   PSpice User's Guide, Cadence
[2]   OrCAD Capture User's Guide, Cadence
[3]   Leistungselektronische Schaltungen, D. Schröder, Springer
[4]   http://en.wikipedia.org/wiki/Newton%27s_method