

# Simulating a switch mode power supply with LTspice

## Aims of this experiment

- Learn how to use an electronic circuit simulator
- Explore the operation of a familiar filter circuit
- Learn how USB chargers and similar power supplies work

## Lab notebook

You should keep a written lab notebook while doing this and other experiments. It should contain

- Diagrams of circuits you simulated or built
- Waveforms, analysis, observations and explanations as prompted by this manual, and things that you decided to pursue. Be quantitative.
- Enough detail to enable someone to quickly pick up the thread of what you did and observed.

## 1 Install a circuit simulator

Computer programs that deduce and solve all the equations (Kirchoff, Ohm etc.) of an electronic circuit are widely used. If used wisely, they can be quite accurate; the typical reason for a mismatch with reality is that reality does not live up to the idealisations of your circuit diagram.

These notes are written for the LTSpice simulator, but others are available, e.g. many other Spice derivatives, and Qucs and its derivatives. They are mostly free software, and most run on multiple operating systems. [spice = simulation program with integrated circuit emphasis; qucs = quite universal circuit simulator]

### 1.1 Installing LTspice

LTSpice is built for Windows and Mac. It runs OK within Wine on Linux. Download the latest version of LTspice (presently XVII) from [www.analog.com](http://www.analog.com) at <https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>

## 2 Enter and simulate simple circuits

Check that you can get the same results as shown in section 2.1, and attempt section 2.2.

## 2.1 Resistor-capacitor low pass filter

We begin with a circuit that you may be familiar with, to check that we can reproduce its behaviour. Use LTspice (or other simulator) to draw the circuit shown in Fig. 1.

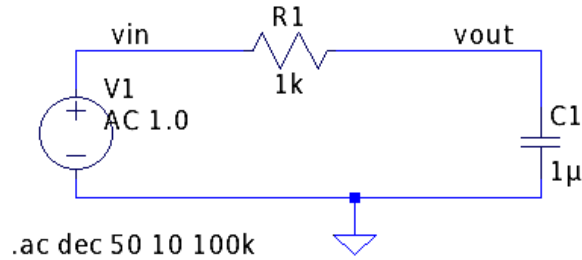


Figure 1: R-C Low pass filter circuit

The resistor, capacitor and ground symbol are selected from the toolbar and placed by clicking. Ctrl-r (or the rotate symbol on the toolbar, that looks like EM) rotates the symbol before placing. Esc deselects the part. There is an Undo option in the Edit menu. The voltage source is found as "voltage" in the list revealed by the Component button (looks like an AND gate). The Wire tool from the toolbar makes connections.

Use a right click to alter the values of the components from their defaults to: R1 = 1k (1000  $\Omega$ ); C1 = 1u ( $10^{-6}$  F); V1: Advanced, small signal AC analysis, AC amplitude = 1.0 (1 volt). Use the Label Net tool (letter A) to apply vin and vout labels to the appropriate points in the circuit.

Select the simulation required from the Edit Menu » Spice Analysis » AC analysis: Type of sweep = Linear, 250 points, start = 10, stop = 100k. Place the `.ac` command that it generates on the schematic.

Save the circuit, and run the simulation by clicking the Run icon (running man). Any errors will appear in a window. Click on the vout and vin wires to see their voltages plotted as a function of frequency, see Fig. 2

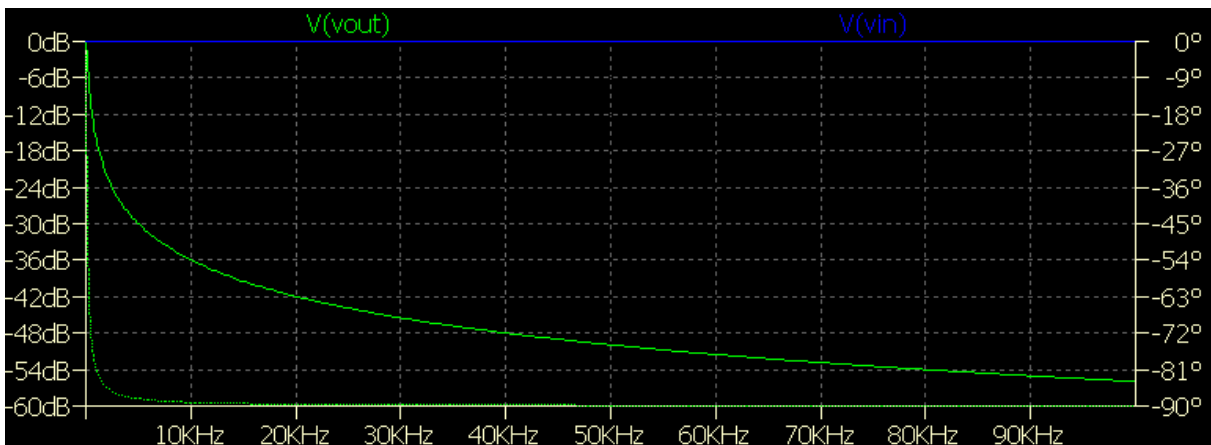


Figure 2: Filter circuit plot

The plot shows that  $V_{in}$  has a constant value of 1 volt at all frequencies, while  $V_{out}$  quickly falls towards zero at high frequencies. A standard representation used in electronics is the Bode plot, which has logarithmic amplitude and frequency axes, so right-click on the frequency axis to make it logarithmic. The amplitude in decibels is automatically a logarithmic scale, but you may prefer to make it logarithmic in voltage units. The linear sweep does not calculate enough points to make a clean plot at low frequencies, so go back and change the ac simulation to use 50 points per decade (a logarithmic sweep), and simulate again.

You can alter the appearance of the plots using the Control Panel tool (hammer) » Waveforms. Dark lines a couple of pixels wide on a light background are advised. The result, Fig. 3, shows why this circuit is called a low-pass filter.

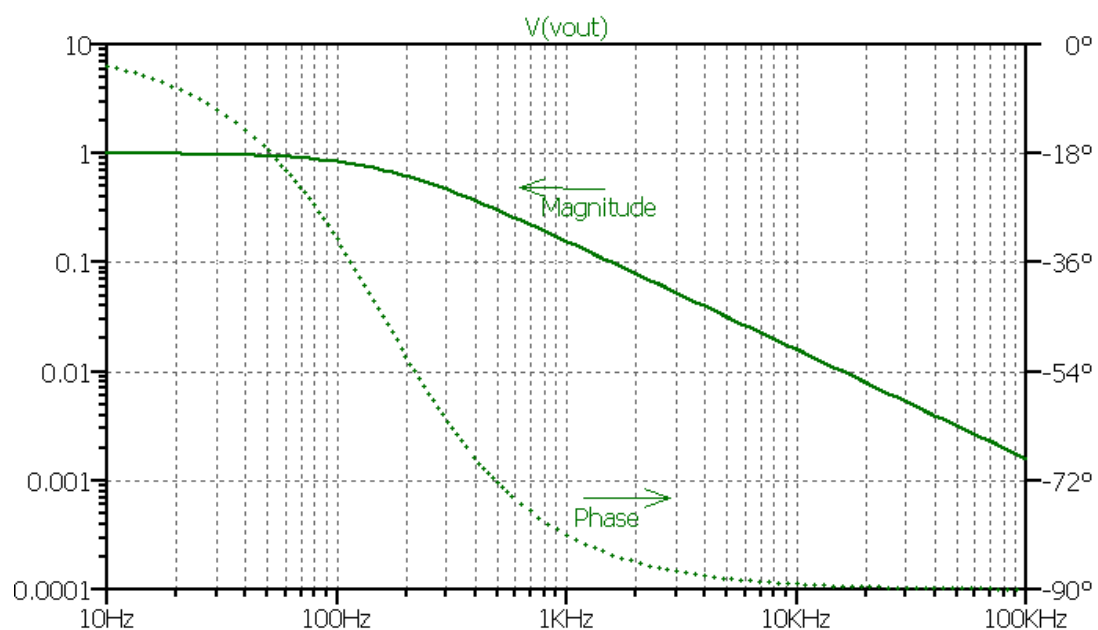


Figure 3: Bode plots of low-pass filter

Low frequencies (below 100 Hz, say) pass from input to output with little attenuation, but high frequencies (above 1000 Hz, say) do not. It can be quite a useful circuit fragment, even though the cutoff point is not that sharp.

## 2.2 Resistor-capacitor high pass filter

Interchange the positions of the resistor and capacitor in the circuit of section 2.1 and produce graphs showing the Bode plots for the new circuit. Comment on the results.

## 3 Switch mode power supplies

The need to convert electrical power from one voltage level to another is very common. A USB charger that converts 240 V mains to 5 V to charge a phone is one example, as are the step-up and step-down

conversions that the phone may need to power its display ( $\sim 10$  V) and its processor ( $\sim 1$  V) from its nominally 3.7 V battery. In this section, you will investigate how this is done using inductors and electronic switches.

### 3.1 Inductor and Switch

Consider the circuit of Fig. 4. Its behaviour is governed by the expression for the voltage across an inductor,  $V_L = -L di/dt$ . Before time  $t = 0$ , the current  $i$  in the circuit is zero. At  $t = 0$  the switch is closed. By applying Ohm's and Kirchoff's laws, you should show<sup>1</sup> that the current has the form of an exponential approach to  $V/R$ ,

$$i = \frac{V}{R} \left[ 1 - \exp\left(\frac{-t}{\tau}\right) \right], \text{ where } \tau = L/R.$$

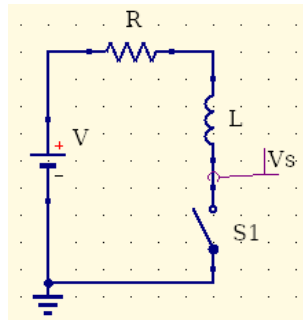


Figure 4: Inductor plus switch

Thus the current builds up, at a rate determined by the  $L/R$  time constant, to  $V/R$ . The exciting part happens when the switch is then opened and the current stops. If this is done instantaneously ( $dt = 0$ ), then  $L \frac{di}{dt} = -\infty$ , so an infinite voltage appears at  $V_s$ . While this is a good way of generating an increased voltage, a numerical circuit simulator cannot handle instantaneous infinities, and in any case they do not occur in a real circuit, so we will next simulate something a little more realistic.

### 3.2 Step up (“boost”) converter

Set up a simulation of the circuit shown in Fig. 5. Set the values of the resistor and inductor to  $1 \Omega$  and  $1$  mH to give a time constant  $\tau = L/R$  of  $1$  ms.

<sup>1</sup>Write down Kirchoff's voltage law, separate the variables  $i$  and  $t$ , then integrate, substitute the limits  $(0, 0)$  and  $(t, i)$ , and take the antilog.

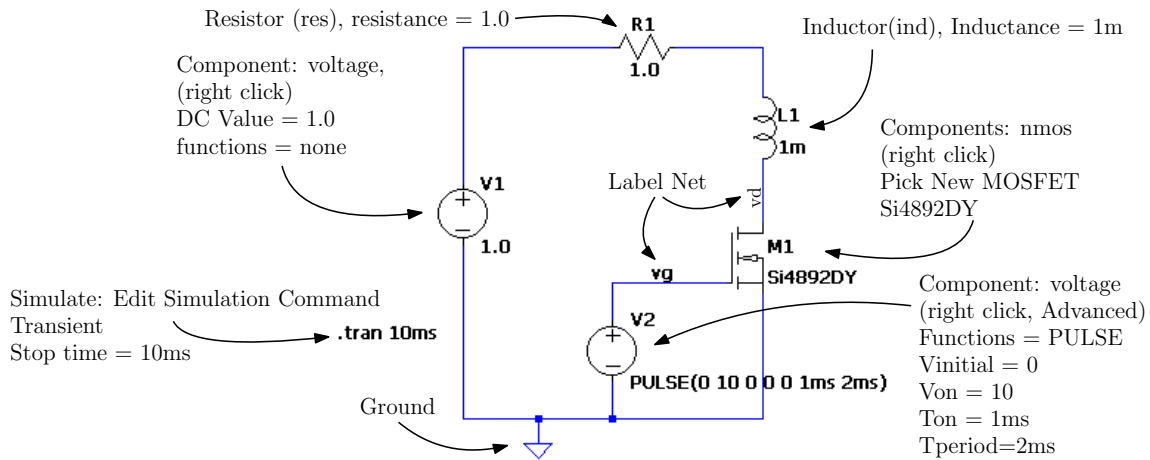


Figure 5: Boost converter

The MOSFET M1 (see Fig. 6) acts as the switch in this circuit. When a positive voltage of a few volts is applied to its gate electrode, it conducts strongly between source and drain. Without the gate voltage, it switches off. Here we replace the electronics that would normally drive the gate by a source of rectangular voltage pulses, V2. In contrast to this, voltage source V1 is set to a constant 1 volt, like a battery.

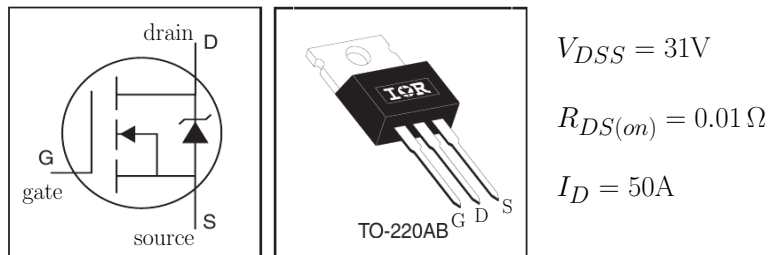


Figure 6: Si4829DY MOSFET.  $R_{DS}$  is the resistance when on;  $V_{DSS}$  is the voltage at which the isolation breaks down when off.

Because the circuit is neither linear nor time-independent, an AC simulation is of little help. Instead, we use a transient simulation that numerically integrates the differential equations of the circuit.

By running the simulation, you should be able to make a plot like Fig. 7, which shows the switch control  $V_g$  in blue, the steady rise of the inductor current in red, and a large voltage transient at  $V_d$  in green. To make a plot like this, click on the  $V_g$  and  $V_d$  wires when the cursor changes to a voltage probe, and click on the body of the inductor when it shows a little ammeter. Plot Settings » Add Plot Pane allows you to add a second graph, then you can drag the curve labels between graphs.

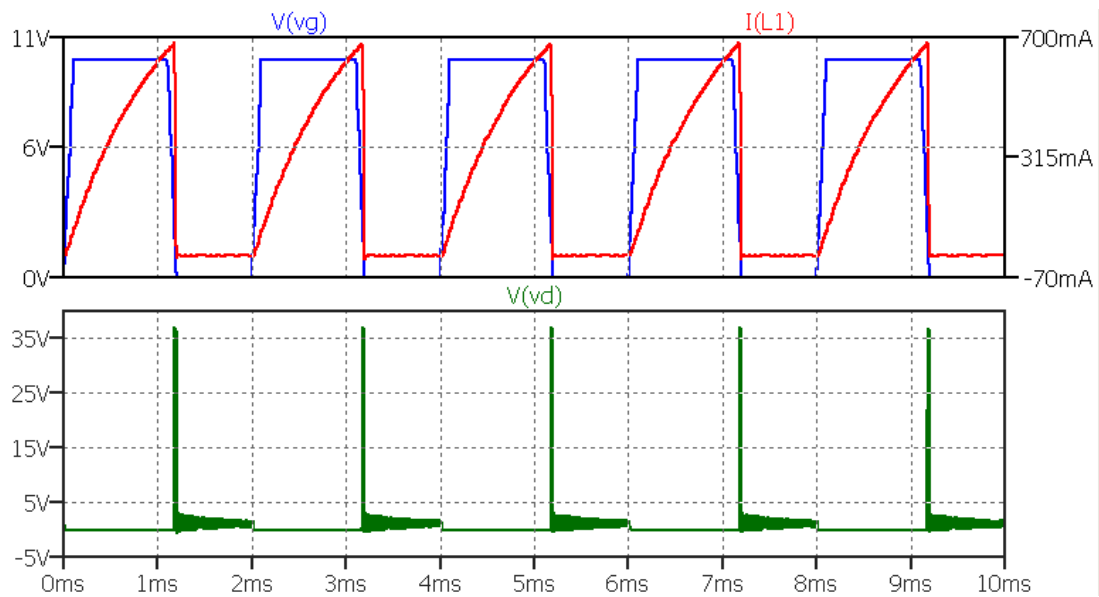


Figure 7: Boost converter in operation

Record the waveforms that you get, and carefully annotate your graph to show how you understand the relationship between the opening and closing of the switch, the varying current in the inductor, and the voltage that is generated. Be as quantitative as possible. What determines the size and duration of the voltage spike? The fat voltage trace near zero volts is worth exploring too. You can zoom into the plotted data using the magnifying lens buttons on the toolbar.

### 3.3 Storing the output voltage

A continuous high voltage output would be more useful than pulses. Add a diode (from the toolbar), a capacitor of capacitance  $1\mu$  and resistor of resistance  $50k$  as shown in Fig. 8. The idea is that the  $1\mu\text{F}$  capacitor C1 stores the output voltage, while the  $50\text{ k}\Omega$  resistor R2 represents a load circuit that makes some use of the high voltage power.

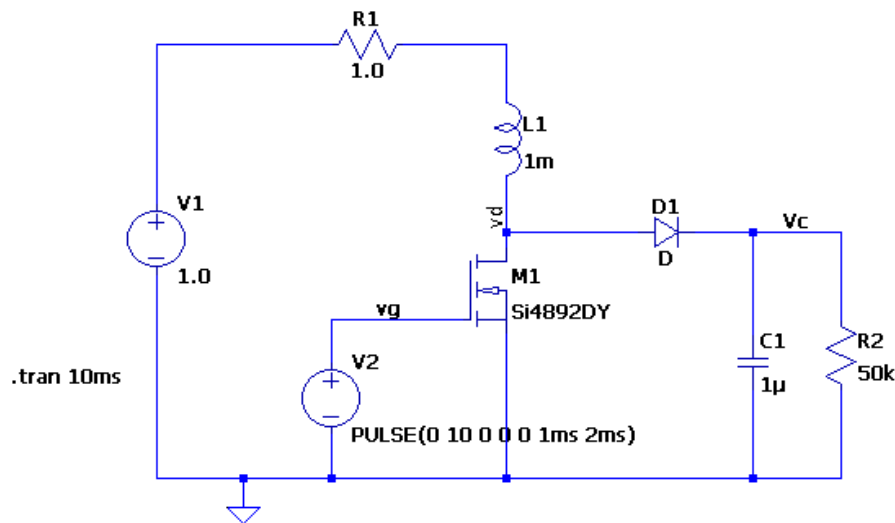


Figure 8: Smoothing the output voltage

Simulate this circuit, plotting also the capacitor voltage  $V_c$ , which you should see does not fall away like  $V_d$ . Annotate your plot and use it to explain when the diode conducts and when it does not, and the shape of the  $V_c$  waveform. Also simulate for longer (Edit the `.tran 10ms` to be e.g. 30ms) and explain the steady state behaviour.

### 3.4 Efficiency

Ideally, all the power entering a circuit like Fig. 8 would be delivered to the load. Considering just one of the pulses, compare the maximum energy stored in the inductor with the amount delivered to the capacitor [the energy stored in an inductor is  $\frac{1}{2}LI^2$ , that in a capacitor is  $\frac{1}{2}CV^2$ ]. Also note the power lost in  $R1$  and comment on your result. You may find it helpful to use the cursor on the graphs to read values (shown at the bottom of the window).

## 4 Flyback converter: output isolation

A disadvantage of the circuit of Fig. 8 is that the output (e.g. 5 V dc USB power) cannot be isolated from the input (e.g. derived from 240 V ac mains). This could be a safety hazard. Now we add a second winding on the inductor to allow us to take power out of it without any electrical connection; the input and output circuits are coupled only by their shared magnetic flux.<sup>2</sup>

### 4.1 Simulating a flyback converter

Convert your model to Fig. 9 (the Cut tool [scissors] is useful for removing circuit elements). Tell LTspice that the two inductors are coupled magnetically by using the Spice Directive button (looks like

<sup>2</sup>The term flyback relates to the movement of the electron beam in vintage cathode ray tube televisions; the magnetic flux was used to deflect the electron beam as well as to generate a high voltage.

.op) » Edit text on the schematic » How to netlist = SPICE directive to add the text **K L1 L2 1** in the text box. Place the generated text near the inductors. Notice the reversed direction of the diode. For convenience in plotting, the Vc and ground terminals have also been interchanged. Run the simulator, and examine how the signals compare to those in the non-isolated circuit of Fig. 8.

If the two inductors are not oriented the same way, as indicated by the circle near one end, only a small Vc will be seen. Investigate this and explain why.

See if you can get more voltage out of the circuit. Hint: a step-up transformer needs more turns on the secondary than the primary.

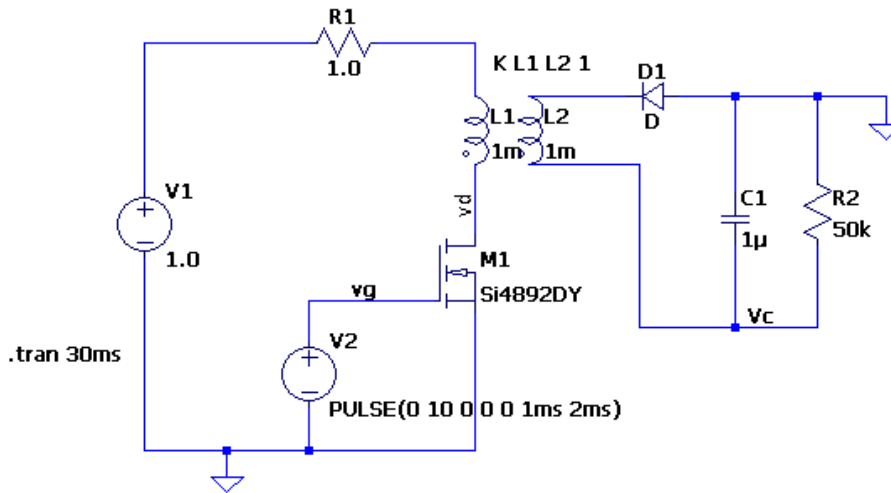


Figure 9: A flyback power converter

## 4.2 How realistic is all this?

In the separate practical “Switch mode power supply” experiment, you can build a circuit very similar to Fig. 9, and use it to generate a few hundred volts from a 1.5 V battery. A real mains power supply uses the same principle, as shown in Fig. 10. You should be able to identify in it all the key elements of Fig. 9, alongside its input power and control circuits.



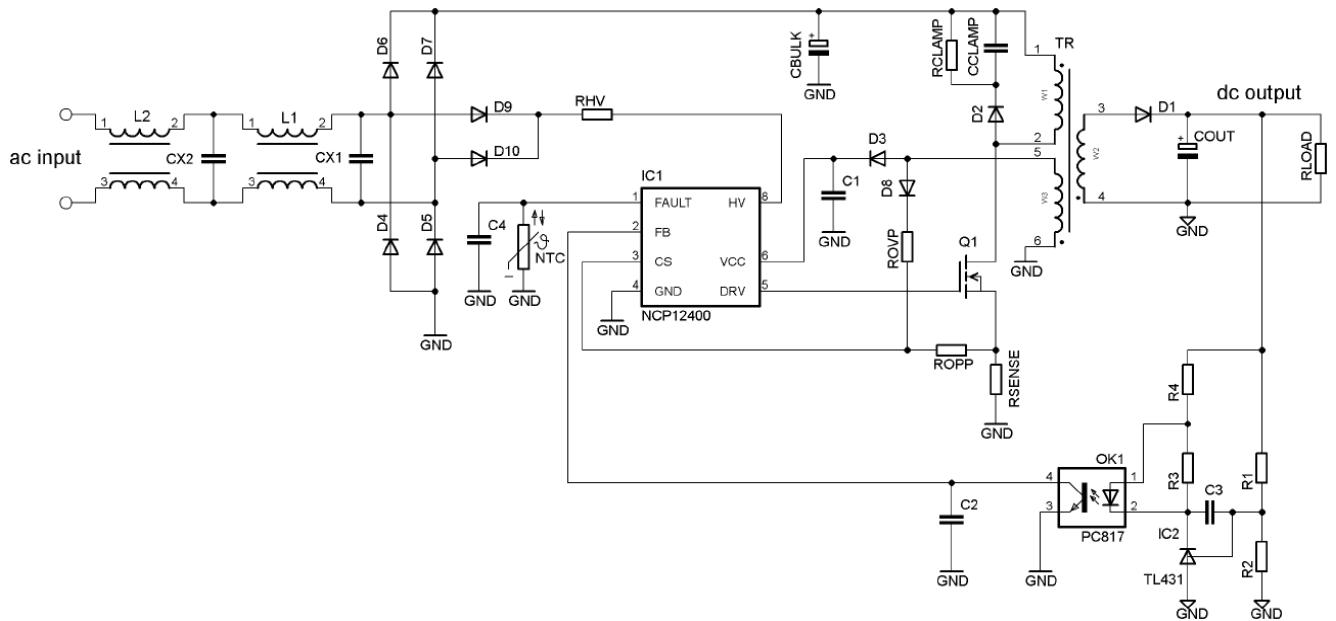


Figure 10: A practical mains power supply (ON Semiconductor NCP12400 data sheet)

## Recreational reading

- <http://www.righto.com/2012/10/a-dozen-usb-chargers-in-lab-apple-is.html>
- <https://spectrum.ieee.org/tag/switching+power+supplies>