Introduction to LTspice

School of Engineering

This script introduces LTspice, a simulation tool for circuit analysis used in the electronics courses at ZHAW.

Up to and including chapter 5the basic functions are explained. These chapters can be worked through individually in 2 to 3 hours.

It is advised that you work through these chapters in the given order as necessary previous steps are assumed in each example case and these steps are therefore not explained again.

Chapter 6collects special simulation elements, which are needed less often and serves as a reference rather than belonging to the introduction.

All examples discussed can be found on the server http://dublin.zhaw.ch/~hhrt/LTspice/

Notice:

If a circuit works in the simulation, it is not yet guaranteed that the real design will work too. Nevertheless, simulation is a time-saving tool when designing and later testing the real circuit.

© Hanspeter Hochreutener, hhrt@zhaw.ch , February 14, 2021. Translation: Mathis Nussberger

Institute of Signal Processing and Wireless Communications (ISC)
https://www.zhaw.ch/de/engineering/institute-zentren/isc/

School of Engineering https://www.zhaw.ch/de/engineering/

Zurich University of Applied Sciences http://www.zhaw.ch

Table of contents

1. Install LTspice

Download LTspice free of charge from
http://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html

and install it. LTspice can, via WINE, also be used under Linux.

There is also a version for the Mac, the handling being slightly different.

1.1. Library for ZHAW Electronics Lab

There is a symbols compilation and a library for the semiconductors used in the basic electronics courses at ZHAW. Install both to follow this introduction.
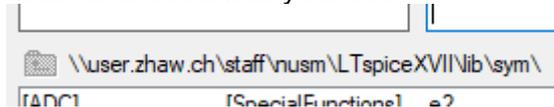
- Start LTspice and click on
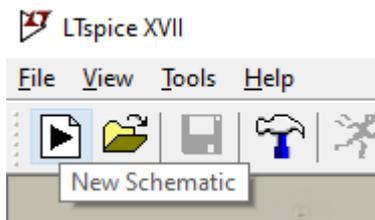
which creates a new schematic. Now click on

If you do not see this icon, you may have to enlarge the LTspice window horizontally.
In the component selection window take notice of the mentioned path, which will be different than shown below in your case


- Download the symbol file http://dublin.zhaw.ch/~hhrt/LTspice/Symbole.zip, unzip and copy it to this directory: **….\sym\ZHAW**.

- Download the Spice library http://dublin.zhaw.ch/~hhrt/LTspice/ZHAW.lib and copy it to this directory:**…\lib\sub**
  Attention: The file name must be "**ZHAW.lib**"; "**ZHAW.lib.txt**" does not work. MS Windows does not show you file extensions per default. Open an explorer window (shortcut windows key – e), select the tab "Ansicht" and check the checkbox "Dateinamenerweiterungen"
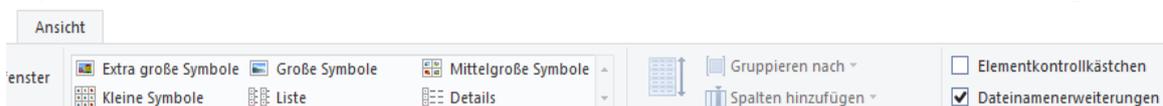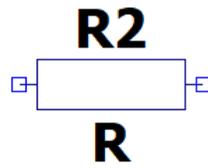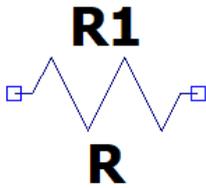

  Note: **Do not use the browser "Chrome",** because it changes the file content.

- Restart LTspice, repeat the steps above up to the opening of the component selection window. The folder ZHAW should now show up in this window.
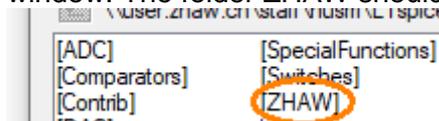
1.2. European Resistor

After installation, LTspice uses an anglo-american resistor symbol, here shown for resistor R1.

If you like to generally use an european resistor symbol like the one shown for R2, proceed as follows:

1. Open the folder C:\Users***\Documents\LTspiceXVII\lib\sym (for the Mac and older LTspice versions, see locations mentioned in Section 1.1)

2. Find the file "res.asy" and rename it to "res_us.asy".

3. Navigate to the folder C:\Users***\Documents\LTspiceXVII\lib\sym\Misc

4. Copy the file "EuropeanResistor" to the "sym" folder (one level up)

5. Rename the copied file to "res.asy"

6. Restart LTspice. All circuits, even preexisting, will now use the european symbol. The icon in the icon list below the menu list will not change, however.

7. By reversing these steps you can go back to the anglo-american resistor symbol at any time.

2. Schematic Capture

The easiest way to learn how to draw the schematic of a circuit is to use an example. The individual steps for a preamplifier with single-supply operational amplifier according to the schematic below are demonstrated. You do not need to know anything about this circuit or its components right now. In fact, you will find out what it does and which components are used by executing the following exercises and study the explanations in between.
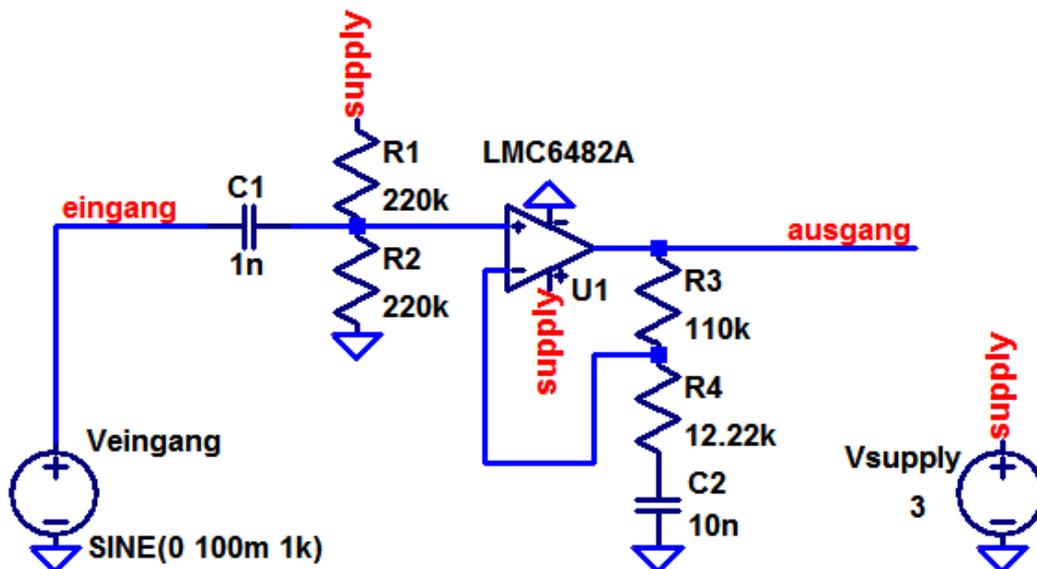
2.1. Move, wire and delete components

These are the editing commands you need to draw the schematic:

⌕ (**Edit => Rotate)** to rotate a part **before** placing **it.**
Note: An already placed part must first be selected with Drag or Move.

⟡ (**Edit =>** Mirror**)** to mirror a part **before** placing **it.**
Note: An already placed part must first be selected with Drag or Move.

✋ (**Edit => Drag)** to move the part. Select the desired part or an entire area by **left-clicking**, move it to the desired location and place it by left-clicking again. The wire connections are dragged along.

✋ (**Edit => Move)** does the same, but the wire connections are not pulled along.

⊽ (**Edit => Place GND)** connects to ground = 0 volts reference potential for simulation

✎ (**Edit => Draw Wire)** connects components. Left-click at start of wire, change of direction and end of wire.

✂ (**Edit => Delete) deletes** connections and components. You can also use the mouse to select an entire area to be deleted.

Right-click on a connection and **Delete Net** deletes the whole net and not just a wire segment.

ESC key each time to end a function.

🔍 🔍 🔍 🔍 to enlarge, center, reduce and fit the schematic.

2.2.   Passive components

Exercise

Start LTspice and open an empty schematic by clicking the button 🔲 on the toolbar or using the menu **File => New Schematic.**
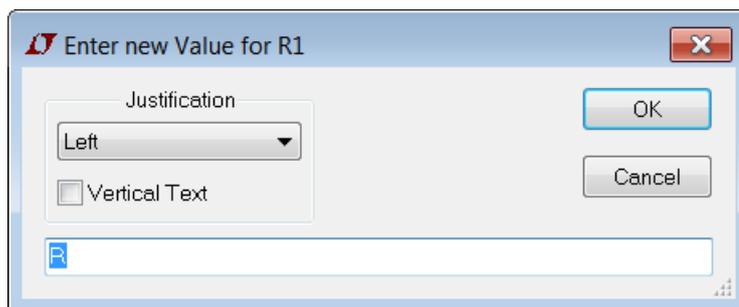
Exercise

For the resistors click on the icon (⟩ Menu **Edit => Res**istor**)**.

Place the four resistors.

ESC key to exit.

Change resistance value with **right-click** on the letter **R** next to the icon

⎐ Enter new Value for R1

| Justification |
| Left ▼ |
| ☐ Vertical Text |

OK

Cancel

R

R overwrite with 220k, etc.

Similarly, the part name can be customized by **right-clicking on the name**.

For the numerical values, you can use these engineering abbreviations:

Suffix	Multiplier
T	1e12
G	1e9
Meg	1e6
K	1e3
Mil	25.4e-6
M	1e-3
u(or µ)	1e-6
n	1e-9
p	1e-12
f	1e-15

Risk of confusion: **"M"** or **"m"** is **milli = 10^{-3}, "MEG"** or **"meg"** is **mega = 10^6**
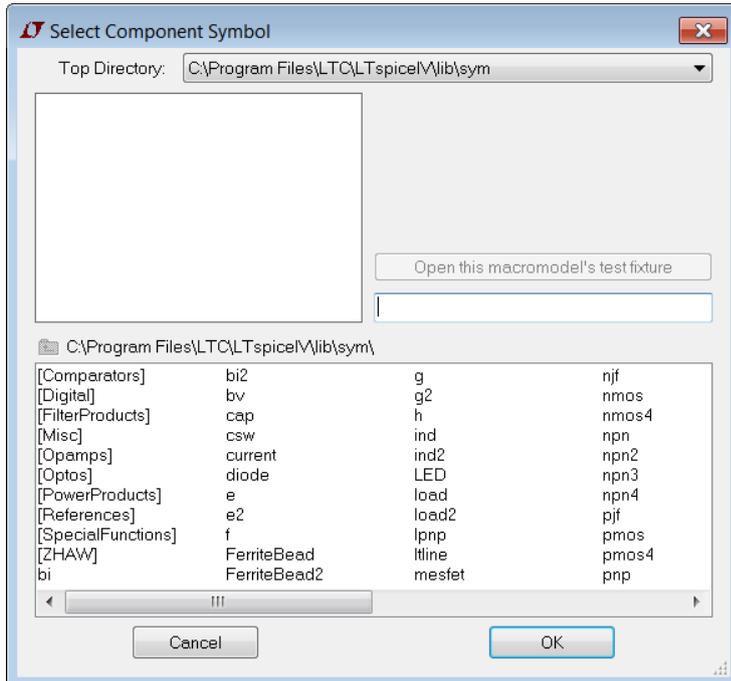
<u>Exercise</u>

Proceed in the same way for the capacitors: Icon ($\perp$ Menu **Edit => Capacitor).**

Change capacitor value with **right-click** on the letter **C** next to the icon.

2.3. Active components

All components (even resistors and capacitors) can be obtained via the icon (⊅ **Edit => Component).**

The square brackets denote folders with the more components.

Double click on **[Opamps]** shows the operational amplifiers from Linear Technology.

Back by double-clicking **[...]**.

The **[ZHAW]** folder lists the semiconductors used in the basic electronics courses at ZHAW.

> Exercise
>
> Select the LMC6482 from the [ZHAW] folder and position it in the schematic.
>
> Move, rotate and wire all components mentioned so far until the schematic looks something like this (pay attention to the polarities of the LMC6482):
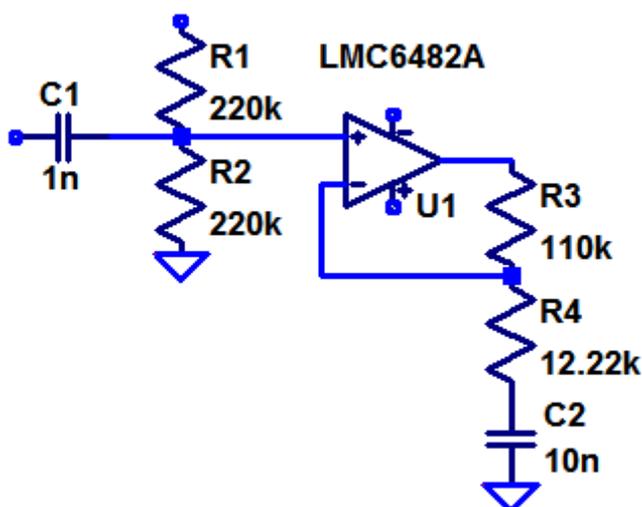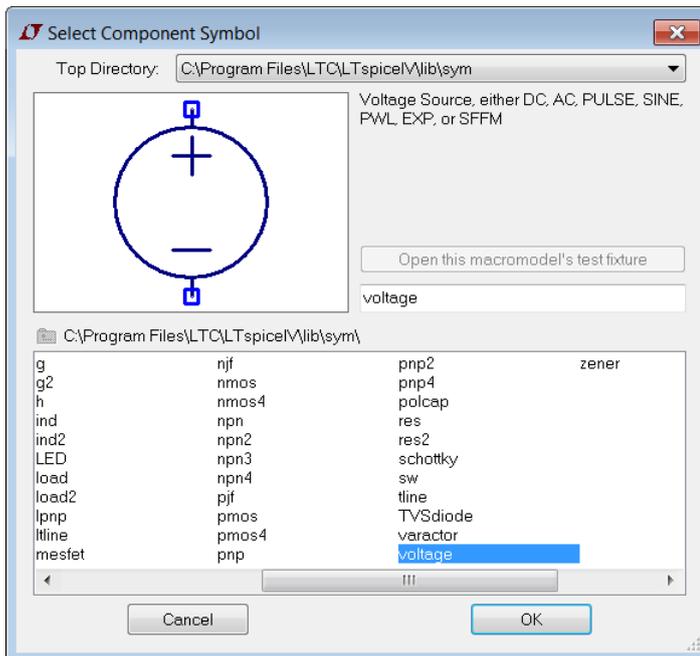

The voltage supply of the operational amplifier and the input signal are still missing, as well as a dedicated output port.

2.4. Voltage and current sources with fixed values

Exercise

Select a voltage source via the icon (⚬ **Edit => Component)** then select **voltage.** (For a current source you would select **current**).

Right-click on V1 and change the name to Vsupply.

Right-click on V and type 3 (= 3 volts).

2.5. Signal generators

The voltage and current sources described above can be configured to behave other than just like a simple fixed voltage/current source.

Exercise

Select another voltage source for the input signal as described above. Change the name V2 to Veingang.

Right-clicking the icon and clicking the **Advanced** button opens the menu with advanced settings:
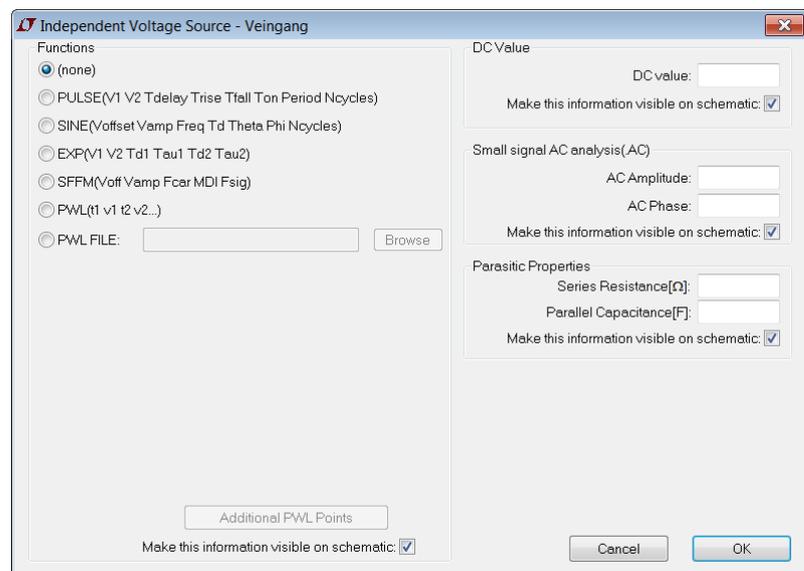
(none) indicates a DC voltage source.

PULSE is used for square wave signals with defined rise and fall times.

SINE is a sinusoidal signal.

Exercise

Set V2 to 1 kHz, 100 mV amplitude without DC offset, sine shaped.

2.6. Signal names and comments (label, text)

In order to keep the schematic well readable, the lines for the supply voltage are often not drawn. You simply name corresponding connections using the same expression. This done, they are automatically connected to each other.

🔲 (**Edit => Label Net)** names a circuit connection (a circuit node)

> Exercise
>
> Place a label with the name "supply" on the + terminal of the voltage source Vsupply, on the + supply of the operational amplifier and on the upper terminal of the resistor R1.
>
> Put a ground connection to the - terminal of the voltage source Vsupply and to the - supply of the operational amplifier.
>
> Proceed analogously for the input voltage.
>
> Label the "input" and "output" nodes too. For the latter you draw a horizontal wire starting at the top of R3 and ending somewhere to the right of it as done in the example schematic. Wires are allowed have open endings.

_Aα_ (**Edit => Text)** for comments, titles, etc.

> Exercise
>
> Complete the schematic with a heading.
>
> Your schematic should now look like the one shown at the beginning of chapter 2.
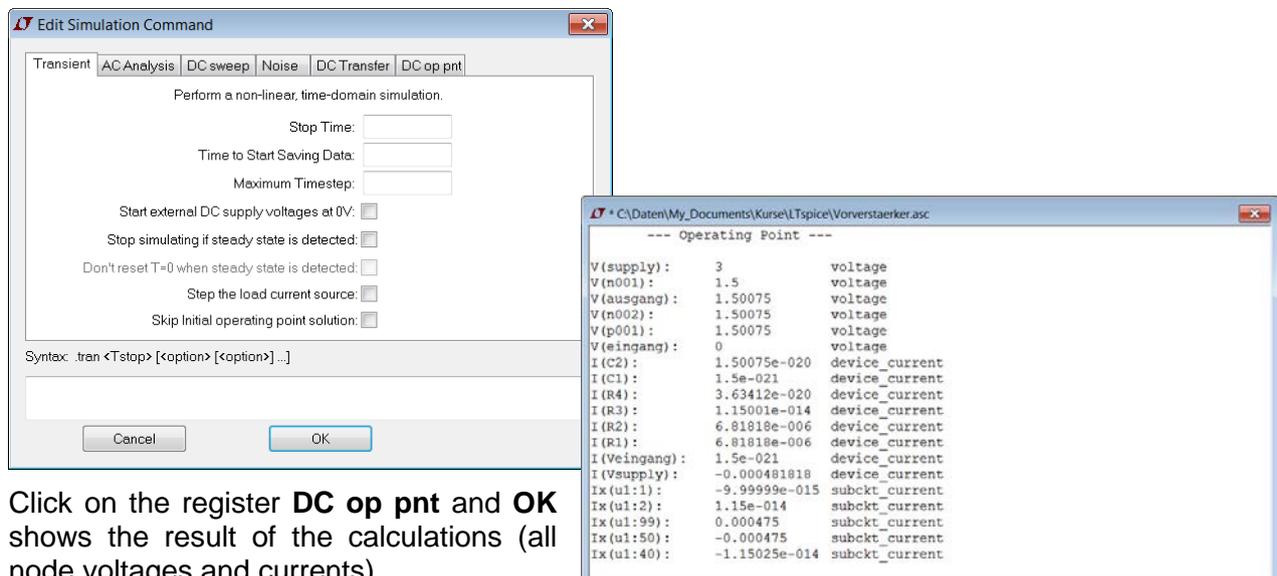

3. Simulation (LTspice Dot Commands)

You can use the previously drawn schematic for the simulation or load the schematic from the server http://dublin.zhaw.ch/~hhrt/LTspice/Vorverstaerker.asc

3.1. Operating point (DC Operating Point)

Before doing any further simulations (or measurements), it is worth checking that the so called DC operating point is correct.

For the operating point calculations, capacitors are considered as interruptions (current = 0) and inductors (used in an example further below) as short circuits (voltage = 0).

🏃 (**Simulate => Run)** opens this dialog window

Click on the register **DC op pnt** and **OK** shows the result of the calculations (all node voltages and currents).

Where you have specified names for nodes, these are used; the other nodes are numbered by LTspice.

When the cursor is moved to the node between R1 and R2 in the schematic, the status line (bottom left) shows: This is node N001. DC operating point: V(n001) = 1.5V

Or cursor over R1: Right click to edit R1. DC operating point: I(R1) = 6.81818µA   Dissipation=10.2273µW

Thus, the same information as in the table is also available when you move the cursor to the corresponding position in the schematic.

Important: You have to re-run your simulation after each change within the circuit, as minor it may be, otherwise results shown do not correspond to those of the modified circuit.

The schematic now contains **.op** . If a text **starts with** a **dot**, it is a **Spice command**.

> Exercise
>
> To what voltage is capacitor C2 charged?

3.2.   Transient Analysis (Transient)

Right-click the **Spice command** **.op** to select a different simulation type.

Select the register **Transient** and enter **3m** (= 3 ms) at **Stop Time.** The Spice command is now **.tran 3m**.

(**Simulate => Run)** opens an empty diagram window.

**Click on the** "input" and "output" **nodes in the diagram to** display the signals.

> Exercise
>
> Theoretically, the input signal should be amplified by the factor 1+R3/R4. Check if this is true.

Note: During the transient analysis, the operating point is calculated first before the signal sources are switched on. Under certain circumstances this may not be desired. In such a case one of the options "Start external DC supply voltage at 0V" (startup) or "Skip Initial operating point solution" (uic) should be selected in the simulation command.

3.3.   Frequency Response (AC Analysis)

Adjust the simulation command as follows
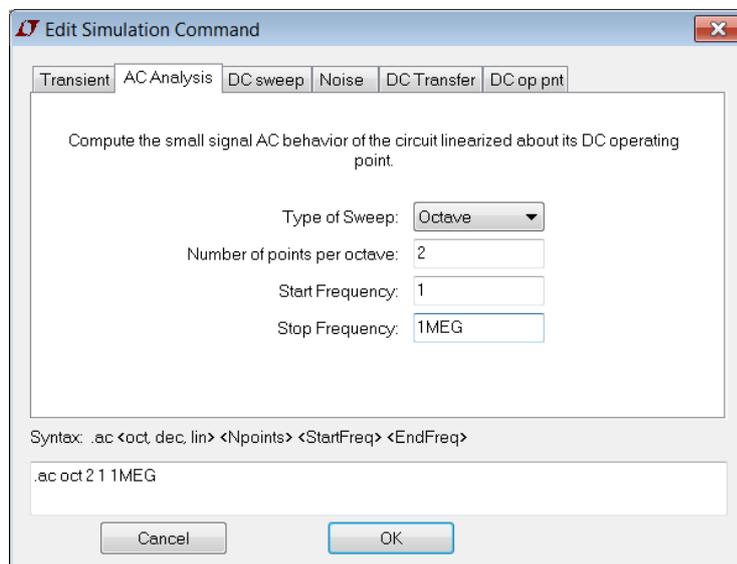
Note: **Octave** corresponds to a frequency doubling, **Decade to** a tenfold increase

The schematic now says **.ac oct 2 1 1MEG**

✗ (**Simulate => Run)** shows a window with the error message

```
No AC stimulus found:
     Set the value of a current or voltage source to "AC 1."
     to make it behave as a signal generator for AC analysis.
```

This means that you still have to specify which source should be used as input.

Right-click on the source Veingang and enter a 1 (= 1 V) on the right side at

Small signal AC analysis(.AC)
AC Amplitude: 1
AC Phase:
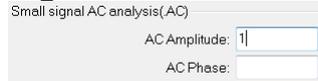
✗ (**Simulate => Run)** click in the **schematic on the nodes "output".**
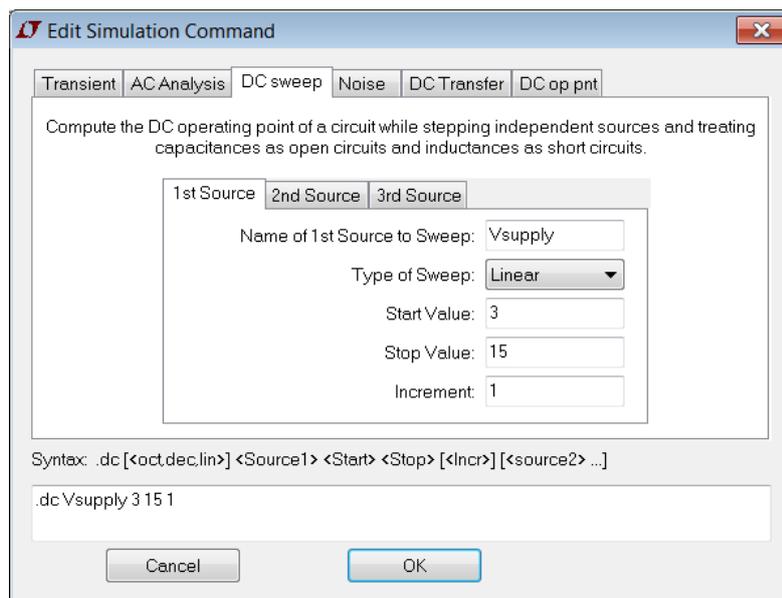
<u>Exercises</u>

What is the gain and phase shift at 10 kHz? The gain is shown in the pseudo-unit dB (decibel): Gain in dB = 20 · log10 (Gain), Gain = $10^{\text{Gain in dB}/20}$

Where are the lower and upper cut-off frequencies (3 dB drop)?

3.4.   Transfer characteristic (DC sweep)

To simulate the dependence of the operating point on the supply voltage, change the simulation command to


<u>Exercise</u>

How does the operating point change with the supply voltage?

DC Sweep can be used nicely for plotting characteristics, i.e. the voltage-current dependance of a selected basic component.

4. Diagrams (Waveform Viewer)

To illustrate the possibilities of the evaluations and diagrams, the example of a clocked buck converter is used. As for the first example, you do not have to know anything about this circuit to be able to follow the next steps.


Getakteter Abwärstwandler
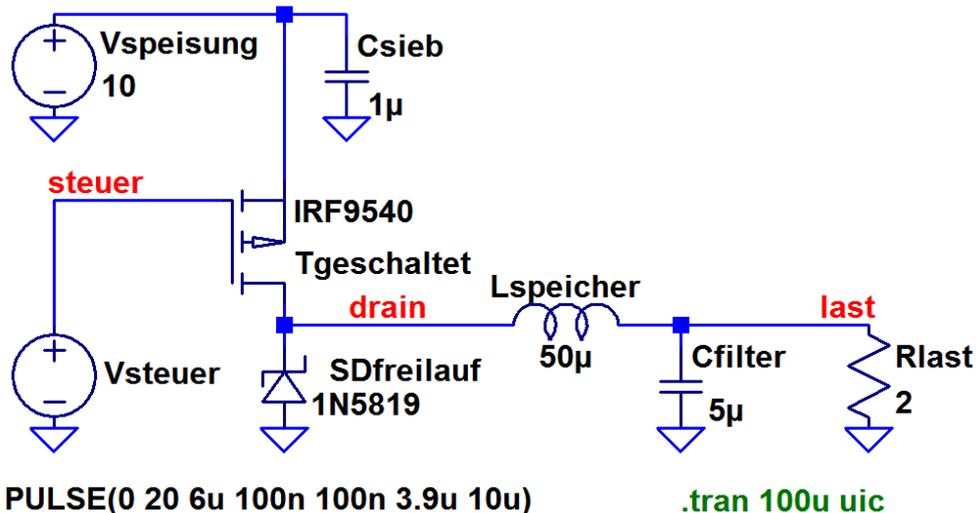
You can draw the schematic of the circuit yourself or load it from the server http://dublin.zhaw.ch/~hhrt/LTspice/Abwaertswandler.asc

Note: In clocked circuits, there is no stable operating point. Therefore the option "Skip Initial operating point solution" (uic) was selected here. Otherwise, LTspice would try to find the noexistent operation point and keep on calculating forever.

4.1. Curve selection (Data Trace Selection)

Run the example. After the simulation results are available, curves can be plotted.

### 4.1.1. Voltage versus ground (potential)
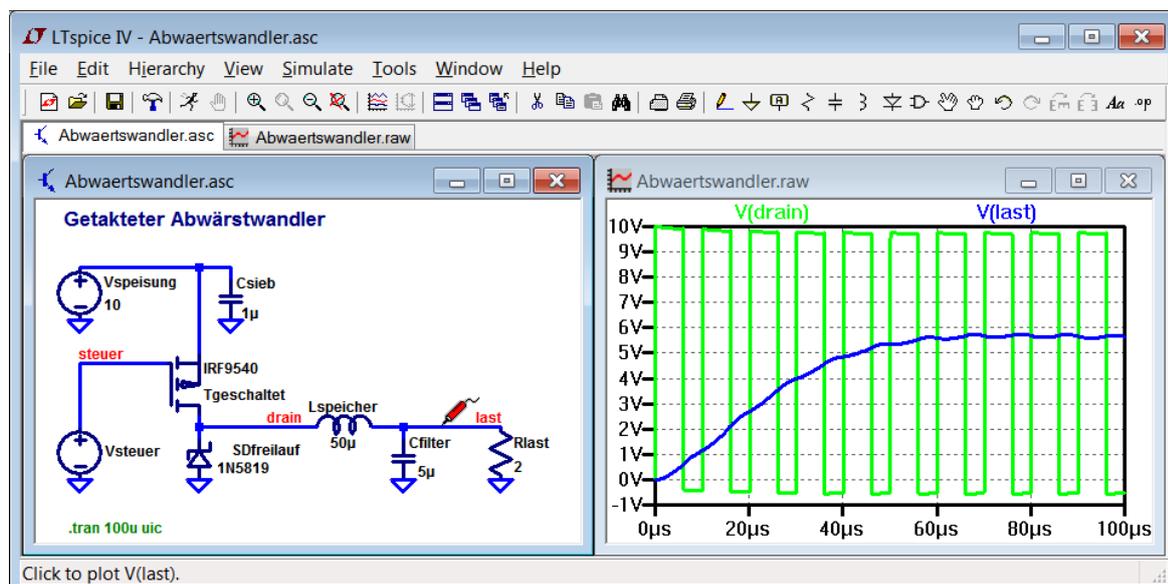
In the diagram click on the node (lines) from which you want to see the voltage curve with voltage referenced to ground. In the screenshot you can see the **red measurement probe** on the line with the name "last".

Exercise

Plot the voltage at the drain of the transistor (node labeled drain).

4.1.2. Voltage across an element

If, for example, you are interested in the voltage across the coil Lspeicher, **left-click** on the node on the right of the coil (left-click) and **keep the mouse button pressed** while moving the mouse to the left of the coil, where you release the mouse button. Alternatively, **right-click** on the node to the right of the coil and select **Mark Reference** (**black measuring probe**). **Left-click** on the node to the left of the coil (**red measuring probe**) shows the desired differential voltage **V(drain,last) = V(drain) - V(last)**
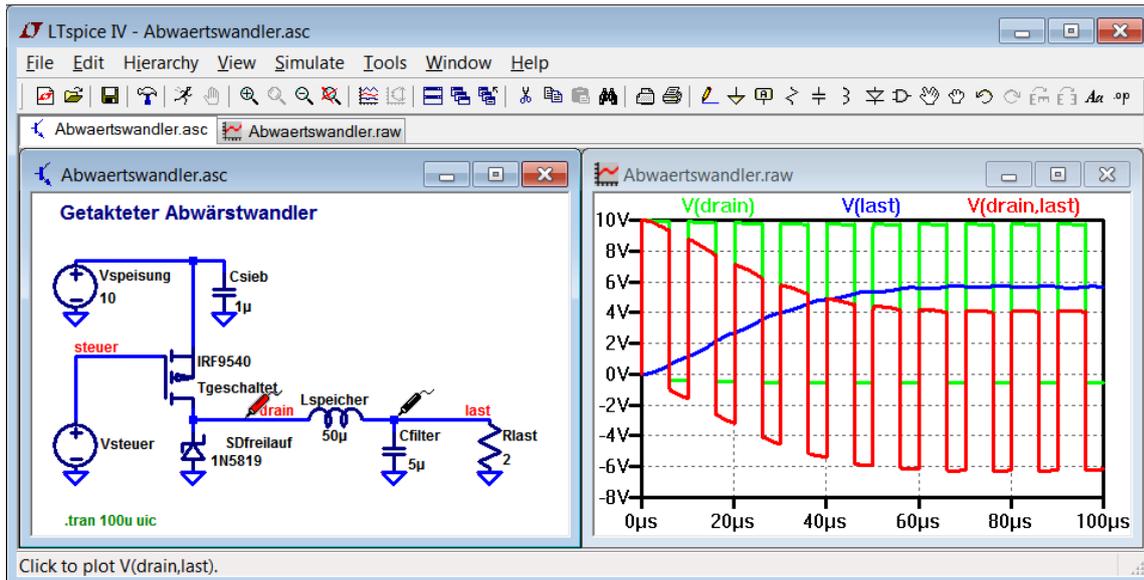
ESC key deletes the reference.

Exercise

Plot the voltages between drain and source and between gate and source of the transistor.

4.1.3. Current through a component

Hovering the mouse over a component changes the shape to a current clamp **Left-clicking** plots the current.

Exercise

Plot the drain current.

4.1.4. Current through a line

Hovering the mouse over a wire and pressing the **ALT key at** the same time changes the shape to a current clamp **Left-clicking** plots the current.

Exercise

Plot the current on the line to the resistor Rlast.

4.1.5. Power

If you move the mouse over a component and press the **ALT key at** the same time, the shape

changes to a thermometer 🌡️ **SDfreilauf** **1N5819** **Left-clicking** plots the curve of the power.

> Exercise

> Plot the power in the resistor Rlast.

4.1.6. Add Curve (Add Trace)

With this you are able to plot quantities or expressions that cannot be selected directly with the cursor.

Right-click in the diagram window => Add Trace

For example, to plot the voltage across the coil, you can click "V(drain)", type "-" in the field at the bottom, and click "V(last)" at the top.

Of course, one could have typed "V(drain)-V(last)" directly or the equivalent expression "V(drain,last)".

Or to display the energy stored in the coil (W = $\frac{1}{2} \cdot L \cdot I^2$), one can type the formula "1/2*50uVs/1A*I(Lstorage)**2" with ** for the power function. If the coil is specified with the correct units (50µH = 50µVs/A = 50µVs/1A) the axis labeling with µJ is correct too.

Note: All functions: **Help => Help Topics => Register: Index => Waveform Arithmetic**

> Exercise

> Plot the energy stored in the capacitor Cfilter, which is W = $\frac{1}{2} \cdot C \cdot U^2$.

> Control: If your formula is correct, the unit of measure is µJ.
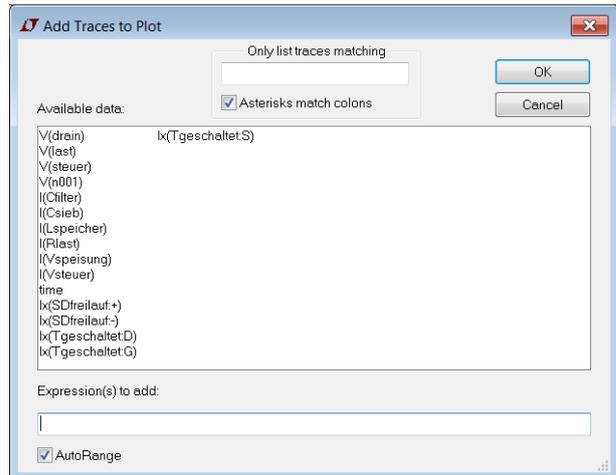
4.1.7. Curve selection (Visible Traces)

Right-click into the diagram window => Visible Traces

Individual curves can be selected/deselected with the **CTRL** key and **left-clicking.**
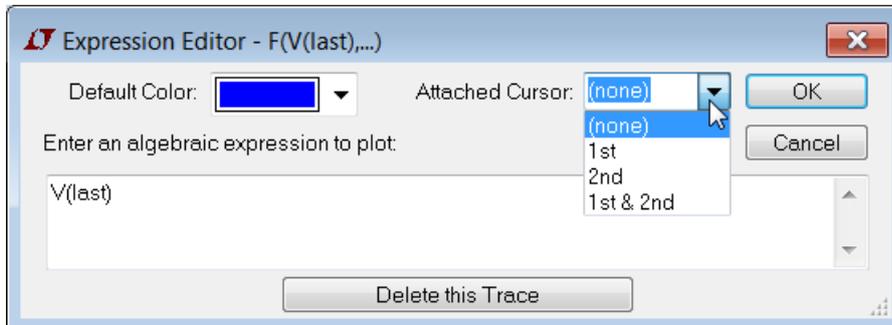
> Exercise

> Plot voltage, current, and stored energy for capacitor Cfilter.

4.2. Read out numerical values

The graphical progression of a curve shows instantly whether it qualitatively meets your expectations. For a more precise analysis, quantitative statements are often needed as well.

4.2.1. Instantaneous values (Attached Cursors)

Right-click on curve name in the diagram window opens

With **Attached Cursor the** number of cursors can be selected. With 2 it looks e.g. like this
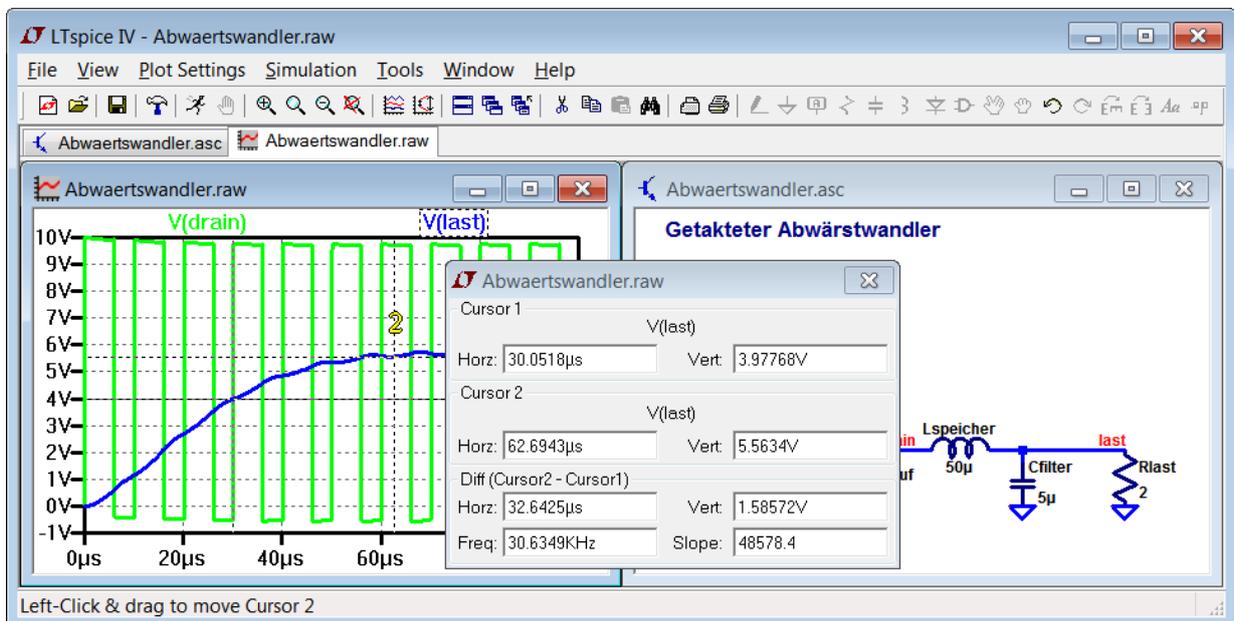
The cursors are marked with 1 resp. 2 and can be moved with the mouse.

The cursor window shows their positions and the differences.

> Exercise
>
> Find the clock frequency from the measurement curve of V(drain).

4.2.2. Mean, effective value (Average, RMS, Integral)

Pressing **CTRL** key and **left-clicking on curve name** in the diagram window on voltage, current and power of Schottky diode opens these windows
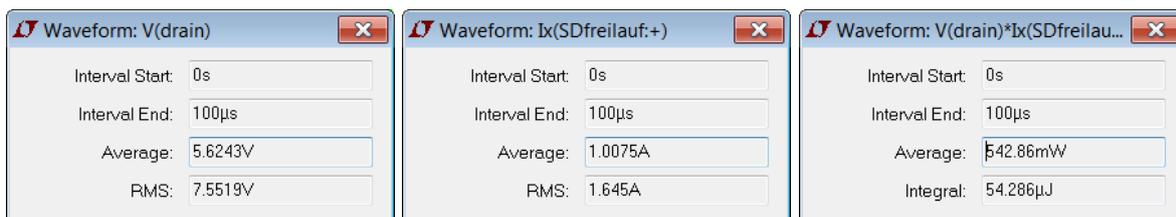
Note: Calculations are made over the entire visible time range. RMS = Root-Mean-Square, the second order mean (quadratic mean).

Exercise

What are the three mean values for the period 80µs to 100µs?

4.2.3. Fourier Analysis (Fourier Component)

Activate Diagram window, then Menu **View => FFT** opens a dialog where you can select from which curves the frequency spectrum should be calculated.

Note: From **frequency resolution = 1/simulation time it** follows that the simulation time must be chosen relatively long, e.g. 10 to 100 periods of the lowest frequency.
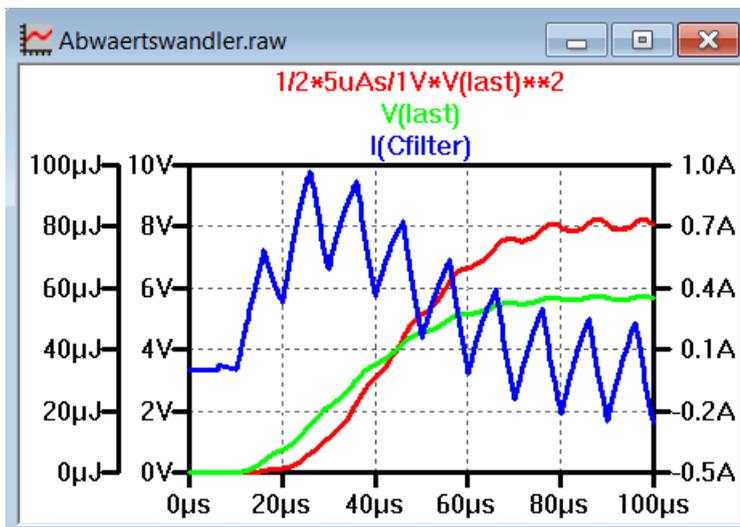
4.3. Diagram Settings (Plot Settings)

4.3.1. Set axes manually (Axis Control)

When the mouse is over an axis, a scale appears. Click to adjust it.

Exercise

Adjust the axes as follows

4.3.2. Clipping (Zooming)

Crops can be selected using the zoom tools or by manually adjusting the axes as described above.
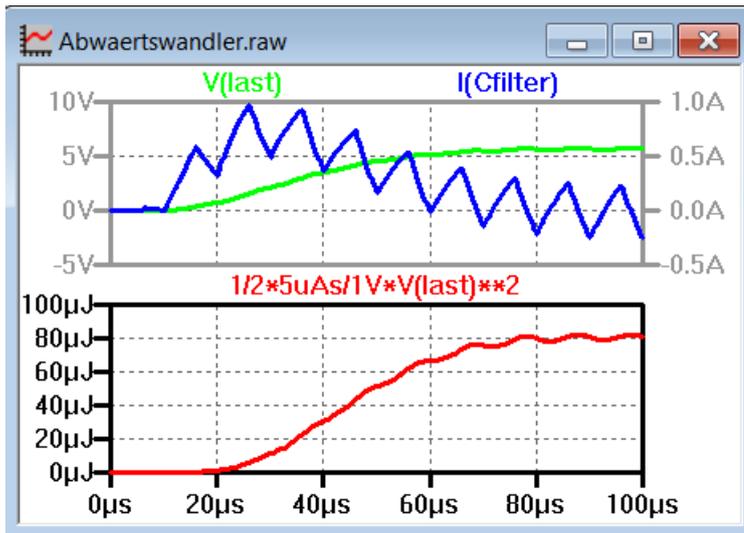
4.3.3. Separate curves (Plot Panes)

With many curves the overview suffers quickly. **Right-click in the diagram window => Add Plot Pane** opens an additional diagram.

The curves can be moved from one diagram to the other by **dragging the curve label over with the mouse**.

<u>Exercise</u>

Separate the curves like this

4.3.4.   Save plot settings (Save Plot Settings)

Activate Plot window, then Menu **Plot Settings => Save Plot Settings**

The settings can be reloaded with **Plot Settings => Reload Plot Settings.**

5.   Settings (Control Panel, Color Preferences)

At **Tools => Control Panel the** following registers are interesting:

SPICE (solver, integration method and accuracy for simulation)

Drafting Options (font size and style, line thickness in the schematic)

Waveforms (font size and style, line thickness in diagram)

With **Tools => Color Preferences** the colors for schematics and diagrams can be selected. E.g. I prefer a white background to the default setting black.

6. Special simulation elements

The SPICE elements can be parameterized in great detail. In addition to the usual mathematical functions, special "simulation" functions are also available. In other words, SPICE can be used to simulate almost any electronic circuit, if you knew how.

In this chapter I show some powerful and at the same time simple applications.

6.1. Simulation without operating point calculation

Usually SPICE first calculates the DC operating point before the transient analysis to determine the initial values for the simulation. For the operating point calculations, capacitors are considered as interruptions (current = 0) and inductors as short circuits (voltage = 0).

For clocked circuits e.g., the preliminary operating point calculation is meaningless, since by definition some voltages are switched over quickly.

Skip Initial operating point solution turns on all sources simultaneously and instantaneously at the start of simulation.
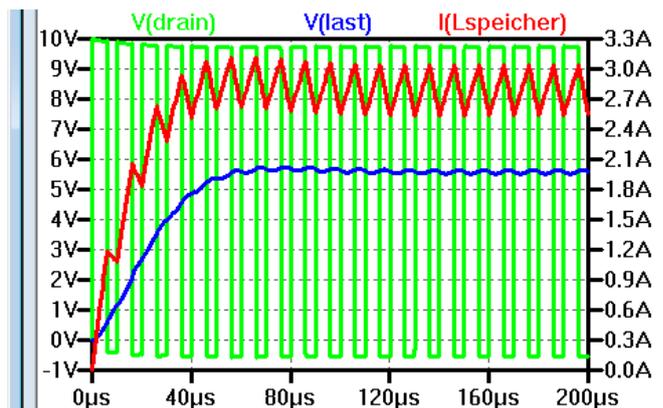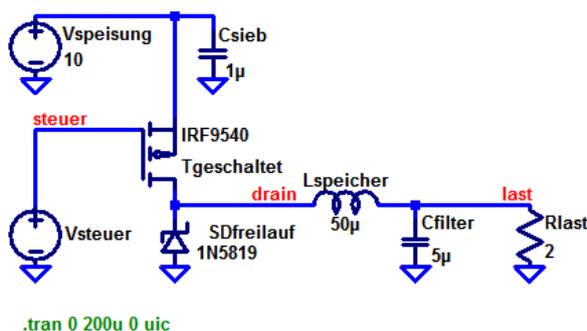
Start external DC supply voltages at 0V switches on the DC voltage sources within 20µs at the start of simulation. This prevents current peaks during the initial charging of capacitors.
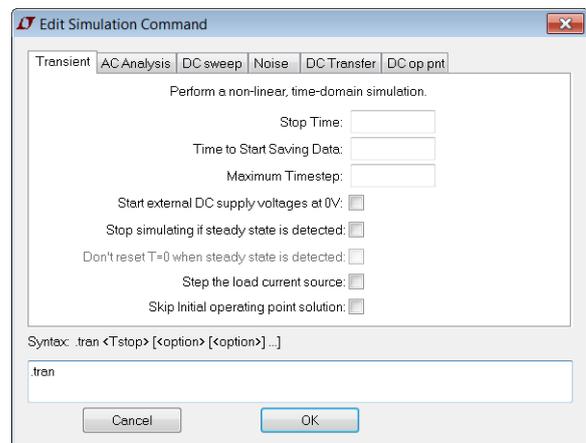

6.2. Initial Values (.ic = Set Initial Conditions)

Often, transient processes take a relatively long time and require a correspondingly large amount of simulation time.

It is much faster to set reasonable initial values for node voltages (for capacitors) and for coil currents with which the simulation should start.

Reasonable initial values can be obtained from a one-time long simulation, from given boundary conditions or specifications.
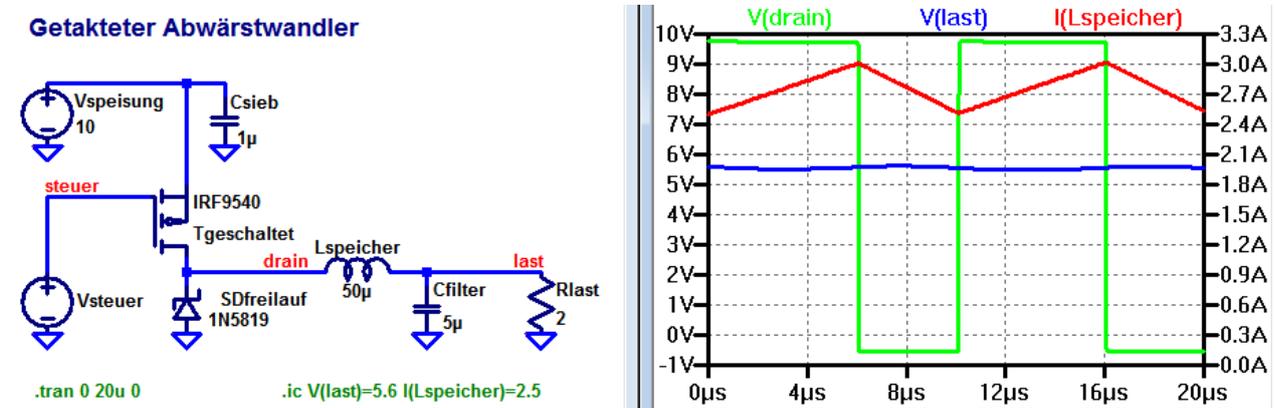
With the SPICE command **.ic V(last)=5.6 I(Lspeicher)=2.5** the simulation starts with a current of 2.5A through the coil Lstorage and with a voltage of 5.6V at the capacitor Cfilter.

Note that neither **Skip Initial operating point solution** nor **Start external DC supply voltages at 0V** must be selected in the **.tran** simulation command if initial conditions are to be considered.

6.3. Compare circuit variants

If you want to **compare the** behavior for different **component values**, you have to configure the components accordingly using **SPICE commands and parameters.** Some frequently used elements are presented here.
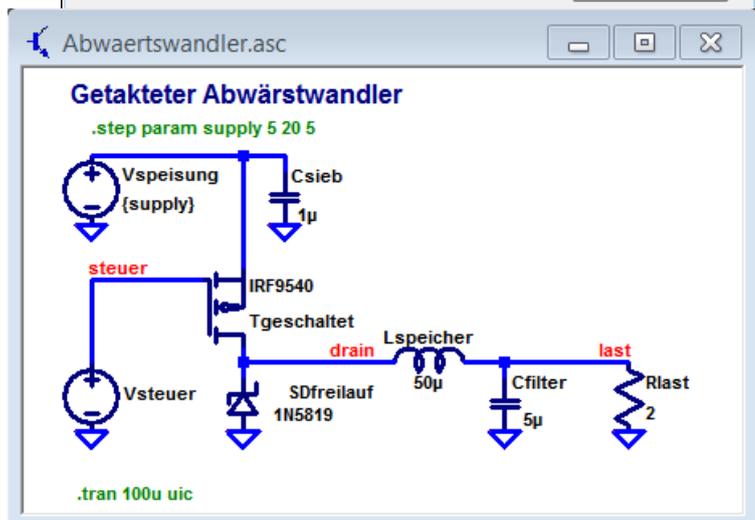
6.3.1. Parameterize voltage source

For example, to see how the buck converter behaves at different supply voltages, one must parameterize the source Vspeed. Two steps lead to the result.

Right-click on the source Vspeisung and enter as DC value [V]: **{supply}.**
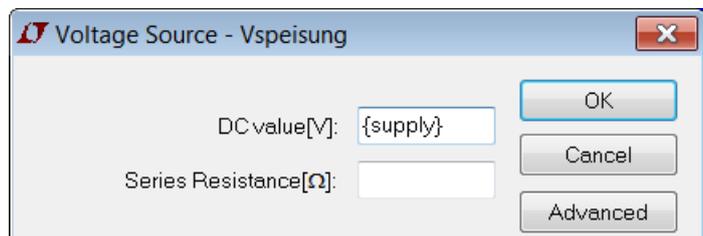
The **curly braces {...}** mean that the expression will not be evaluated until simulation time. **supply** is an arbitrary name you decide to use.
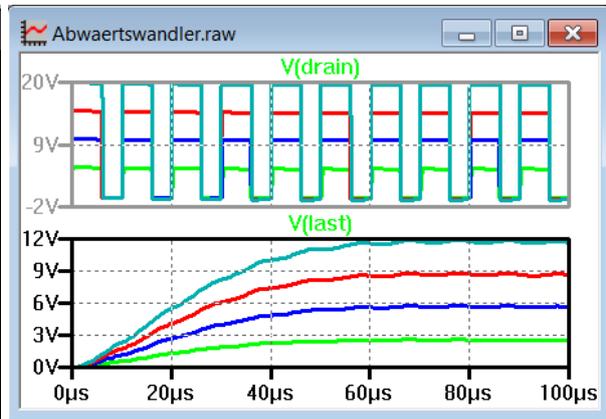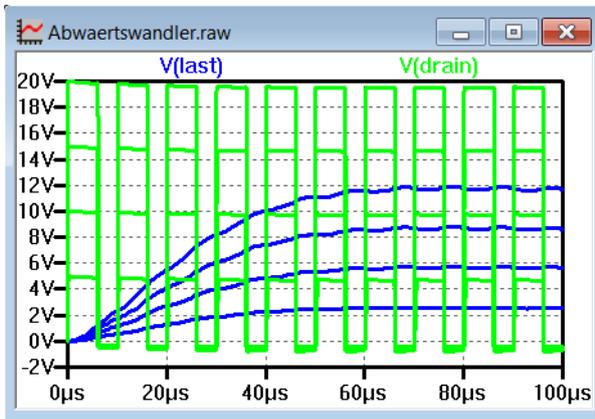
To define what is to be done with the expression **supply**, place this command in the schema with **.op** (**Edit => SPICE Directive**) **.step param supply 5 20 5**

.step is the command; **param** is followed by the parameter name, from 5 to 20 with step size 5

Here are the simulation results, presented in two ways:

The diagram on the right with the separated curves is definitely clearer, i.e. always think of how to arrange plots rather than using the default plotting.

6.3.2. Parameterize pulse width

For example, to see how the downconverter behaves at different pulse widths, you have to parameterize the source Vsteuer. The procedure is analogous to the example above. Two steps lead to the result

- **Right-click on the** Vsteuer **source** and enter **{ton}** for Ton [s].

- with ᵒᵖ (**Edit => SPICE Directive)** add to schema .step param ton 1u 9u 4u
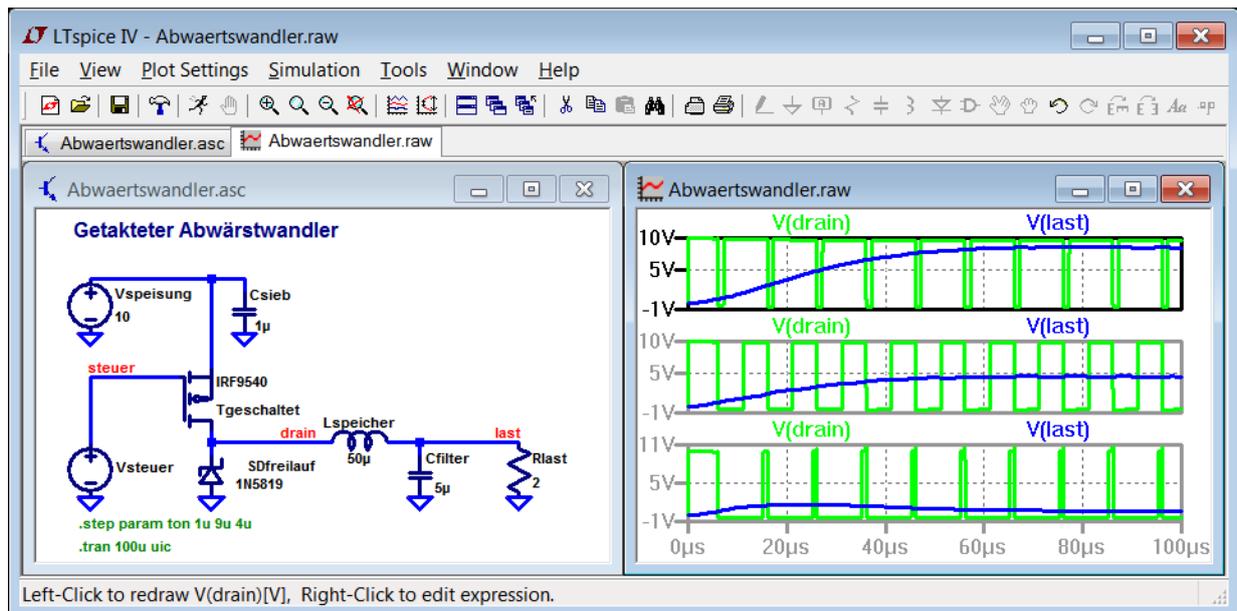
In the upper diagram, only the curves of step 1 were selected; in the middle, those of step 2 and in the lower, those of step 3.

To view the curves from step 1 in the top diagram, **right-click** on the top diagram and select **Select Steps**
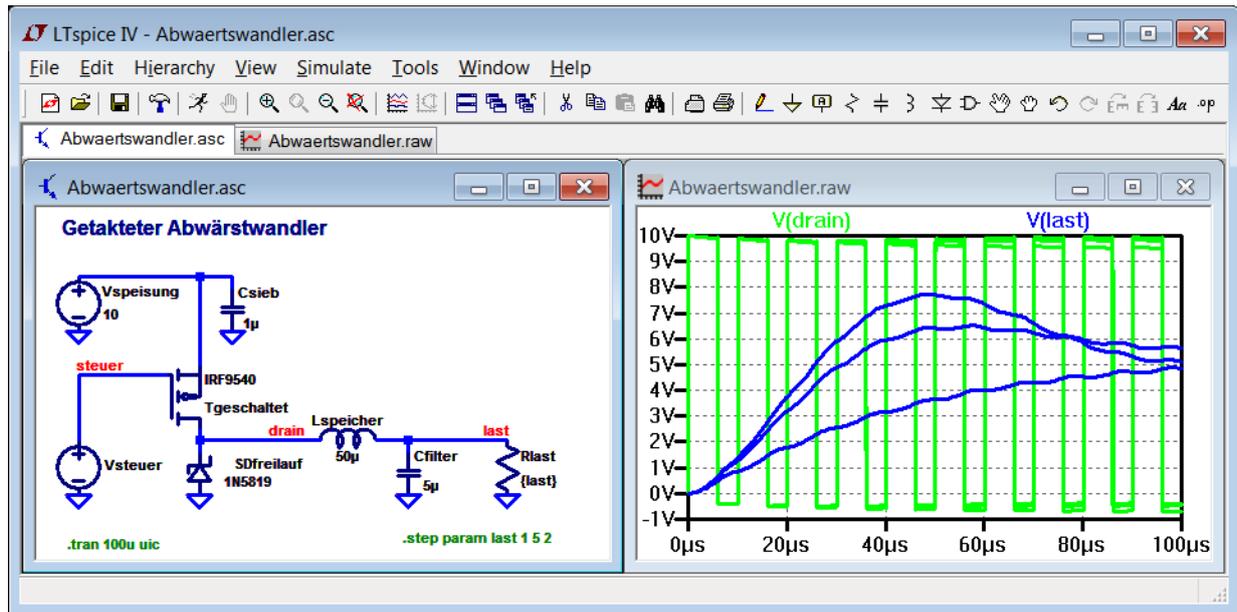
It is important to **uncheck Apply to All Panes at** the bottom of the dialog window.
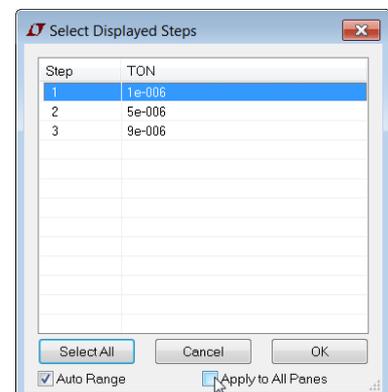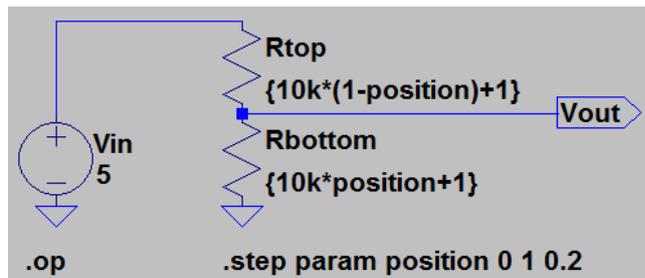
6.3.3. Parameterize resistance

The procedure is exactly the same as described for the voltage source.

E.g. compare different load resistors

6.3.4. Potentiometers

are formed with the help of two resistors parameterized in opposite directions.

position	0	0.2	0.4	0.6	0.8	1
Rtop	10001Ω	8001Ω	6001Ω	4001Ω	2001Ω	1Ω
Rbottom	1Ω	2001Ω	4001Ω	6001Ω	8001Ω	10001Ω

To exclude a possible division by 0 during the simulation, 1 Ω is added to each resistor.

6.4. Changes in function of time

How does the behavior change when the **load or signal changes**? An element in the circuit can be told that a value changes in the course of the simulation by continuously recalculating it using the **SPICE functions and the variable** *time*. Some frequently used elements are presented here.

6.4.1. Timed voltage sources

are already available as a standard in SPICE. All you have to do is select **PULSE**, **SINE** or **PWL** as the source function.

Take notice of the parameters **delay** (activate signal delayed) and **Ncycles** (deactivate signal after a desired number of periods) which may come in handy.

### 6.4.2. Timed current source (as load)

To investigate the control behavior, a current source is often used as a load instead of a resistor. For example, the reaction to a load step is interesting.

The procedure is analogous to that described for the time-controlled voltage source.

Example:

Current during 50µs 1A, then for 20µs 3A and again 1A with repetition every 40µs

If you select "**This is an active load**", it is guaranteed that the source behaves as an active load. I.e. at voltages ≤ 0 the current = 0.
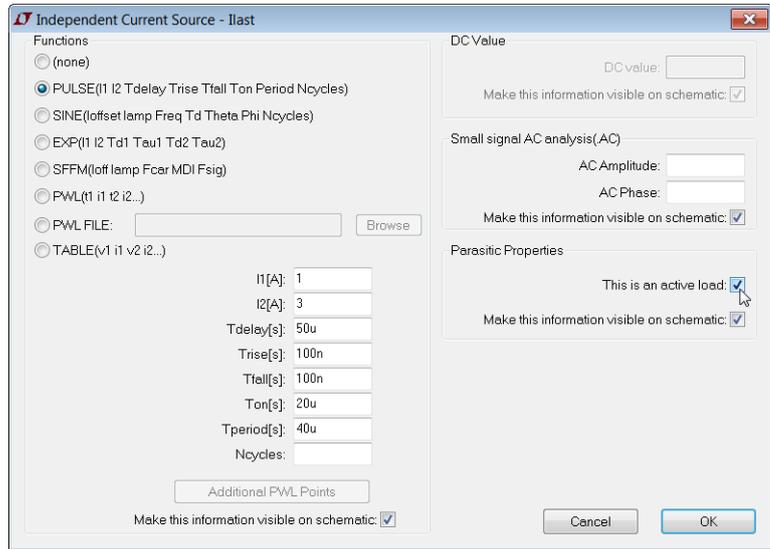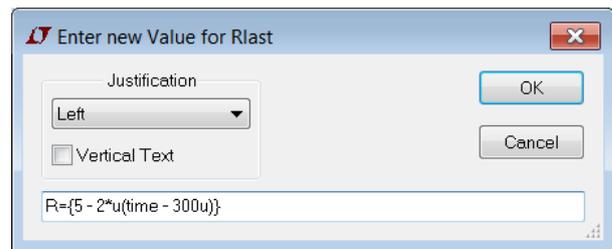

6.4.3. Timed resistance

Illustration based on an example:

For the step-down converter, the resistance Rlast should be 5Ω during 300µs and 3Ω afterwards.

The resistance value must therefore be described by a formula with time as a variable.
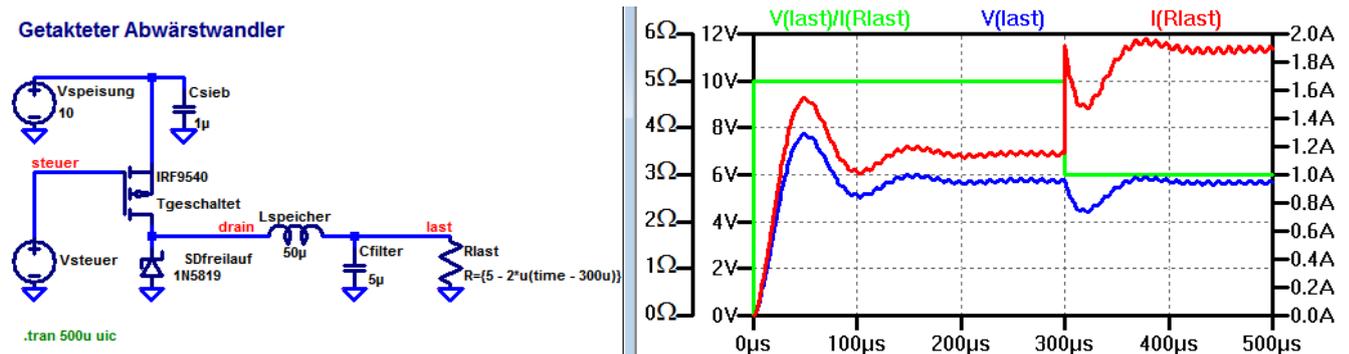
Right-click on the value of the resistor Rlast and enter **R={5 - 2*u(time - 300u)}.**

The function u stands for unit-step: u(x) = unit step, i.e., 1 if x > 0, else 0.

With the variable time = simulation time, the resistance values are calculated as follows

time ≤ 300µs => R = 5 - 2*0 = 5Ω and time > 300µs => R = 5 - 2*1 = 3Ω

Hint:
You can find the available functions in **Help => Help Topics => Register: Index => .PARAM**

6.4.4. Timed switch

Time-controlled switches do not exist as a SPICE element. They are often implemented with a voltage-controlled switch and a voltage source with desired timing.

Instead, it seems simpler to me to use a time controlled resistor (description see above) with a large (switch open) and small (switch closed) value.

Example: A "switch" which closes after 10ms: Resistance is first 1MΩ and "switches" after 10ms to 1mΩ.

6.5. Transformers

are somehow hard to get running in LTspice. Here is a tutorial: Using Transformers in LTspice
http://www.analog.com/en/technical-articles/using-transformers-in-ltspice-switcher-cadiii.html

6.6. Accelerate simulation

Depending on the circuit and the input signal, the calculations may converge slowly.

Accelerate convergence with:

- In the **Tools => Control Panel** menu, **click on the** SPICE tab and select **Alternate** for Solver.

- If this is not enough, **Abstol: 1e-011** and **Reltol: 0.01** can be set additionally, also in the **SPICE** register.

7. Help and tutorials

There is an official LTspice Getting Started Guide (slides 14 - 38) in English:
http://www.analog.com/media/en/simulation-models/spice-models/LTspiceGettingStartedGuide.pdf

Mathis Nussberger has written a German short tutorial
http://dublin.zhaw.ch/~hhrt/LTspice/LTSpice%20Tutorial%201%20V03.pdf and for advanced
users a tutorial for including additional Spice models
http://dublin.zhaw.ch/~hhrt/LTspice/LTSpice%20Tutorial%202.pdf

I often use the LTspice internal help (F1 key).

Additionally there is a detailed German tutorial http://www.gunthard-kraus.de/LTSwitcherCAD/CD_LTSwitcherCAD/pdf-file/LTspice_4_d.pdf.