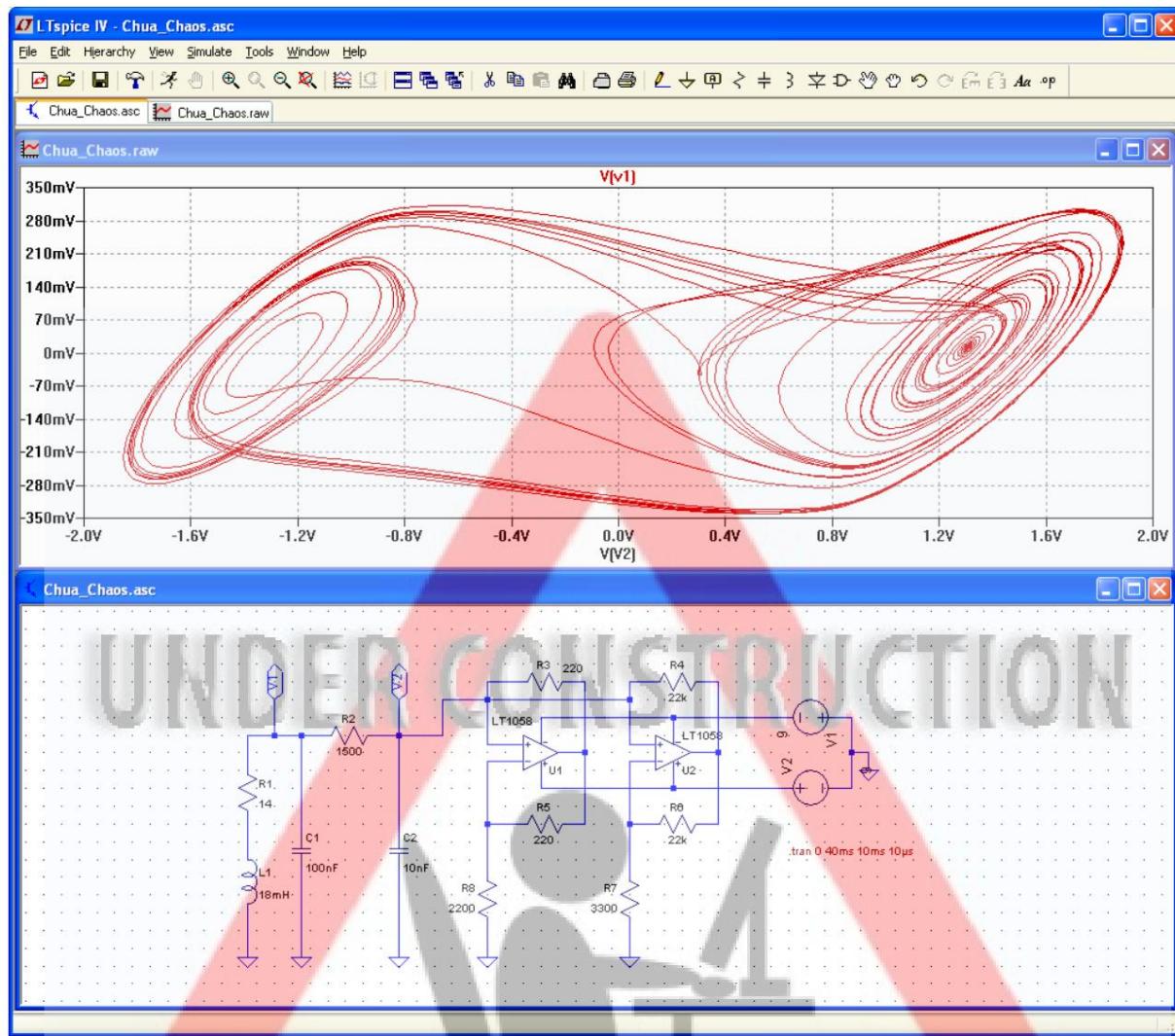


Circuit simulation with LTspice (XVII)



Physical implementation of chaos (Prof. Leon Chua, University of California, Berkeley)

Version 1.0 03/2011
 Version 2.0 09/2016
 Version 2.1 11/2019
 Version 2.2 01/2023
 Version 2.3 06/2023
 Version 3.0 12/2023

Restructuring and additions
 minor additions (thu; new / own components)
 Drafting chapters 10/11/12
 Restructuring and additions (8 ý 9; edit)
 Chapter 13 example circuits, minor restructuring and additions

• Table of contents

Circuit simulation with LTspice (XVII).....	1
1 Preliminary remarks.....	4
download and installation.....	4
(manufacturer information FAQ).....	4
Suggestions and suggestions for improvement.....	4
2 Overview of the most important types of simulation.....	5
3 Interface – Operation/Help.....	6
of the menu bar.....	6
Panel.....	7
Manual.....	7
4 Draw a circuit diagram.....	8
Libraries.....	8
Components.....	8
sources.....	9
Connections.....	9
potentials	9
Modify circuit diagram.....	10
5 Component attributes, net names, net list, texts.....	11
and values.....	11
Comments.....	12
names.....	13
6 Simulating circuits.....	14
command.....	14
(calculation).....	14
simulation.....	14
(DC op pnt / .op).....	14
7 Display and measure simulation results.....	15
the circuit diagram.....	15
(transmission channels, impedances).....	16
(cursors).....	16
8 Formatting the Output/Documentation.....	17
off.....	17
Points.....	17
graph.....	17
types.....	18
plots.....	18
axes.....	18
elements	18
documents.....	19
window.....	19
9 beginner examples/types of analysis.....	20
Point.....	20
Sweep.....	21
9.1 Operating	
9.2 DC	

9.3 Time Domain (Transient) (Large Signal Analysis).....	22	9.4 AC Sweep (Small Signal Analysis).....	22
.....	25	9.5 Measurement series - parametric analyses.....	26
10 Model / Symbol / Netlist / Subcircuit / Library.....	27	10.1 Storage locations of component models and libraries.....	27
.....	28	10.2 Models of the Spice basic elements.....	28
.....	29	10.3 Circuit diagram symbols.....	29
.....	29	10.4 Netlists.....	29
.....	29	10.5 Subcircuits.....	29
.....	30	10.6 Libraries.....	30
11 Hierarchical Blocks / Designs with Multiple Pages.....	30	11.1 Create symbol.....	30
12 Integrate or create new / own components.....	31	12.1 Add Spice Model.....	31
.....	31	12.2 Add subcircuit.....	33
.....	35	12.3 Add components as hierarchical blocks to the library.....	35
13 example circuits.....	36	13.1 Examples in the .../LTspice/examples/Applications folder.....	36
.....	36	13.2 Examples in the .../LTspice/examples/Educational folder.....	36
.....	36	13.3 Switch.....	36
.....	36	13.4 Transistors as switches / High side – Low Side.....	38
.....	38	13.5 Operational amplifiers.....	39
.....	39	13.6 Transformers / Transformers.....	41
.....	41	13.7 Solar cells.....	41
14 Additional information and techniques.....	42	14.1 User-defined functions and parameters / Plot Definitions File	42
.....	42	14.2 Option uic - Skip initial operating point solution (transient analysis).....	42
.....	42	14.3 Option plotwinspace=0.....	42
.....	42	14.4 FFT ѕ GK.....	42
.....	42	14.5 Transformer ѕ GK xy, TUM 3.1.....	42
.....	42	14.6 Controlled sources ѕ TUM 3.2.....	42

1 Preliminary remarks

LTspice is a further development of the SPICE program (Simulation Program with Integrated Circuit Emphasis), which was developed at the University of California in Berkeley, USA in 1972. The leading

"LT" stands for the manufacturer "**Linear Technology**", which produces these

Software made available free of charge. In 2016, Linear Technology was acquired by **Analog**

Devices. The LTspice libraries primarily contain components from this manufacturer and standard types such as resistors, capacitors, coils, diodes...

In principle, the software has no restrictions regarding the complexity of the circuit to be simulated. Additional "SPICE" libraries (*from other manufacturers*) can be integrated.

1.1 LTspice download and installation

LTspice can be obtained [from the manufacturer's website](#) be obtained. (for Windows, Mac OS X, Linux [Wine](#))

1.2 LTspice - License and distribution (manufacturer information FAQ)

1.2.1 Can I re-distribute the software?

Yes, you can distribute the software freely whether you are a Linear Technology customer or not. See the license section for more details.

Technical support for non-Linear Technology customers is purely discretionary.

1.2.2 Is it a shareware, freeware or demo?

This program is not a shareware or a demo. It is fully functional freeware.

The purpose of this software is to help our customers use our products.

It can also be used as a general-purpose circuit design package with schematic capture and SPICE simulation.

We do encourage students using the program to become familiar with the analog design process.

We cannot guarantee support for non Linear Technology related program usage, but we'll fix all general program bugs and appreciate such reports.

We do extensive in-house testing and believe the program has superior convergence capability.

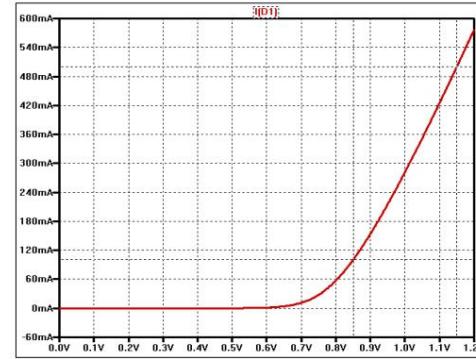
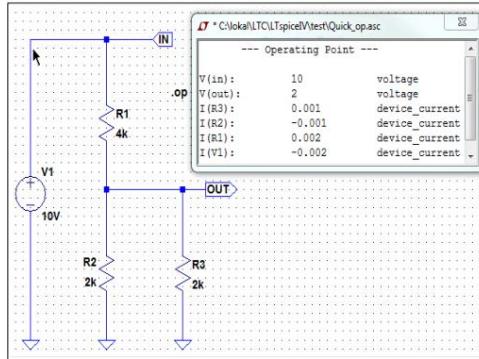
There are no known outstanding bugs.

1.3 Suggestions and suggestions for improvement

The aim of this quick reference is to provide easy-to-understand instructions for students. These should be able to familiarize themselves with the circuit simulation "independently". Suggestions on how this could be further improved are most welcome. (e-mail to: Volker.Schilling-Kaestle@thu.de)

2 Overview of the most important types of simulation

SPICE offers different types of simulation to examine circuits. In this quick reference, the most important basic types are presented using examples.



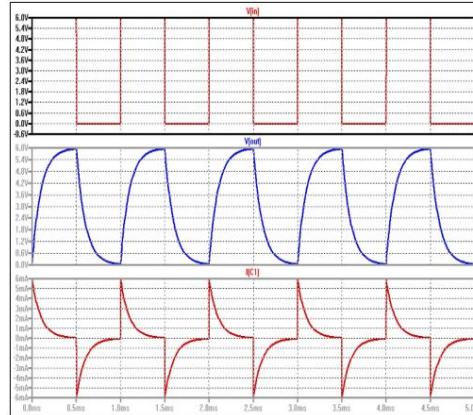
DC op pnt: -

Operating point analysis with fixed values of all DC sources.

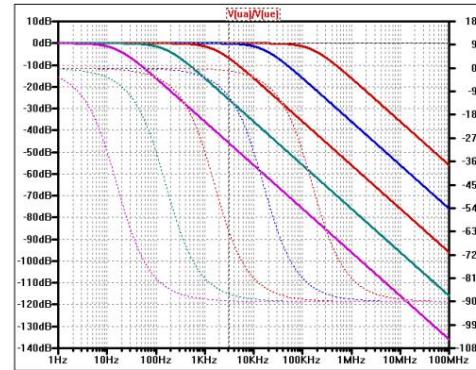
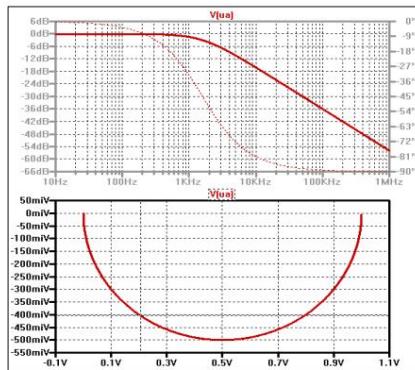
DC Sweep: -

Series of operating point analyses by varying one or more sources.

- Generation of characteristic curves (e.g. diode)



Transient analysis: - for processes in the time domain



AC Analyse:

- Small signal alternating current analysis - for frequency-dependent behavior

Parametric analysis:

- Multiple execution of an analysis
Variation of a parameter

3 Interface – Operation/Help

The program is started with **/ Start / Programs / LTspiceIV / LTspiceIV v.4.xxx/** started.

The interface contains the menu bar, the tool bar for the most important functions and a status bar (at the bottom) to display additional useful information.

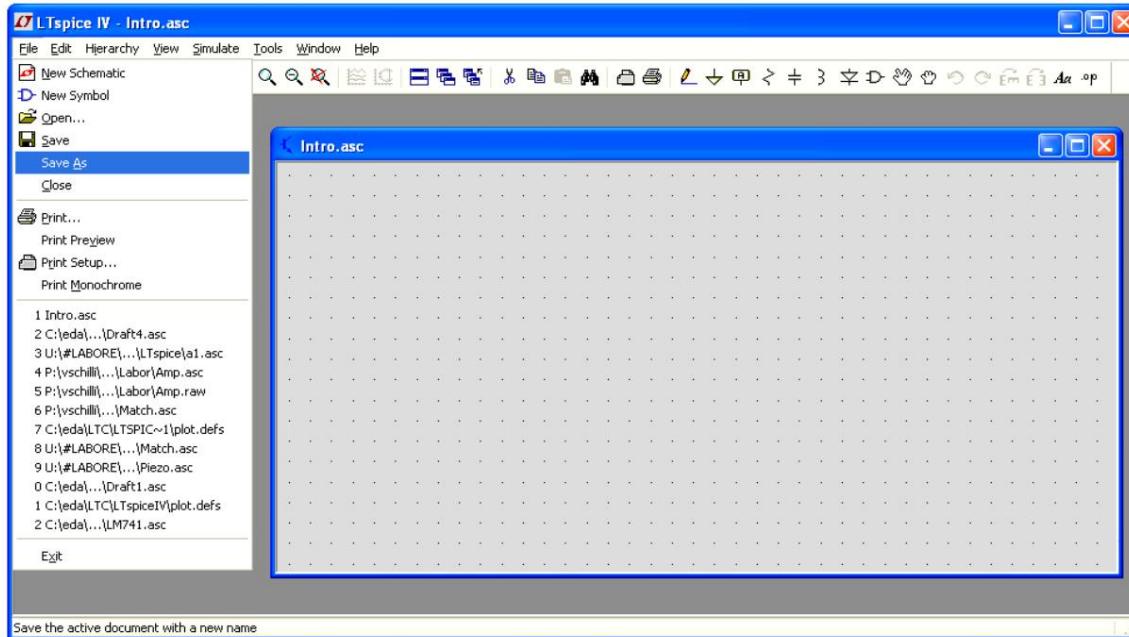
With **/ File / New Schematic** or a new window is opened which shows the drawing area for the circuit to be simulated. The window can be minimized, maximized and moved using the usual controls.

The newly created plan was automatically assigned a file name "**Draft x .asc**".

If you simply save, this file would end up in the LTspice installation directory.

Therefore, it is best to save the (still empty) plan immediately under a meaningful name and in a useful location in order to reduce the waste in the program directories and increase the chance of finding the simulation again if necessary.

The most recently opened documents are listed as "History" at the bottom of the **/ menu File /** displayed.



This makes it easier (*if the name is sensible ;)* to find previous simulations.

3.1 Operation / tools of the menu bar

Most of the icons in the menu bar are relatively easy to understand. You will also receive a message when the mouse pointer is on the corresponding icon for a while.

It also makes sense to memorize a few [**shortcuts**] that are at least easy to use

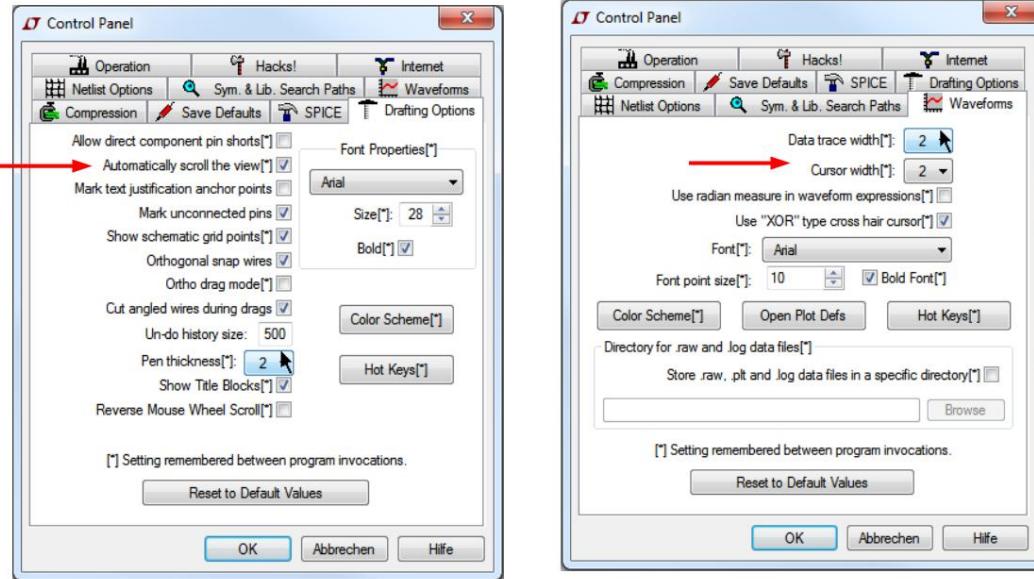
Make it easier for users who have two hands. (see 3.2 Help)

A click of the **right mouse button** in an empty area often helps.



3.2 Settings / Control Panel

In the control panel, which can be accessed via the symbol in the menu bar, there are a few tabs with setting options for various topics.

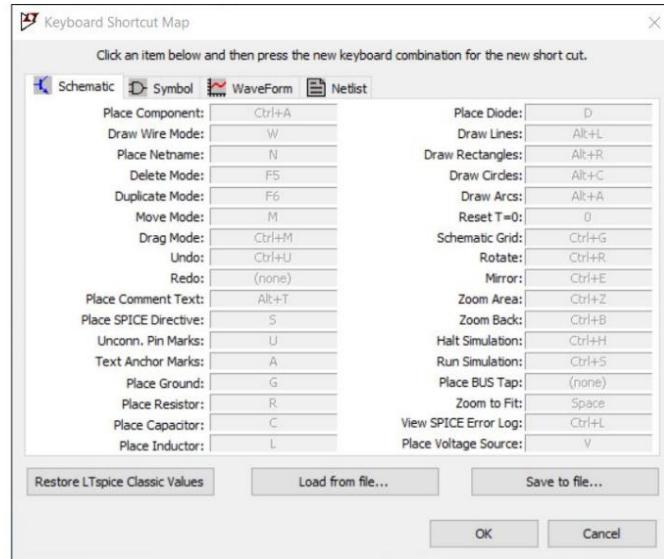


The **Drafting Options** and **Waveforms** tabs are particularly interesting for beginners.¹

Here you can, for example, set colors, **line widths** (1->2) and grid, or adjust the keyboard shortcuts (**hot keys**) according to personal preferences.

You can also turn off the potentially irritating automatic window **scrolling** here.

The other tabs are more suitable for more experienced users.



The configuration of the **Hot Keys[*]** or **Keyboard Shortcuts[*]** can also be configured in both settings menus. can be called up to adapt them to your personal taste or to set them up so that they are particularly easy to remember.

From LTspice version 17.1 also exists the possibility of importing and exporting shortcuts.

3.3 Help [F1] / Manual

The “User Guide” available in earlier versions is completely included in the help.

Help can be accessed via **[F1]** or **/ Help / Help Topics** / be called. Even when using LTspice for the first time, it doesn't hurt to take a look and get an overview. (e.g. via the setting options in the control panel or shortcuts)

4 Draw a circuit diagram

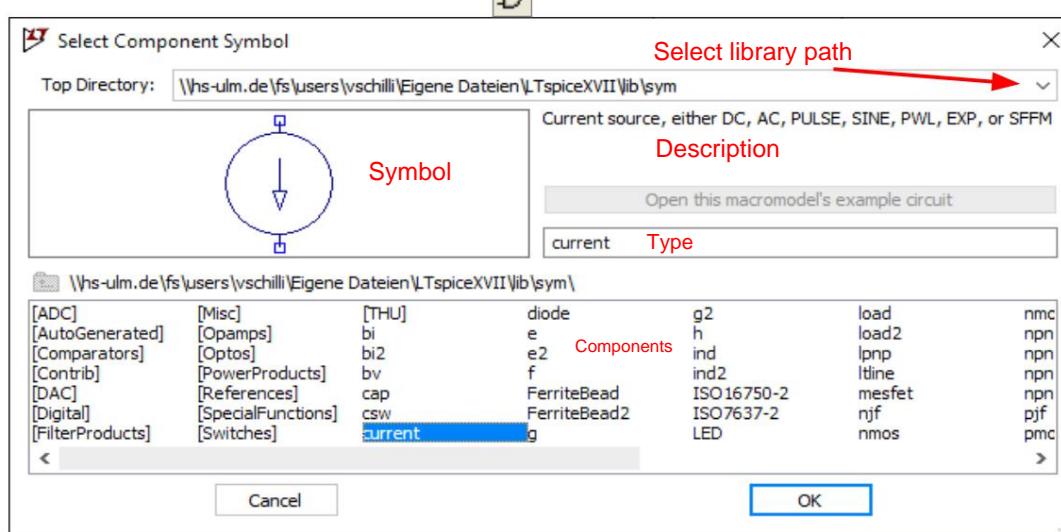


4.1 Libraries

The libraries of the LTspice installation (`.../LTC/LTspiceXVII/lib`) are installed on the first start of the software and copied by a new user into his user directory and only this Copies are then used when creating new designs. Under Windows, these copies are located in the path: `.../users/<username>/My Documents/LtspiceXVII/lib`.

The directory in which the circuit diagram was saved can also be selected as a further source. Any additional libraries should be in one of the both paths.

You can access the component libraries via the `/` menu **Edit / Component**, the function key **[F2]** or the symbol in the toolbar.



The components and, if applicable, the subdirectories [...] in the selected library path are displayed at the bottom of the library browser.



4.2 Components

Use the mouse to find the required component in the library. The symbol for the graphical representation of the component then appears in the gray part of the window and to the right of it a short description with the name of the component.

The standard types for R, C, L and diodes can also be found directly using the symbols in the toolbar or the buttons **[R]**, **[L]**, **[C]**, **[D]**.)

After confirming the dialog with "OK", the component can be placed on the drawing sheet. While moving, the component can be moved using the key combination **[Ctrl+R]**, rotated and mirrored with **[Ctrl+E]**. These key combinations are used during the Also displayed at the bottom of the LTspice status bar when moving!

Type **Ctrl+R** to rotate or **Ctrl+E** to mirror.

Another option for rotating or mirroring is in the menu bar.



The mode for adding components can be ended with the **[ESC]** key or the right mouse button.





4.3 Signal sources

Signal sources are in the same library and are treated in the same way Components. The most important standard sources have the names “**voltage**” and “**current**”.

In addition, there are several other and controlled sources (**e, f, g, h**). If you need it or have the opportunity, just browse through the help [**F1**] in LTspice



Voltage sources can also be added using the [**V**] key.



4.4 Connections

To do this, use the “Wire” command from the / menu **Edit / Draw Wire** / the function key [**F3**] or the button in the toolbar is used.

The mouse pointer then changes to the crosshairs, which you place on a connection of the component and press the left mouse button to start the connection.

By clicking with the left mouse button you can create a corner to change the drawing direction.

Left mouse button on a connection terminates the connection at the respective connection pin. Right mouse button or [**ESC**] terminates the connection at the last corner.

The free connections on components are marked by a small, unfilled square that disappears as soon as a connection is assigned to a connection.

Components should always be connected using the **Wire** command and not directly

Stacking the connections on top of each other!

Connections of crossing wires are indicated by solid square symbols.



The connections are usually only drawn horizontally or vertically. However, oblique connections are also possible if you hold down the [**Ctrl**] key while drawing .



4.5 Reference potentials

Spice simulations didn't work without reference potential (ground)!



It is imperative that every analog node has a DC path to a reference potential.



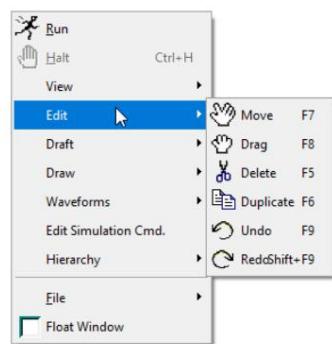
Reference potentials are set via the menu / **Edit / Place GND** /, the [**G**] or key added the button in the toolbar.



4.6 Modify circuit diagram

The tools for editing a circuit diagram can be selected via **shortcuts**, the **edit** menu, the **toolbar** or via a pop-up menu that appears when you right-click in the circuit diagram editor.

Basically, you first select the tool and then the element that you want to modify with it. The operation method differs from the operation in text editors or the like, where the object is usually selected with the right mouse button and then the action in the dialog that appears.



Edit popup menu with commands to:

- move
- cut / delete
- copy
- undo actions

 Individual elements can be edited by clicking on them, groups can be edited by opening a window with the mouse.

4.6.1 Remove (delete) elements

You can enter deletion mode via / **Edit / delete /**, The key combination **[CTRL+X]**, the function key **[F5]**, the **[Del]** key on the keyboard or the button in the tool bar. The mouse pointer then turns into a  symbol to indicate delete mode.

4.6.2 Moving elements 1 (Move)

 The move command hand symbol, which has spread index and ring fingers, is to be understood as meaning that the element to be edited is first cut out (*detached from other elements*) and can then be moved.

4.6.3 Move elements 2 (drag)

 The drag tool allows you to move elements without detaching any associated elements.

 In drag mode you can also pick up and move loose line ends. To do this you pull Use the mouse to open a small window above the end of the line.

For slanted lines, you can add corners by clicking in the slanted element.

4.6.4 Copy elements (Copy)

 The Copy tool allows you to copy individual elements as well as groups of elements within a circuit diagram and also into other diagrams.

4.6.5 Undo/Redo actions (Undo/Redo)

 These commands are of course also available and carry out the usual actions.

5 component attributes, net names, net list, texts

5.1 Component names and values

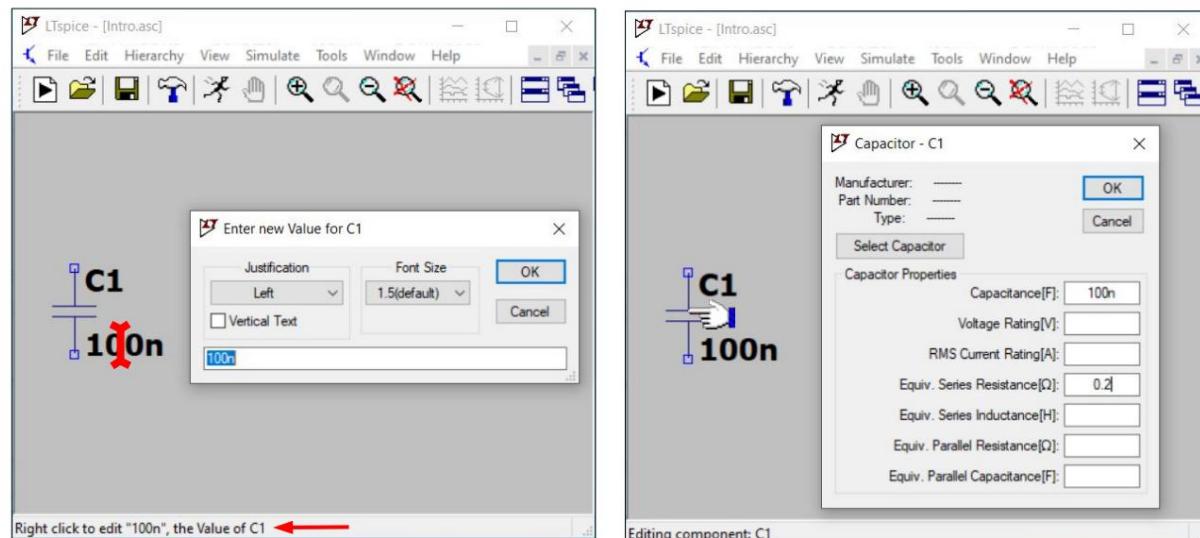
The construction of the circuit is complete when the names and values of the components are entered correctly.

Dialogs for changing attributes are opened **with the right mouse button** called when the mouse cursor is over an object.

The editor must be in basic mode. This mode can be recognized by the small cross as a mouse cursor whose shape changes like an "I" when it is moved over a text element.

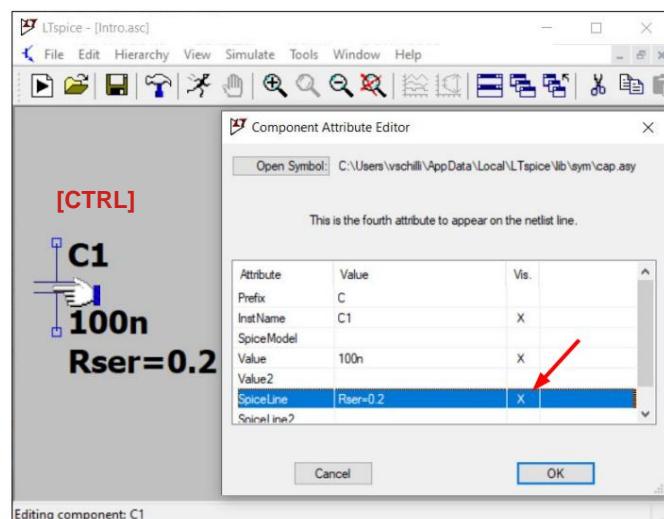
If you move the cursor over a graphic object, the shape changes to the hand 

 If the editor is in one of the other modes (*Add, Cut, Move ...*) you can use **[ESC]** or the right mouse button to go to basic mode.



5.1.1 General (Component) Attribute Editor

Additional component attributes are often not displayed in the schematic, even if they have been entered as in the dialog above. (*Series Resistance[\ddot{y}] = 0.2*)



Using **[Ctrl]+right mouse button** you can you can change the visibility of attributes in the

Component Attribute Editor dialog under the **Vis** column .

If you know the additional attributes of components, you can enter them directly in the **SpiceLine** line of the General Attribute Editor.

5.1.2 There are a few points to note when entering component values:

- Decimal number values should be separated with **a point, not a comma**, e.g. 2.7k
- Names and values cannot contain **spaces**.
- Spice is not case sensitive e.g. (M = m = milli.)
- **Units may only be added if they do not lead to conflicts with the powers of ten. (ŷ no F for Farad ...)**



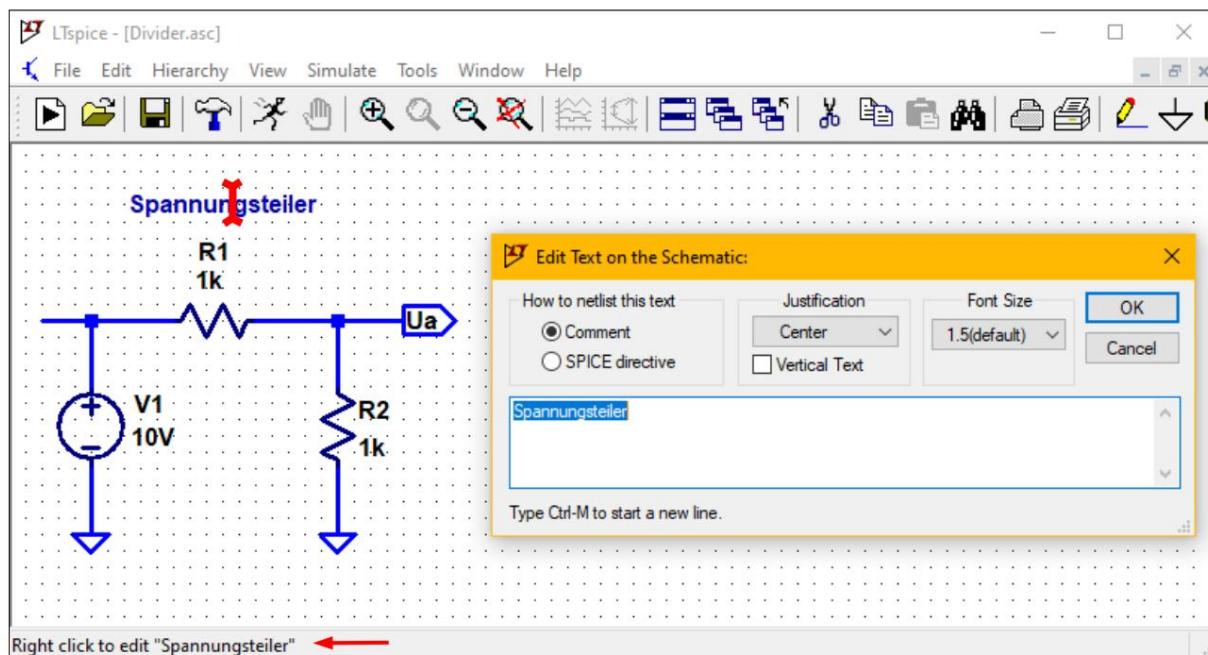
5.1.3 Factors used in LTSpice:

M = m = milli = 10 ⁻³	U = u =	K = k = kilo = 10 ³
micro = 10 ⁻⁶	N = n = nano =	MEG= meg = mega = 10 ⁶
10 ⁻⁹	P = p = pico = 10 ⁻¹²	G = g = height = 10 ⁹
= f = femto = 10 ⁻¹⁵		T = t terra = 10 ¹²

5.2 Texts/Comments



With the **text** tool [**T**] you can add comments and small descriptions to the circuit diagram.



In the input dialog, which also appears again when elements already exist by clicking with the right mouse button, the font size can be changed, among other things.

The settings regarding the alignment make little sense because **vertical text** only causes the text to rotate and not the letters to be written one below the other, as with a vertical solution word in a crossword puzzle.

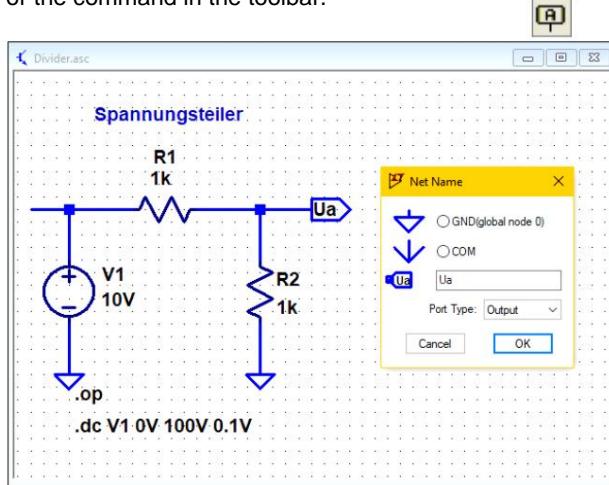
You can also rotate the text while storing it, or when moving it later using the key combination **[Ctrl] + [R]**.



5.3 Network names

To clarify how circuits work, connections can be given self-explanatory names. This also makes it much easier to assign the simulation results later.

You can access the mode for assigning network names via the menu | **Edit / Label Net** / the function key **[F4]** . or the command in the toolbar.

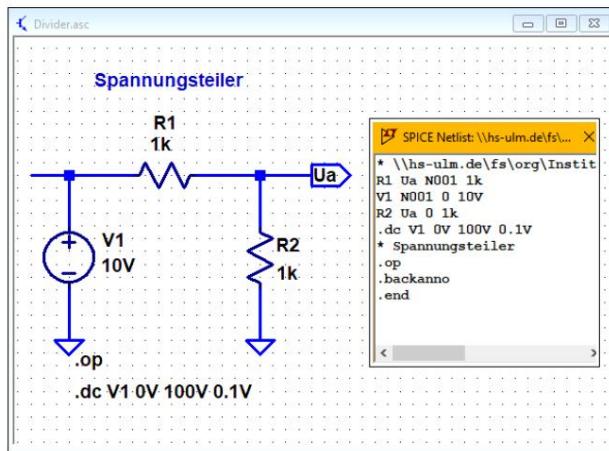


Network names at the free end of a connection can also be provided with symbols that illustrate the type of signal (*input*, *output*, etc.)

5.3.1 Netlist

For the actual simulation with Spice, a description of the circuit in text form is required. This is created automatically by LTspice before the simulation. You can also view this via the menu | **View / SPICE Netlist** | display.

The netlist must contain all the information that a spice simulation requires.



The net list contains all components ($R1$, $R2$, $V1$) with their connected networks. ($N001$, Ua and 0) and the values ($1k\ddot{}$, $1k\ddot{}$, $10V$).

In addition, you can also see instructions ($.op$ and $.dc V1 0V 100V 0.1V$) that describe what type of simulations should be carried out . Information about this will follow in the next chapter and in beginner examples/types of analysis.

Simulate 6 circuits

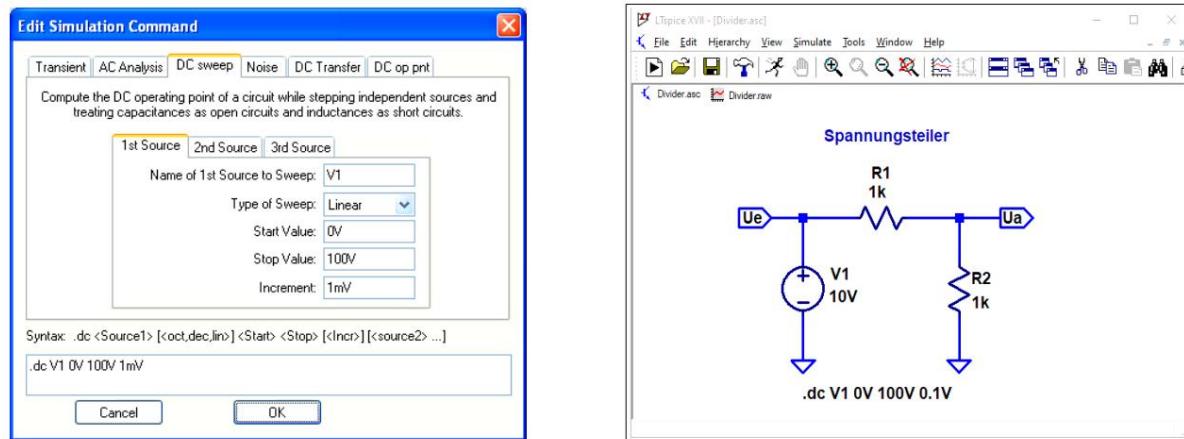
.op 6.1 Creating a simulation command

Spice enables different types of simulation, which are defined via commands.

These commands could (*assuming in-depth knowledge*) use the menu command **/ Edit .op / SPICE Directive** or the **[S]** key can be entered as a line of text.

This is also easier and clearer in the dialog **/ Simulate / Edit Simulation Cmd /** possible, in which several analysis types with different options are available to choose from. The resulting command is automatically created for the user from the parameters entered in the dialog and made available for placement in the circuit diagram with "OK".

The dialog can also be accessed via the integrated help of the SPICE Directive dialog



 become. (Right mouse click in the text field → Help me Edit → Analysis Command)

The different types of analysis are described in Chapter 9 using examples.

6.2 Start simulation (calculation).

The simulation is carried out using the command **/ Simulate / Run /** or started via the  switch. You can see the progress of the calculations in the status bar at the bottom left.

The results of the simulation include the voltages of all nodes and the currents of all meshes. Depending on the type of simulation, the results can then be output graphically or just as text.

6.3 Stop simulation

If the settings for the simulation are unfavorable, the calculations may take a very long time. In this case you can use the simulation if necessary

Hold your hand in the menu bar or use the keyboard shortcut **[Ctrl+H]**.

6.4 Operating point calculations (DC op pnt / .op)

The results of an operating point calculation are not displayed graphically, but as text in an additional window. The output lists the voltages of all nodes relative to the reference potential and the currents through all components

 The values for the objects under the mouse pointer are also displayed in the status bar.

7 Display and measure simulation results

When the simulation results are output graphically, initially only an empty plot window appears, into which the desired signals can then be inserted.

(SPICE calculates "all" signals; displaying "all" immediately would make little sense)

Using the simple example from Chapter 6, the following options can be tried out:

7.1 Signals selectable in the circuit diagram

The selection of signals to be displayed graphically can be done interactively by selecting them in the circuit diagram window. After the simulation has been completed and a window is opened for the display, the mouse cursor takes on three new appearances.

If the mouse pointer is moved over connections, the display changes to that of a probe for detecting **voltages**.



Voltage probe cursor

With the left mouse button the voltage signal of this node is converted into the graphic display adopted.



Difference signals can be displayed if the button is pressed at the first node and only released when the probe pointer is above the second node.



Current probe cursor

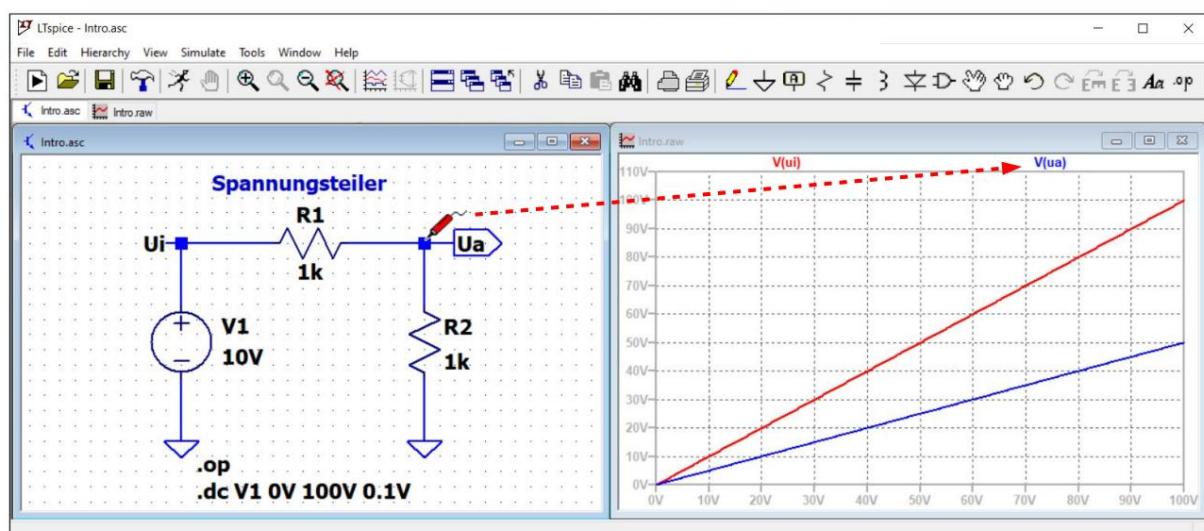
The symbol of a clamp meter is displayed above a component (or connection). With the left mouse button the signal is included in the display.



If you hold down the **[Alt]** key, you can also record the current in a network segment.



The power loss of a component is determined by pressing **[Alt]**
Button selected when the cursor is over the component.



7.2 Representation of functions (transmission cables, impedances)

If not only simple signals are to be displayed, but also the transfer function as the ratio of the output voltage to the input voltage, for example, you can follow the plot using **[Ctrl+A]** the menu / **Plot Settings / Add Trace** / or **[Ctrl+A]** add an “Expression”.

This expression can be a simple formula such as **V(out)/V(in)**, or it can already be one defined own function like **F_(V(out),V(in))** (see also [6.2.1 TODO](#))

 SPICE also has some standard functions such as **sin()**, **cos()** ... which can be queried via the online help ([F1] :-)

7.3 Measuring the graphical output (cursors)

7.3.1 Mess-Cursor

There are up to two measurement cursors in the plot window. The first cursor can be attached to a signal by left-clicking on the signal name.

With the right mouse button you can add a signal via the drop-down menu under **“Attached Cursor”** both cursors can be assigned in any way (*none, 1st, 2nd or 1st & 2nd*)

The cursors can be controlled with the mouse or with the **[ÿ]**, **[ÿ]** keys.

You can **use the mouse** to drag the cursors horizontally, **i.e. not on vertical lines**.

when this is moved over a cursor and then the respective number appears as the mouse cursor. Then press and hold the left mouse button to drag.

Using the keys, the last cursor moved with the mouse becomes the next value
Simulation controlled. (*Be careful with the [up] button ;-*)

7.3.2 Maus-Cursor

Regardless of the use of a special measurement cursor, quick measurements can also be carried out with the mouse.

The coordinates of the mouse pointer are always displayed at the bottom left of the status bar.
Differences can be measured by opening a window (*as when zooming*).

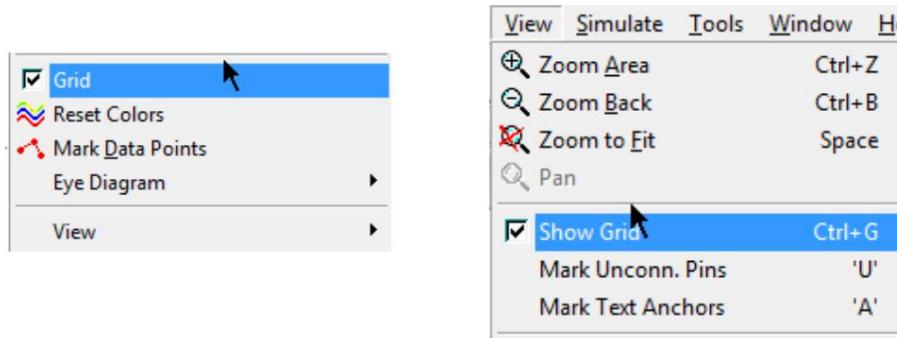
 Zoom is not executed unintentionally if the **[Esc]** key or the right mouse button is pressed. The view then remains unchanged.

8 Formatting the Output/Documentation

8.1 Switch grid on/off

In the waveform viewer and circuit diagram editor you can either switch the *grid on* or off. To do this you can use the key combination **[Ctrl+G]**.

In the Waveform Viewer, this can also be done using the pop-up menu that appears when you right-click somewhere in the plot window. In the schematic editor via / **View** / in the menu bar.



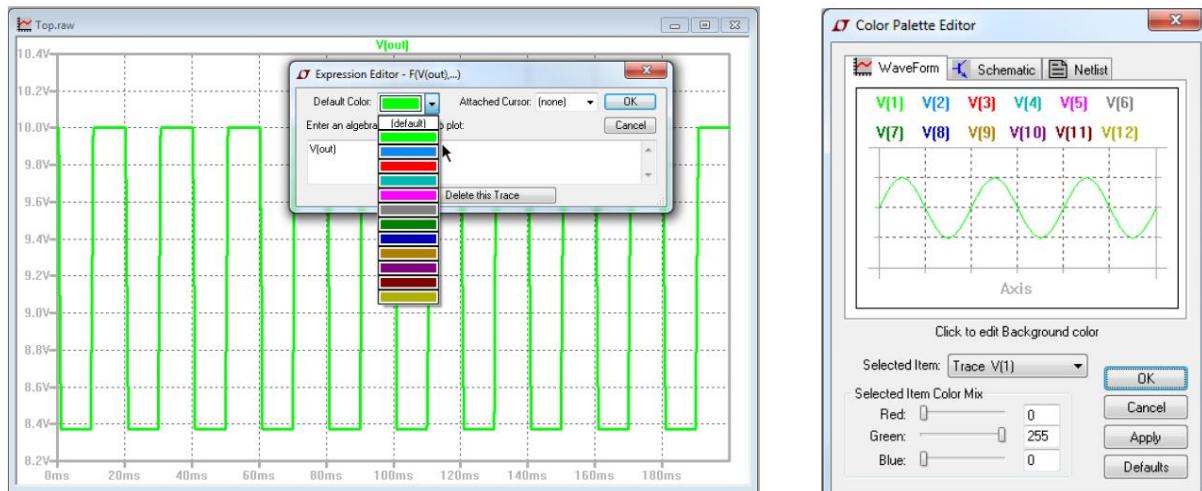
8.2 Mark Data Points

The graphs in the Waveform Viewer are created by connecting the points calculated by Spice with lines. Sometimes it can be useful to view the calculated points to check that they are not too far apart in a section of interest. This goes over / **Mark Data Points** / in the pop-up menu shown above.

8.3 Change colors of the graphs

The color of the graphs in the Waveform Viewer can be changed very easily. To do this, right-click on the signal name. You can then select the desired color in the pop-up menu. The colors defined in the palette are available for selection.

If other colors are desired, for example because only light colors are available and the background is also light, then you can change the color palette via the **control panel** or via the menu / **Tools / Color Preferences** / set.



8.4 Axis scaling / display types

The scaling of the axes can be done via the menu **/ Plot Settings / Manual Limits** | can be set globally for a maximum of 2 Y-axes and the X-axis.



When the mouse cursor is over the scale of an axis and turns into a ruler, depending on the version of LTspice, clicking with the left (*I/V*) or right (*XVII*)

Click on the mouse to open the settings menu for this axis.

In addition to axis scaling for different types of simulation, the setting options also include different display types.

For AC analysis, for example, this can be represented as a Bode plot or locus curve.

When displaying a signal with magnitude and phase or real and imaginary parts, the components can be switched off or on via their respective axes.

8.5 Additional Plots

If the curves shown in the graphic have different sizes, for example due to low amplitudes, then they cannot be viewed correctly. They can be accommodated in a second, separate plot and therefore scaled independently of each other.

Inserting another plot is done via the **/ menu Plot Settings / Add Plot Pane** | deleting via **/ Plot Settings / Delete Active Pane** |.

Signals can be moved between plots by dragging with the mouse.



If **[Ctrl]** is pressed while moving , a copy of the signal is created.

8.6 Separate X-axes

When using multiple plots, different scalings can also be used for the x-axes. This is useful, for example, if there is a signal overview at the same time

and should be displayed in detail (zoom) . To do this, the checkbox in the menu **/ Plot Settings / Sync. Horz. Axes** | be removed.

8.7 Inserting comments and drawing elements

/ Plot Settings / Notes & Annotations | provides various elements to develop the plot.

However, since the plots for documentation are usually inserted as a bitmap into an external document, it is often easier and more flexible to insert any additions using the external software.



/ Plot Settings / Notes & Annotations / Label Curs. Pos. | can be used to mark a point on a curve including the associated values.

8.8 Documentation / transfer to external documents

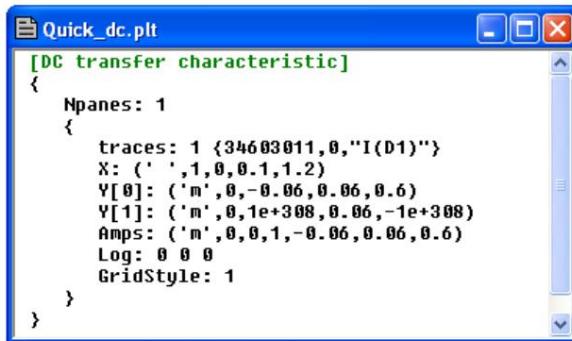
The simplest option circuits as well as the output plots of the simulation results
 The menu offers the option to transfer data to other documents for documentation purposes
| Tools | Copy bitmap to Clipboard |.

Here the entire visible area (including any other windows ;-) that is within the active window is copied to the clipboard as a bitmap.

8.9 Saving the settings of the output window

The settings of the curves displayed in the Waveform Viewer can be changed in the menu
| Plot Settings | Save Plot Settings | be saved for later reuse.

A text file with the ending "plt" is created that contains all the information about it
 Restore output window.



| Plot Settings | Open Plot Settings File | is used to load previously saved settings.

| Plot Settings | Reload Plot Settings | can be used to restore temporarily changed settings.

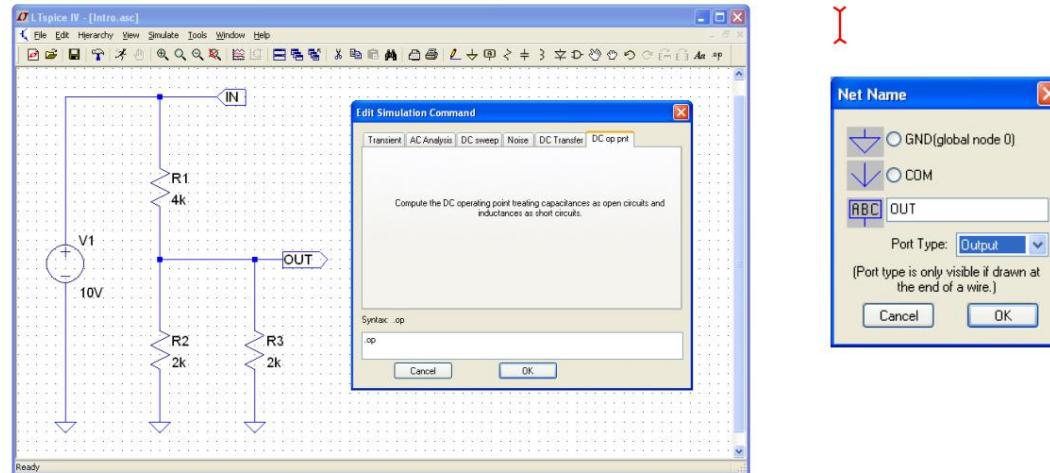
9 beginner examples/types of analysis

9.1 Operating Point

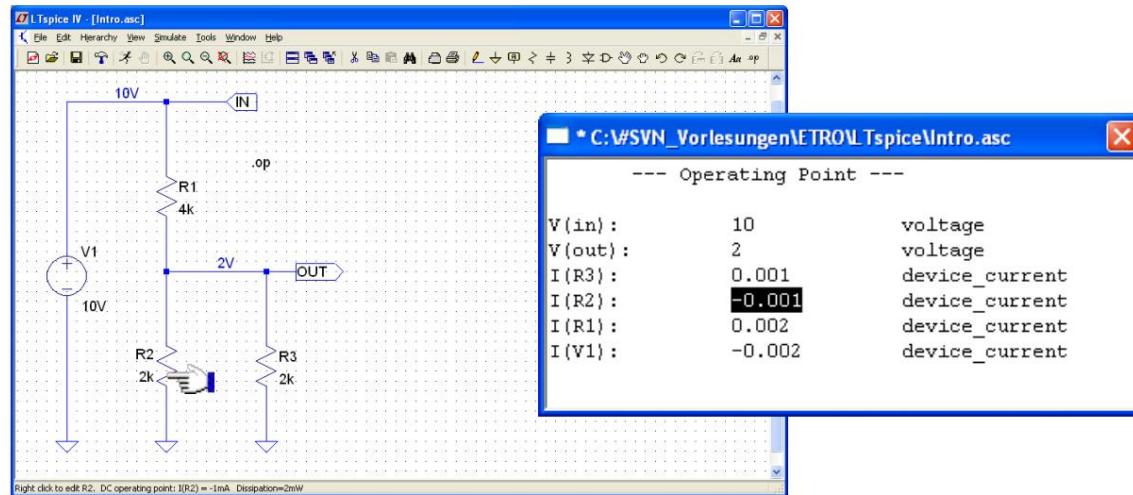
Calculation of the DC operating point for specified DC values of all current and voltage sources in the network. (also carried out for all other types of analysis)

To carry out the analysis, at least one source with a “DC” parameter is used needed.

- Drawing the circuit with [R]esistor, source “**voltage**” [**V**] and [G]round.



- Editing the parameters of V1 **[DC Value [V]]** and Rx (right mouse button)
- Adding network names 
- Creating the SPICE directive | **Simulate | Edit Simulation Cmd |**



The results of the operating point calculation are not displayed graphically, but as text in an additional window. The output lists the voltages of all nodes relative to the reference potential and the currents through all components



The values for the objects under the mouse pointer are also displayed in the status bar.



The direction (sign) of the currents through components is influenced by their installation direction! (compare I(R2) with I(R1))

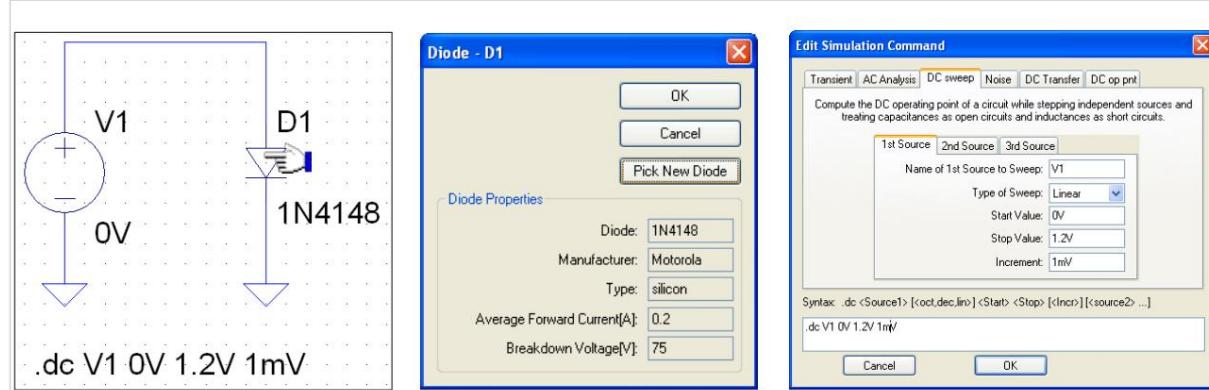
9.2 DC Sweep

Calculation of direct current characteristics as a set of operating points. electricity or Voltage from a source passes through an interval of discrete points. In addition, a global parameter, a model parameter or the temperature can be varied.

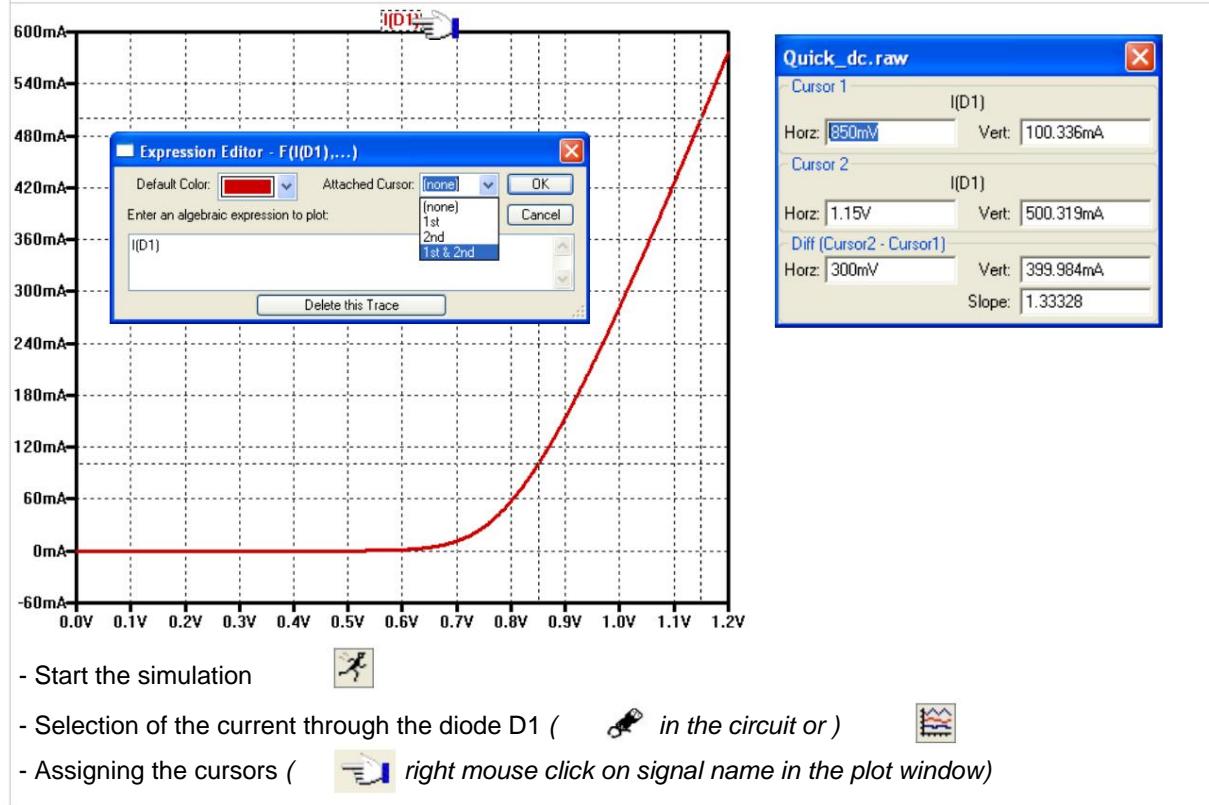
At least one source is used to carry out the analysis with a “DC” parameter required.

The following must be declared: Sweep Source, (U , I , Temp. ,...), Start Value, End Value and Increment.

9.2.1 Example of diode characteristic curve



- Drawing the circuit with [D]iode, source “**voltage**” [**V**] and [**G**]round.
- Editing the parameters of V1 [**DC Value** [**V**]] and D1 [**Pick New Diode**]
- Creating the SPICE directive / **Simulate / Edit Simulation Cmd /**



9.3 Time Domain (Transient) (Large Signal Analysis)

This type of analysis examines circuits in the time domain. The output of the results is similar to that on an oscilloscope screen.

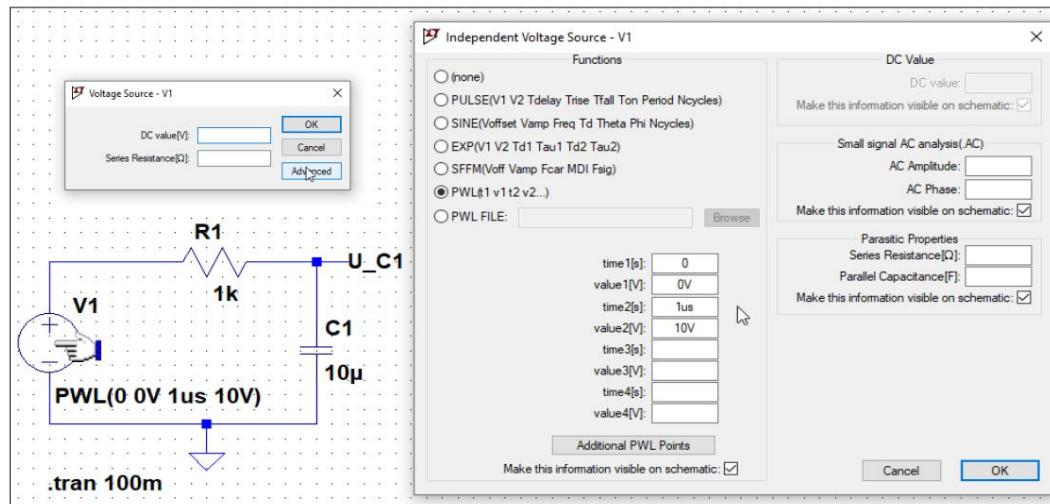
Time is the running variable on the X-axis, voltages or currents are shown on the Y-axis. The time range ("Stop Time") must be declared for the analysis .

To carry out the time domain analysis, sources with parameters are required, which define a signal in the time domain.

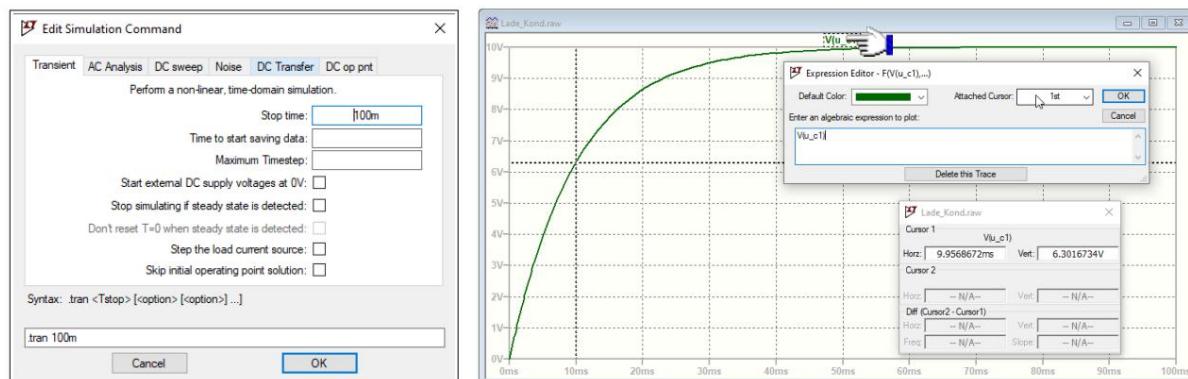
9.3.1 Example of charging time constant for a capacitor

Since the charging voltage must be switched on specifically to determine a charging time constant, the voltage source must be defined in an **advanced** mode.

PWL (*Piece-wise linear*) is suitable for this case . Points are specified here, which are then connected linearly. The voltage value **value1** applies until **time1** . The last voltage value applies from the last specified time. In between, the voltage values are linearly interpolated.



- Drawing the circuit with [R]esistor, [C]apacitor, source "**voltage**" [V] and [G]round.
- Editing the parameters of V1  **[Advanced]**, R1 and C1 (right mouse button)
- Naming the node for the capacitor voltage 



- Creating the SPICE directive **|Simulate| Edit Simulation Cmd|Transient|** (Stop time > 5 μ s)
- Start the simulation  . Selection of the voltage "U_C1"  (or) 
- Activate the cursor and measure the loading time constant

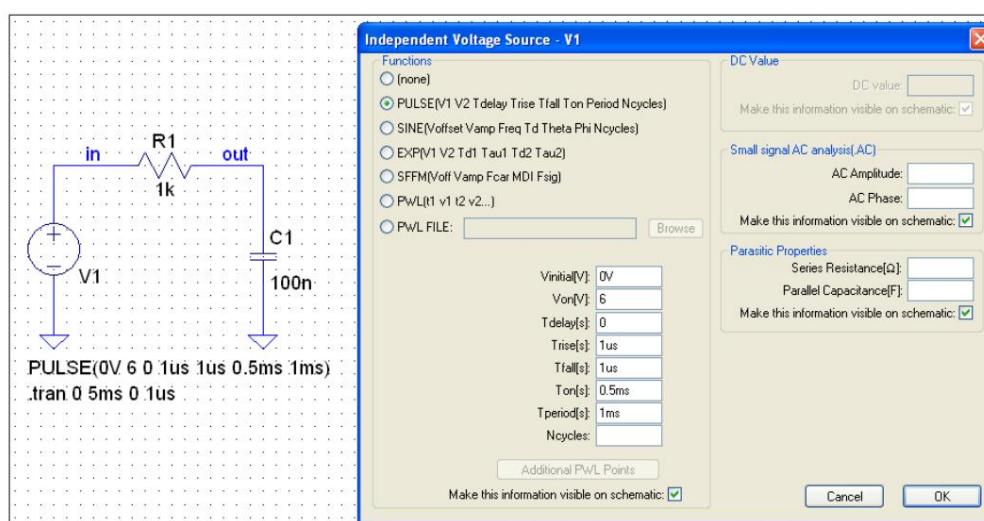
9.3.2 Example RC low pass on square wave voltage (PULSE)

Repetitive/periodic signals are also defined in the Advanced Mode of the sources. Rectangular, triangular and trapezoidal signals can be created with the **PULSE** function create.

Trise and *Tfall* determine the edges of the signal. These times cannot be “0”. If you want to have a square wave signal, then you have to be in proportion to *the tone* and *Tperiod* specify very short values.

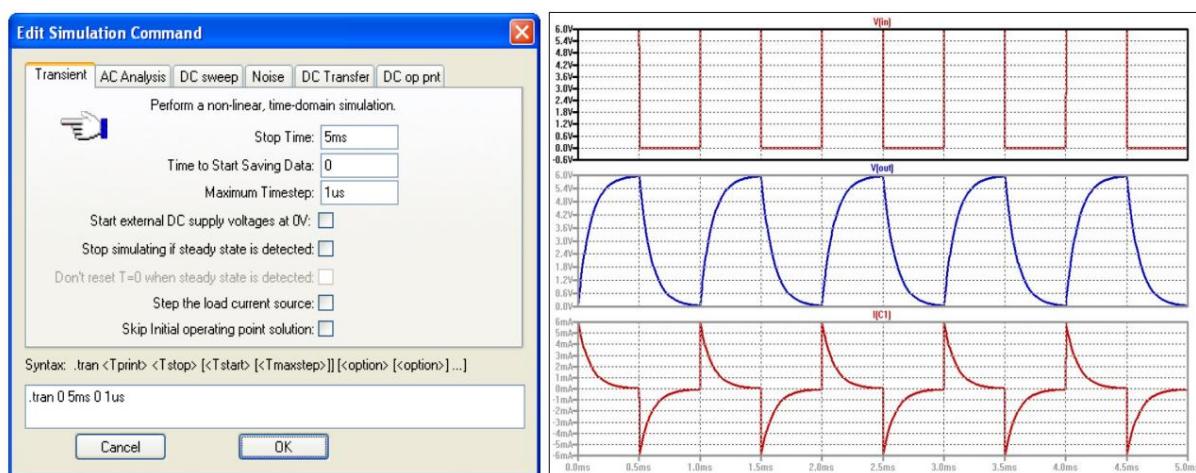
The generated signal then oscillates between the values for *Vinitial* and *Von*. The time for *Voff* ($\neq Vinitial$) results from the period *Tperiod* minus *Trise*, *Ton* and *Tfall*.

The *Tdelay* parameter can be used to set a delay if a signal in the simulation should not be present from time zero. If the signal should end after a certain number of pulses, this is entered into *Ncycles*.



- Drawing the circuit with [R]esistor, [C]apacitor, source “**voltage**” and [G]round.
- Editing the parameters of V1

 **[Advanced]**, R1 and C1 (right mouse button)



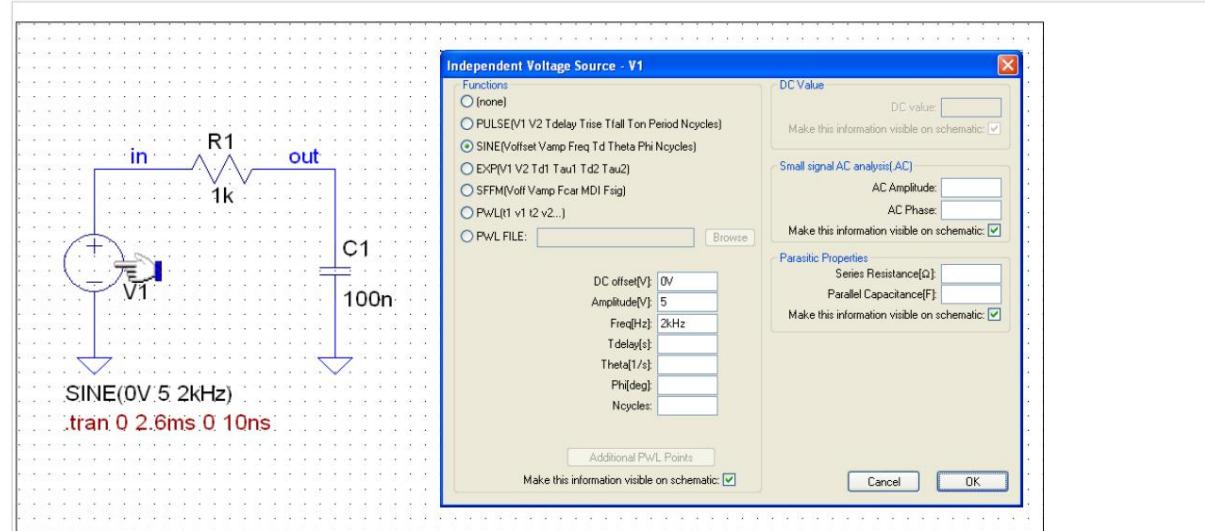
Creating the SPICE directive | **Simulate | Edit Simulation Cmd | Transient |**

- Start the simulation 
- Add two additional plots (**| Plot Settings | Add Plot Pane |**)
- Selection of voltages “in” and “out”  as well as current through  ((or VSK / Ulm University of Technology

9.3.3 Example RC low pass on sine voltage (SINE)

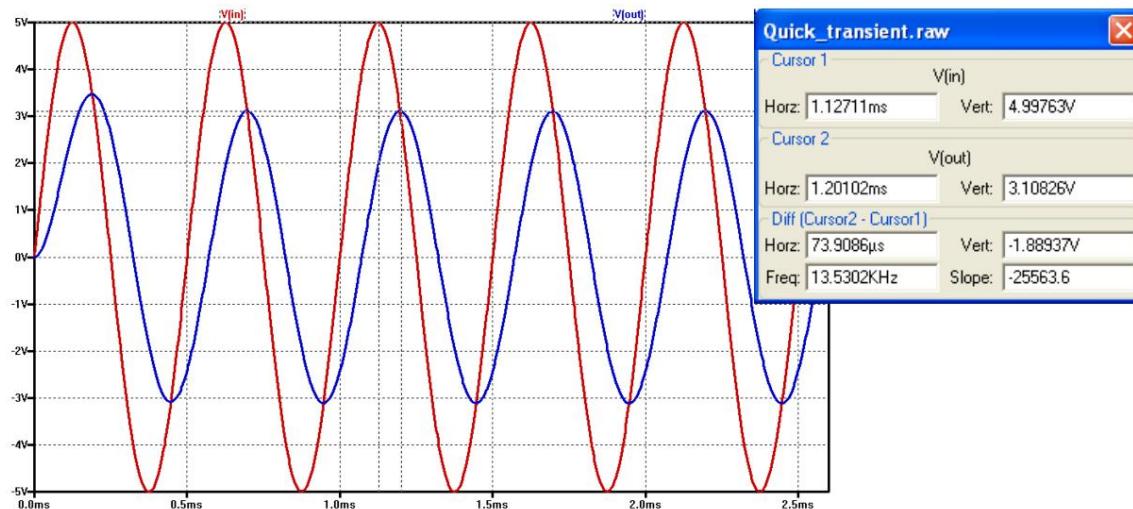
The **SINE** function is available for sinusoidal signals. The most important parameters here are the **frequency**, the **amplitude** and, if necessary, the **offset** of the desired sine signal.

(You can also adjust the **delay**, **start phase angle** and **damping**)



- Drawing the circuit with [R]esistor, [C]apacitor, source “**voltage**” [**V**] and [G]round.
- Editing the parameters of V1

 **[Advanced]**, R1 and C1 (right mouse button)



- Creating the SPICE directive / **Simulate / Edit Simulation Cmd / Transient**
- Start the simulation 
- Selection of voltages “**in**” and “**out**”  (or - Assigning the cursors (right mouse click on signal names in the plot window)
- Setting the cursors

For example, the phase shift can now be determined using the values determined via the cursor positions.

$$\begin{aligned}
 \text{Phase}[\circ] &= 360 * \text{frequency} * \text{Diff-Horiz} \\
 &= 360 * 2\text{kHz} * \sim 74\mu\text{s} \\
 &= \underline{\underline{53,28}}
 \end{aligned}$$

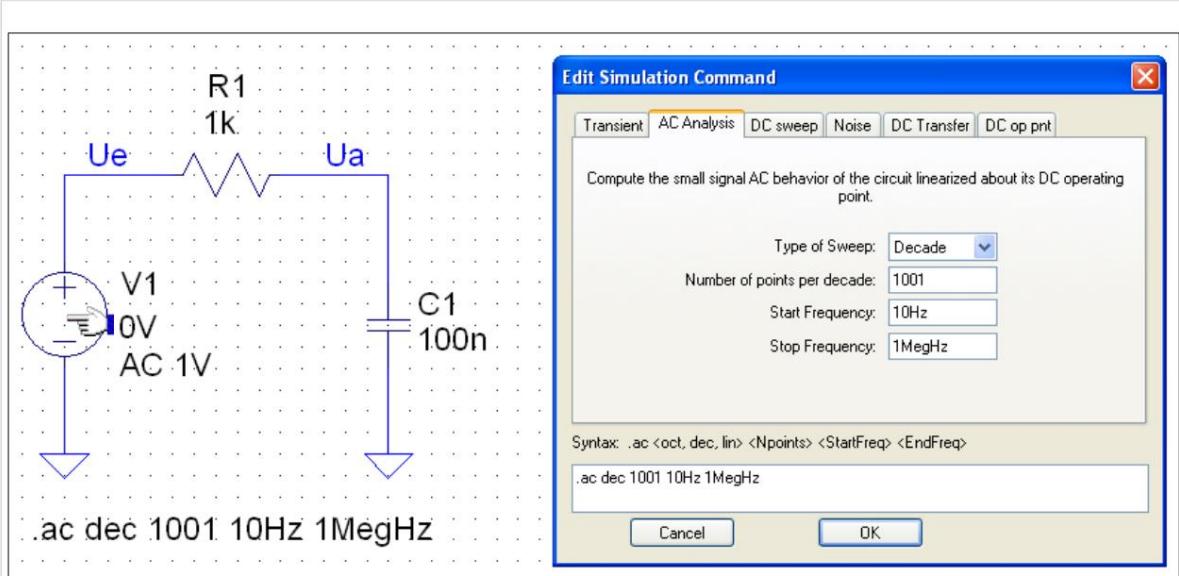
9.4 AC Sweep (Klein-Signal-Analyse)

In this type of analysis, SPICE calculates the response of a circuit to changes in the frequency of the sources. As a result, currents and voltages are displayed according to magnitude and phase, so that Bode diagrams and locus curves can be generated, for example.

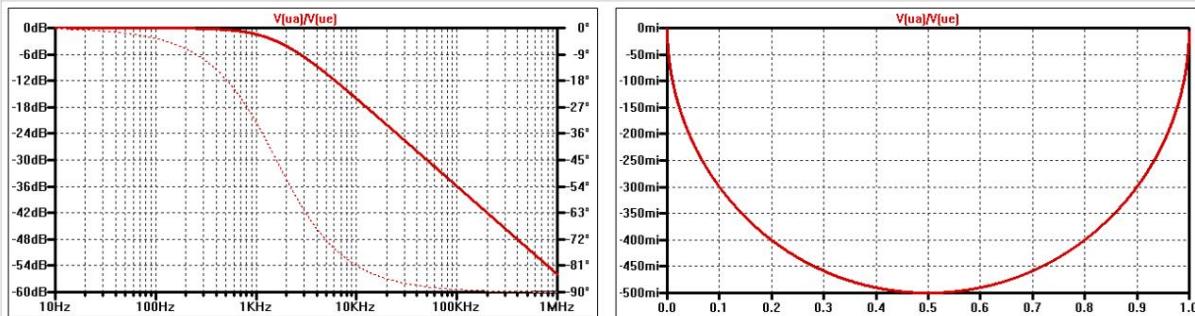
At least one source is used to carry out the analysis with an “AC” parameter required.

The following must be declared: start frequency, end frequency, number of steps (*resolution*)

9.4.1 Example RC low pass



- Drawing the circuit with [R]esistor, [C]apacitor, source “**voltage**” [V] and [G]round.
- Editing the parameters of V1  **[Advanced]**, R1 and C1 (right mouse button)
- Creating the SPICE directive / **Simulate / Edit Simulation Cmd /**

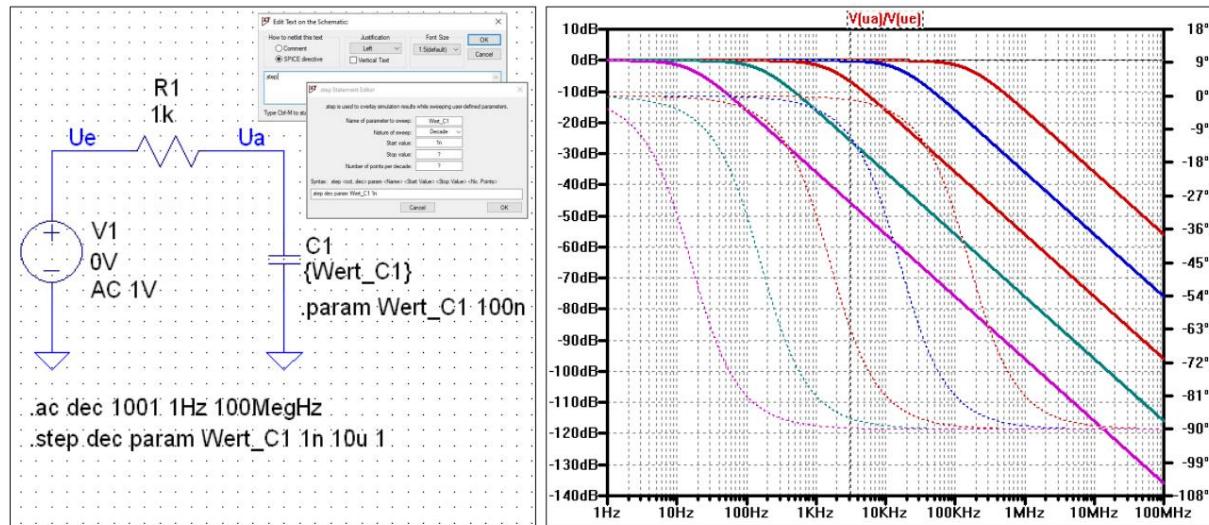


- Start the simulation 
- Enter the desired output function **V(Ua)/V(Ue)** via / **Plot Settings / Add Trace /** oder **[CTRL+A]**
- The function is displayed as a **Bodeplot** by default .
- Switching the amount or phase on or off via the respective Y axes (right mouse button over Y-axis (ruler symbol) → settings menu).
- to the **locus** by switching the representation “**Nyquist**” (via left axis)
- as **real and imaginary components** → representation “**Cartesian**” (via left axis)

9.5 Measurement series - parametric analyses

In parametric analyses, an analysis (*DC, AC, transient*) is carried out multiple times. One parameter (or several) of the circuit is changed. The variation is controlled via the SPICE directive **".step"**. (Optionally also **".param"**)

9.5.1 Example RC low pass – parametric AC analysis



AC analyzes were carried out for the circuit from 1Hz to 100MHz for various values of C1 carried out.

To allow the value to be varied, it must be converted into a function (or a pointer). This is done by changing the value with a new parameter in curly brackets (e.g. **{value_C}**).

The new parameter can now be changed using the **.step** directive. To do this, open the input dialog via **/ Edit / SPICE Directive** in the toolbar or the **[S]** key and enter the text for the desired variation.



The input dialog has integrated help that can be accessed with a right mouse click in the text section.

- As a list: (by specifying the values)
`".step param Wert_C1 list 10n 100n 1u"` ÿ 3 simulations (10nF, 100nF and 1µF)
- Linear: (Startwert, Endwert, Increment)
`".step param Value_C1 100n 1u 100n"` ÿ 10 simulations (100n, 200n, 300n ...)
- Logarithmic: (start value, stop value, values per decade or octave)
 - `".step dec param value_C1 10n 10u 1"` ÿ 4 runs (10nF, 100nF, 1uF, 10uF)



Danger: If the Step directive is not executed, the value_C1 is unknown
Remedy: `.param Value_C1 100n`



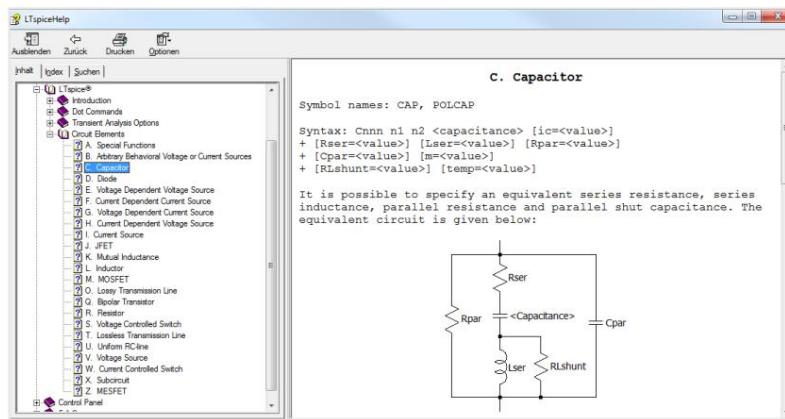
In the menu **/ Plot Settings / Select Steps** / The graphs to be displayed can be selected.
 In standard mode, all are displayed.



If cursor measurements are carried out, you can move on to the next step using the **[ÿ] [ÿ]** keys. Information about which step the cursor is on can be obtained by clicking with the right mouse button.

10 Model / Symbol / Netlist / Subcircuit / Library

The LTspice help also contains documentation of all framework elements. Even the supposedly simplest elements such as resistors (R), inductors (L) and capacitors (C) consist not only of a value for the respective parameter, but of a model that reflects the real properties of the components.



More complex components such as operational amplifiers are formed from several basic elements using netlists (*subcircuits*) or circuit diagrams (*hierarchical blocks*).

Components can also be grouped together in libraries. To be used in the circuit diagram, symbols are required that are linked to the respective descriptions in the model, subcircuit or hierarchical block.

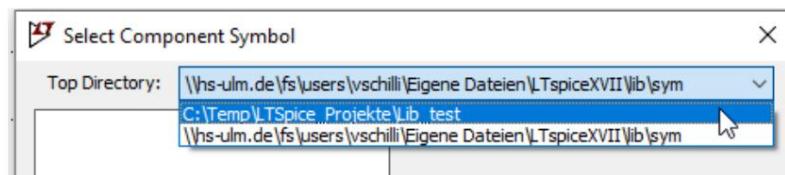
10.1 Locations of component models and libraries

The libraries of the LTspice installation (*.../LTC/LTspiceXVII/lib*) are installed on the first start of the software is copied by a new user into his user directory and only this Copies are then used when creating new designs.

Under Windows these copies are located in the path, for example:

.../users/<username>/My Documents/LtspiceXVII/lib.

Circuit elements can only be added to a design using **symbols** from these copies and from the directory of the current design !



Subsequent changes to the original libraries in the installation directory by users are apparently ineffective!

In addition, changes and additions to libraries in the user directories due to program updates can also be lost if the corresponding files are overwritten by an update.

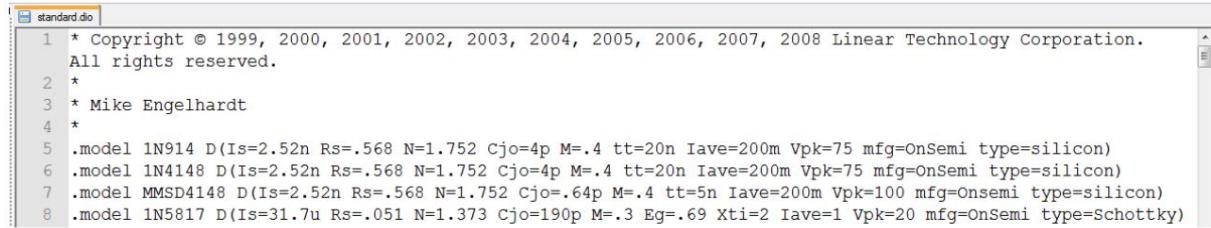
Suggestions for storing new components can be found in the chapter on creating new or own components.

10.2 Models of the Spice basic elements

The basic elements such as diodes, RLC, transistors provided by LTspice are contained in library files in the folder `../LtpiceXX/lib/cmp` in the `standard.xxx` files. Depending on the component, the structure of the library files varies and is described as an example in the following chapters.

10.2.1 Discrete semiconductors

The semiconductor component libraries contain the models of the components, which can be selected via the symbol dialog in the circuit diagram. The libraries are text files that can also be edited or added to.



```

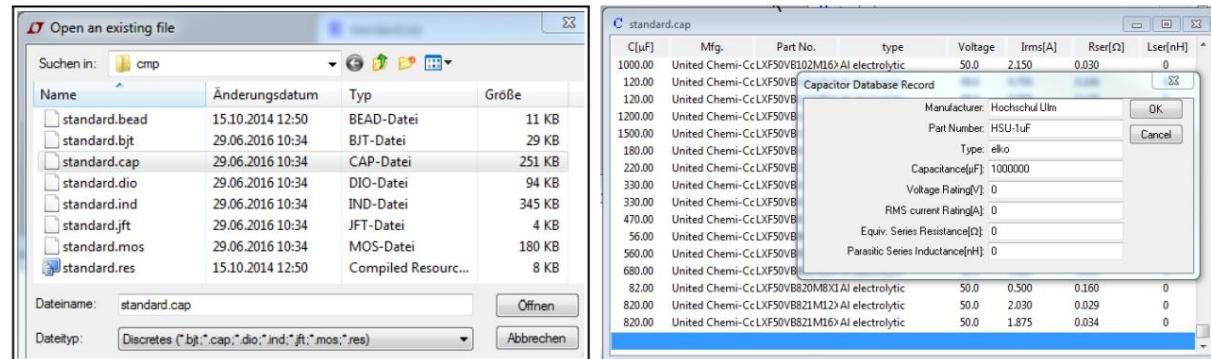
standard.dio
1 * Copyright © 1999, 2000, 2001, 2002, 2003, 2004, 2005, 2006, 2007, 2008 Linear Technology Corporation.
2 *
3 * Mike Engelhardt
4 *
5 .model IN914 D(Is=2.52n Rs=.568 N=1.752 Cjo=4p M=.4 tt=20n Iave=200m Vpk=75 mfg=OnSemi type=silicon)
6 .model IN4148 D(Is=2.52n Rs=.568 N=1.752 Cjo=4p M=.4 tt=20n Iave=200m Vpk=75 mfg=onsemi type=silicon)
7 .model MMSD4148 D(Is=2.52n Rs=.568 N=1.752 Cjo=.64p M=.4 tt=5n Iave=200m Vpk=100 mfg=onsemi type=silicon)
8 .model IN5817 D(Is=31.7u Rs=.051 N=1.373 Cjo=190p M=.3 Eg=.69 Xti=2 Iave=1 Vpk=20 mfg=OnSemi type=Schottky)

```

Diodes, MOS-FETs, J-FETs and bipolar transistors are each organized in their own files, which can be distinguished by their file extensions.

10.2.2 RLC – Resistors, Inductors and Capacitors

The passive components are also in their own files for resistors, capacitors, Inductors and ferrites organized.



The libraries for the RLC components are structured somewhat differently than those for the semiconductors and appear incomprehensible in a text editor. If you open these files in LTspice, you get a database view and can delete, modify, add entries using the edit menu...

10.2.3 Sources

There are no library files for the various current and voltage sources.

The description is created individually for each source. (in the circuit diagram)

An overview of the types of sources available can be found in the help.

10.3 Circuit diagram symbols

In order to add a component to a circuit diagram, you need a symbol is linked to a model, subcircuit or hierarchical block.

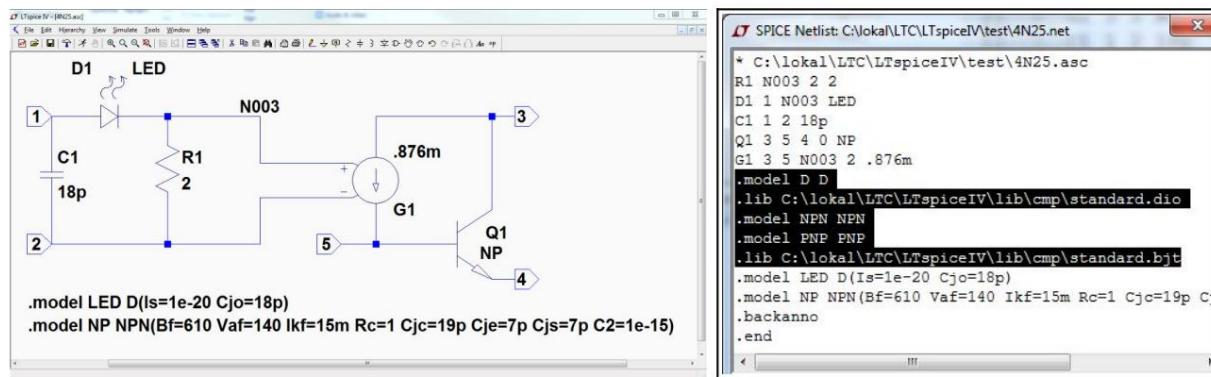
The standard symbols are located in the .../**Own files/LtspiceXVII/lib/sym directory**, which, in addition to the current design directory, is also available as the base directory in the dialog for adding a new component.

10.4 Netlists

For the simulation with Spice, netlists are required that describe the connections between the elements (*models, subcircuits...*) of the circuit diagram.

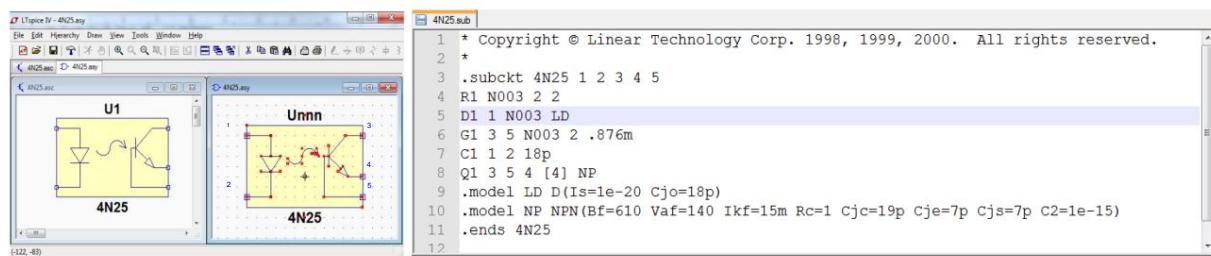
LTspice automatically generates the netlists from the schematic before simulation. These netlists can be accessed for the active schematic window using the command | **View / SPICE Netlist / display**.

The following figures show the replica of a 4N25 optocoupler as a circuit diagram and the netlist generated from it. (*The blacked out part of the netlist is actually not relevant*)



10.5 Subcircuits

Descriptions of components in the form of subcircuits are nothing more than netlists. The 4N25 optocoupler is available in LTspice and stored as a symbol and as a subcircuit in the file .../**LtspiceXX/lib/sub/4N25.sub**.



If you compare the netlist of the schematic above and the description of the subcircuit below, you will see that a subcircuit is just a netlist with a title (.subckt 4N25 1 2 3 4 5) . (.end) .ends)

Files that only contain a description of a single component should actually have the ending *.sub, but you can also often find the ending *.lib in files that originally come with LTspice .

10.6 Libraries / Libraries

As already seen above, the libraries contain collections of models or subcircuits.

Libraries that have been described as subcircuits have the ending .lib and are located by default in the .../**My Documents/LtspiceXVII/lib/sub directory** and, if necessary, in other subdirectories.

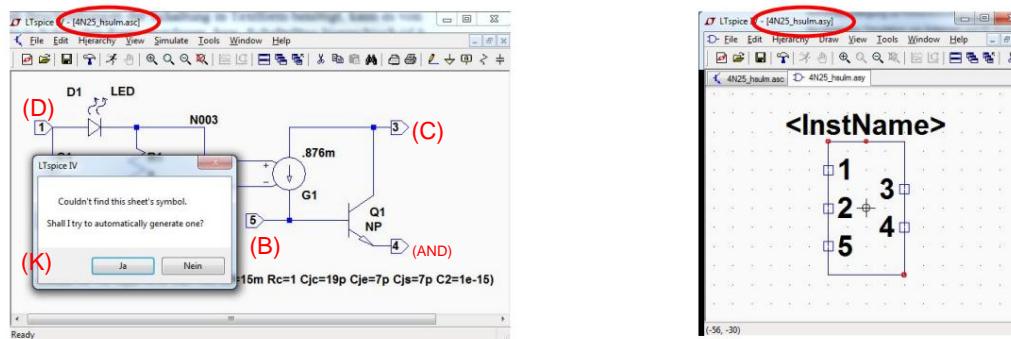
11 hierarchical blocks / designs with multiple pages

If you want a design with multiple schematics hierarchically (*i.e. with sub-schematics*) then you have to create a symbol for each page of the sublevels, which you then integrate into the higher-level schematic page. **All pages and associated icons are stored in the same directory for hierarchical designs!**

11.1 Create symbol

In the menu | **Hierarchy / Open this Sheet's Symbol** | You can have a symbol automatically generated for an open circuit diagram page. If none is available yet, it will come automatically a corresponding request. (*Example from optocoupler 4N25 above*)

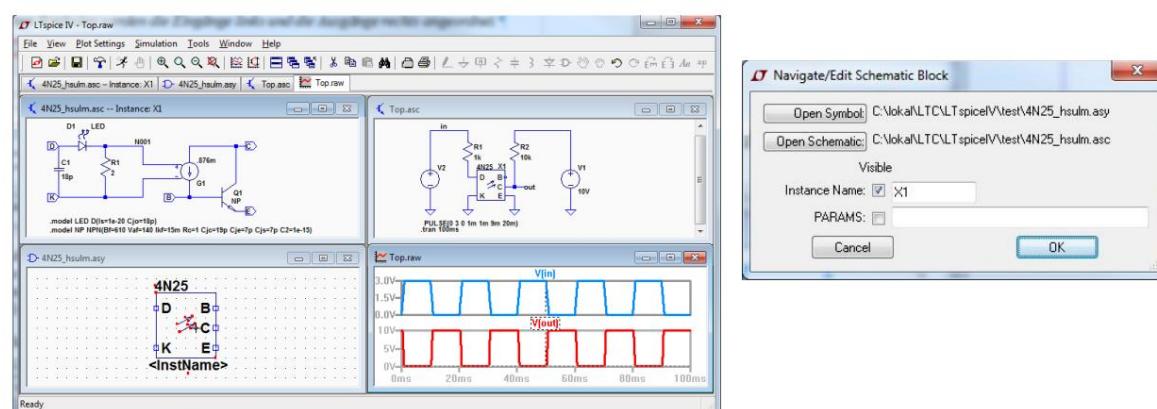
During automatic generation, the signal directions specified in the circuit diagram are taken into account. *Entrances were arranged on the left and exits on the right.*



Of course, the symbol generated is not nearly as meaningful as the 4N25 symbol in the library. The names of the connections are also not self-explanatory. That's why it's better to delete such a symbol again, rename the nets (*to D, K, B, C and E*) and then create it again. (*probably quicker than changing both*)

Please embellish a symbol created in this way at least a little in the editor and save it.

If you then insert it as a component in a new circuit diagram, you can right-click to navigate to the underlying circuit diagram and also to the symbol.



12 Integrate or create new/own components

Many components are not available when installing LTspice, but they can usually be easily installed be added additionally. This primarily affects components from manufacturers other than Linear Technology and Analog Devices. Of course, not all of the countless discrete components such as diodes and transistors, etc. are available in the libraries.

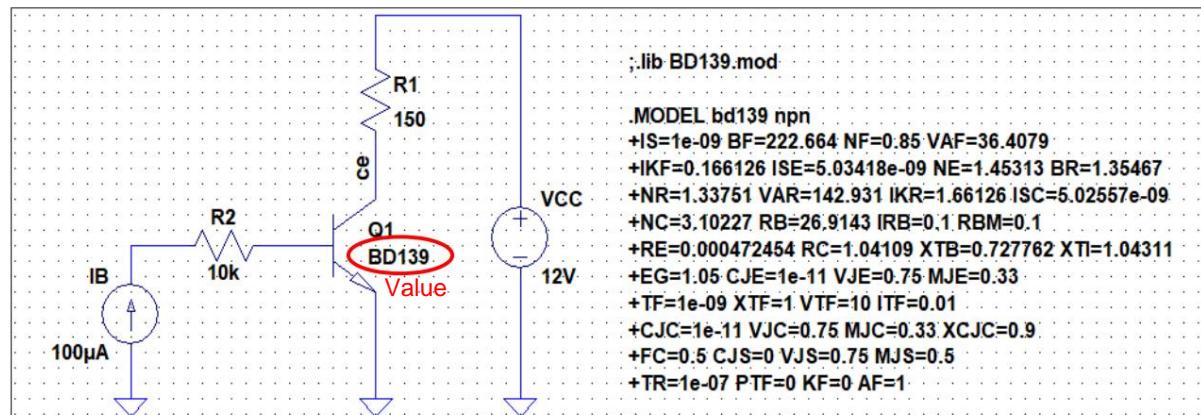
12.1 Add Spice Model

Discrete electronic components such as diodes and transistors are characterized using predefined models, whose parameters are then adapted to the specific component.

You can look up these models and parameters in the LTspice help and will probably be overwhelmed by the number of available parameters. Normally you simply get the model online and incorporate it into the design.

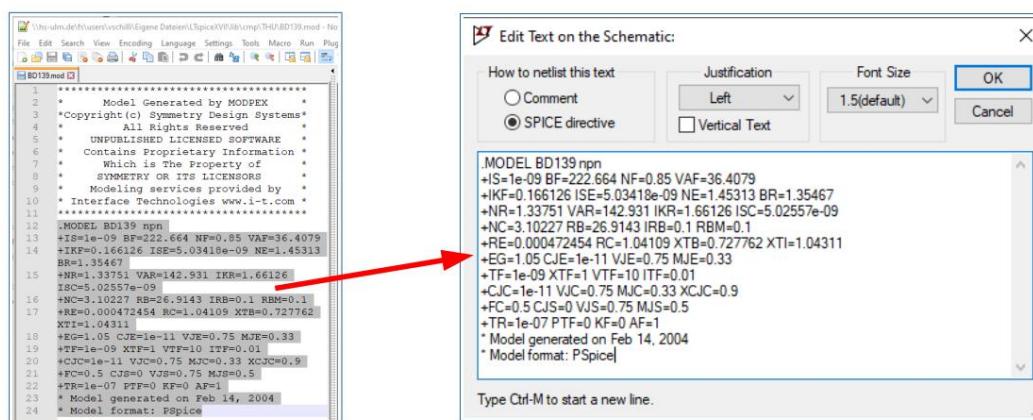
Attention: Even with discrete components you often get a Spice model as a **subcircuit**. In this case, integration is no longer that easy, since diodes and transistors in LTspice initially only expect one **model**. **The type of the LTspice component must match the type in the symbol's attributes or be adjusted in these!**

A new model is assigned to a standard LTspice element. by inserting the appropriate symbol from the library and specifying the name of the new model as "**Value**". (*NPN transistor Q1 of type BD139 in the figure*)



12.1.1 .MODEL Directive

A model description can be inserted directly into the plan as text using the **.MODEL** directive. This has the great advantage that it does not contain any dependencies on other files that have to be taken into account when passing on the design.



12.1.2 .lib/.inc Directives

The textual description of a Spice model contained in a file can also be integrated into the design using the .lib or .inc command. In order for LTspice to find the file with the model description, it must be in the **design directory** or specified with the **absolute path**.

You can insert the .lib or .inc directive via the edit menu of the circuit diagram or the corresponding button in the toolbar. When you first call up the SPICE Directive dialog, the window in the following figure appears on the left.



Of course, the file location can also be a **local location**

act where all models that were not included in LTspice are collected. The path to get there can be arbitrarily complex, which can make entering it a bit tedious.

When editing a .LIB or .INC command that already exists in the circuit diagram, appears the window as shown in the figure on the right. Now you have the advantage that you can easily select a storage location for the model file using the **browse button** if it is not in the design directory.

The absolute path can also be an **Internet address** of a file where the model was found. In this case, the file will be copied into the directory the first time the simulation is called and the path information could then be deleted again. (e.g. <https://www.onsemi.com/download/models/lib/bd139.lib>)

If the additional model files are located within the copy of the library structure created by **LTspice**, the path information can be abbreviated and the standard part of the path can be omitted. (in the figure on the right \.lib **THU\BD139.mod** is enough)

Further details about .lib/.inc and paths can also be found in the LTspice help.

12.1.3 Insert model into standard library

Of course, you can also insert the model of a "new" transistor, e.g. BD139, into the library for bipolar transistors from LTspice.

To do this, open the library file (.../My Documents/LtspiceXVII/lib/cmp/standard.bjt) directly in LTspice and insert a new line into text files. You can use the existing ones as a guide.

If a component editor opens, use the "Edit" menu, which also opens with a right mouse click.

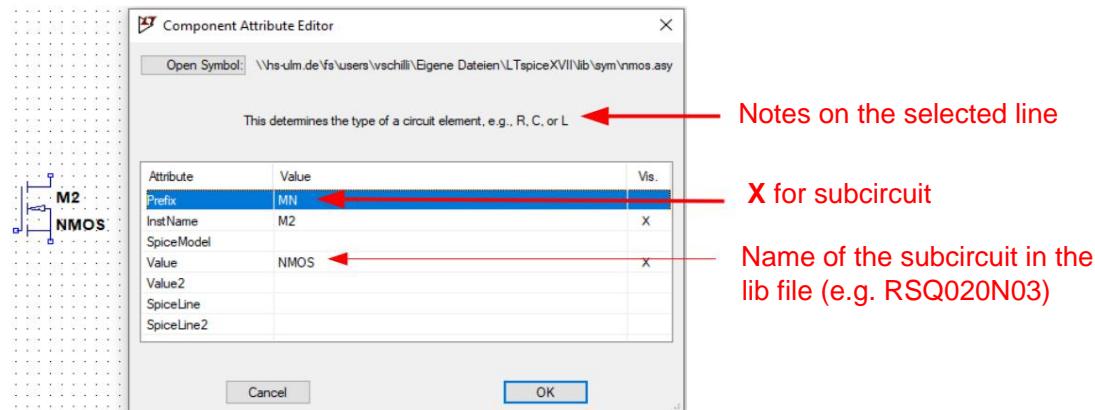
(General: Beware of loss of manufacturer updates)

12.2 Add subcircuit

A component description, which is available as a subcircuit, can be integrated in a similar way to a model description, or you can organize it like the corresponding components in the libraries using your own symbol with a suitable name.

For a standard symbol from the library, which is normally used for Spice Primitives (*diodes, transistors, R, L, C ...*) with model definition, you have to insert one into the schematic and then make some changes in the **Component Attribute Editor**.

The **Component Attribute Editor** is accessed by right-clicking on the symbol while holding down the control key [CTRL].



The most important attribute is called **Prefix** and determines which Spice Primitive is represented by the symbol. “MN” in the figure above stands for an N-Channel MOSFET model. If you do not want to use a primitive, but rather a subcircuit, an “X” should be entered in the prefix.

The name of the subcircuit is then entered under Value as it appears in the lib file.

12.2.1 .SUBCKT directive

Just like a model description when inserting a mode, you can also insert the description of a subcircuit from a lib file directly into the plan as text using the .SUBCKT command, or simply adopt the description completely into the design as a spice directive .

12.2.2 .LIB /.INC Directives

The description of a subcircuit can of course be quite extensive compared to a model and it is more likely to be described using .lib or .inc

Want to integrate commands into the design. As with the model, the file with the description must be in the design directory or specified with the absolute path.

If the additional subcircuit library is again within the copy of the library structure created by **LTspice in the .../lib/cmp directory**, the path information can also be abbreviated and the standard part of the path (.../lib/cmp/) can be omitted.

(Differences .lib/.inc see LTspice help)

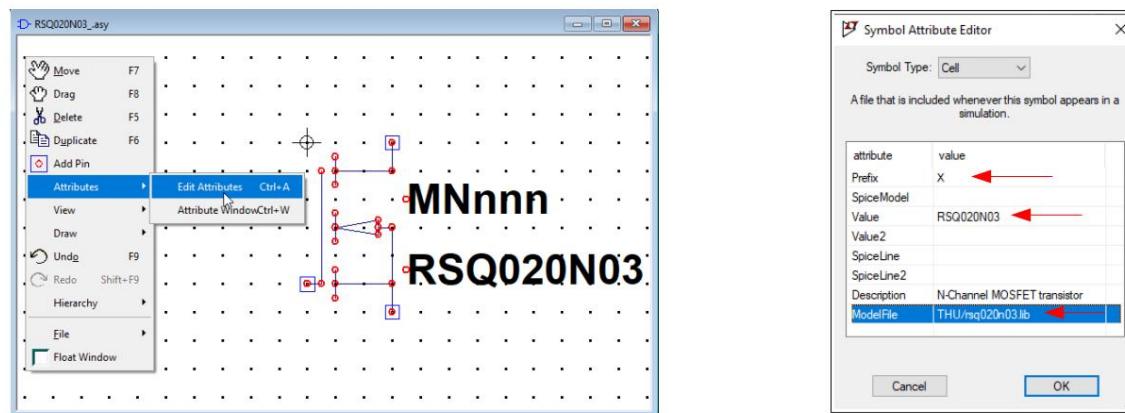
12.2.3 Link a new subcircuit with its own symbol

For components that you would probably like to use more often in a simulation in the future, it is definitely worth creating a complete component by linking your own symbol to the subcircuit.

12.2.3.1 Existing symbol

There is usually a suitable symbol in the standard library, which you can open in the **Component Attribute Editor** (see figure above) using the **Open Symbol** button and save under a new name in a directory for your own symbols. This directory could be, for example, `.../lib/sym/THU`.

The link between the new symbol and the subcircuit library is then created in the **Symbol Attribute Editor**.

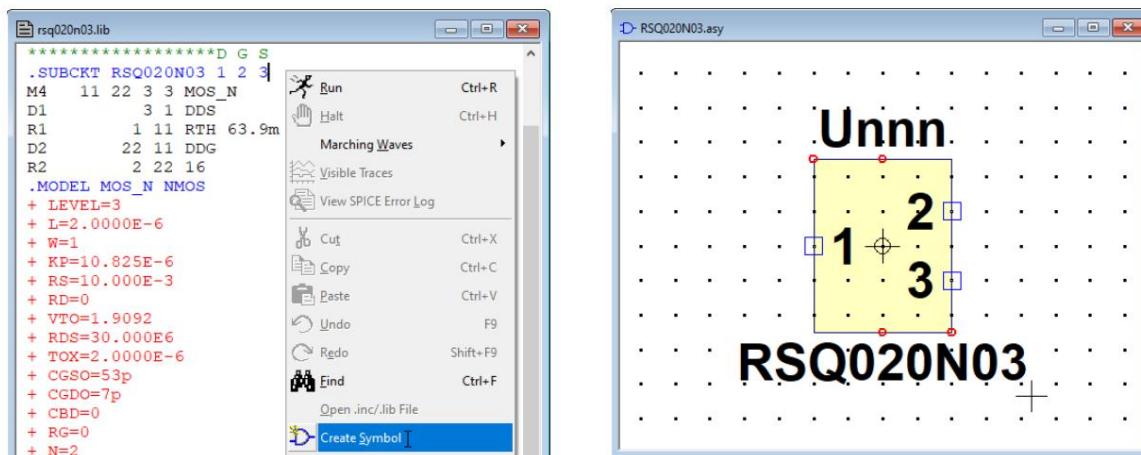


Here you may have to change **the prefix** to **X** again , enter the name of the subcircuit under **Value** and then enter the corresponding file (*with path*) under **ModelFile** .

Adjusting the prefix and value as well as specifying the subcircuit only has to be done once. All of this is already there when the new symbol is inserted into a design and does not have to be adjusted every time.

12.2.3.2 Generate symbol from subcircuit

A very simple symbol for a subcircuit description can be created by opening the library file in LTspice and right-clicking on the corresponding line. A symbol created in this way is saved in the `.../lib/sym/Autogenerated` directory. Of course, you can still process it sensibly and move it to an area of your choice.



12.3 Add components to the library as hierarchical blocks

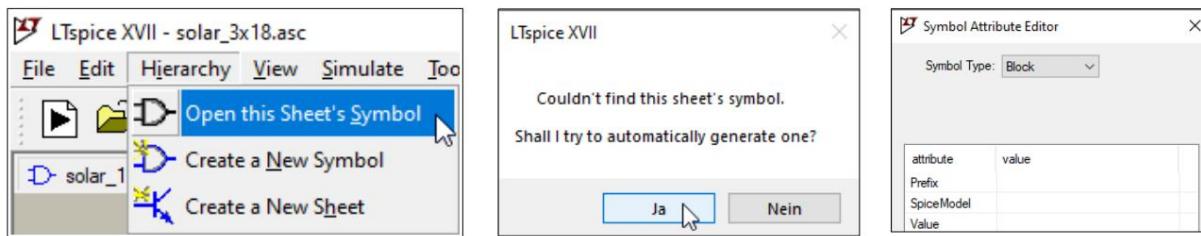
If you want to create and save components in circuit diagram form instead of as subcircuits, then this happens exactly as with the design of hierarchical circuit diagrams above. You have to create a symbol for these components, which are available as a circuit diagram.

If the directory for hierarchical circuit diagrams is usually the directory of the design, it is better to save components as hierarchical blocks in a folder in the symbol path of the library (...lib/sym/THU).

Attention: The circuit diagram file should not have a name that already has a name somewhere. There is a standard symbol of the same name!

12.3.1 Generate icon

When a symbol for a circuit diagram is automatically generated, as described in Chapter 11, the symbol is also given the appropriate name. (*Symbol name must be the same as that of the schematic*). The symbol file is saved in the same directory.



In the Symbol Attribute Editor you can see that the **Symbol Type** has been set to **Block** and all attributes remain empty.

Selecting the **Symbol Types Block** causes LTspice to search for a suitable plan in the symbol's directory. The connection between symbol and plan is created simply by having the same name.

12.3.2 Add existing design blocks to the library

A hierarchical block, which represents a component, is fundamentally no different from a page in a hierarchical circuit diagram. Only the storage location is different.

That's why you can simply add circuit diagram pages that were originally only partial plans of a design to the library as prefabricated blocks by copying the two files for the plan and symbol into the library directory. (...lib/sym/THU)

By adding it to the library, a block becomes available to all designs and does not need to be manually copied to the design directories.

13 example circuits

13.1 Examples in the .../LTspice/examples/Applications folder

By installing LTspice you also receive an extensive collection of example circuits of components from the manufacturer Analog Devices and the former manufacturer Linear Technologies, which was taken over by Analog Devices.

For installations from version 17.1 onwards, the examples and libraries are only included in the program's installation folder as a zip file. The individual files can usually be found in the user directory `<userdir>/AppData/Local/LTspice/examples/Applications`.

In addition, there are examples for the various **UniversalOpAmp** models and various filter circuits in the **examples/Applications** folder.

13.2 Examples in the .../LTspice/examples/Educational folder

Some of the examples in the **educational** folder are circuits that are used in the integrated help. There are also a few others, but unfortunately without more detailed documentation.

13.3 Switch

13.3.1 Simple switch SPST (switch)

The suitable circuit element for switches in LTspice is the **Voltage Controlled Switch**.

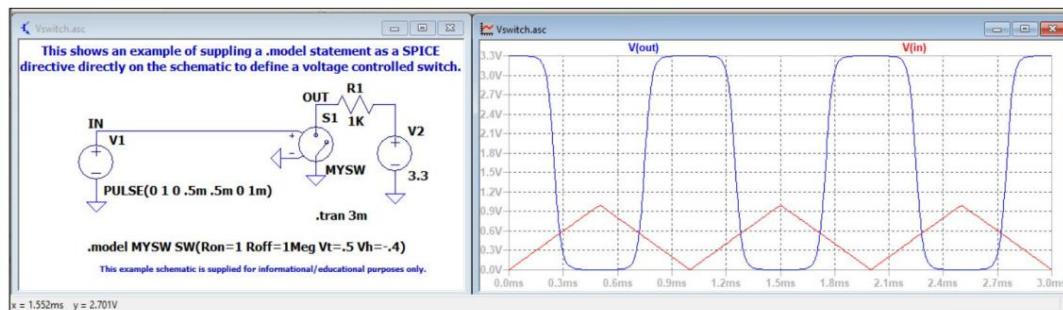
There is also a **current controlled switch**.

Like all elements in the help, these can be found via search or under

Help->Ltpice->Ltpice->Cuicuit Elements. In the help you will also find a reference to a circuit in the .../examples/**Educational** folder which illustrates some attributes.

As the name suggests, for the **Voltage Controlled Switch** you need the switch itself, which consists of the **symbol SW** and a description in the form of a

.model <...> SW(...) directive and also a **voltage source** around the switch steer.



In the example, the PULSE function is used for the source to generate a triangle-shaped control voltage. The switch was assigned “**MYSW**” as a value and the **.model** directive was used to specify that it has a resistance of $1\bar{y}$ when closed and $1M\bar{y}$ when open.

The **switching threshold (Vt)** is **+0.5V** and the **hysteresis (Vh)** was specified as **0.4V**.

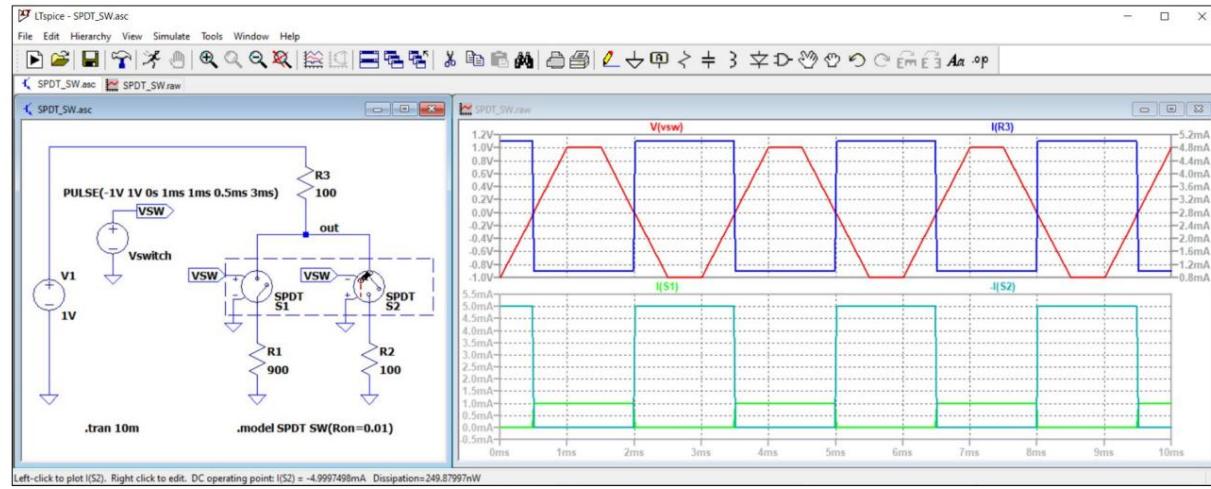
This means that the switching process takes place between 0.1V and 0.9V and the rounded square wave signal **V(out)** is obtained at the **OUT** node.

Attention: The sign of the hysteresis also influences the switching process .

13.3.2 SPDT changeover switch

LTspice only offers the simple switching elements. If you need a changeover switch or another more complex switching element in your circuit, you have to assemble it from several simple switches and control it accordingly. For a changeover switch you need two switches, which you have to control synchronously,

that one is open and the other is closed.



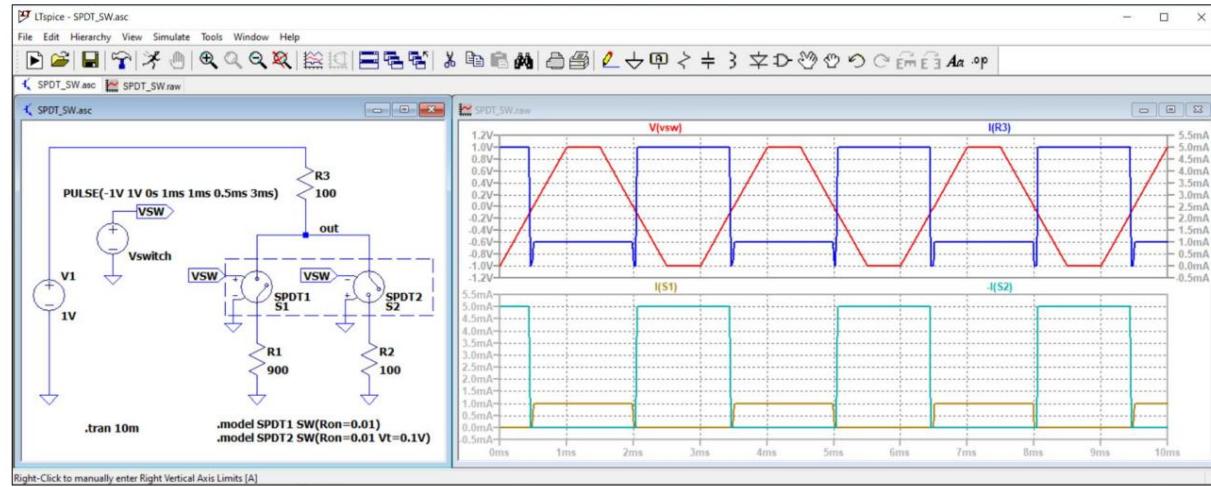
In the example, the two switches S1 and S2 were given the same SPDT value .model description linked, but controlled alternately by simply applying the control voltage at S2 inverted.

The preset voltage at which the switches switch is 0V and they are closed when there is a more positive voltage at the positive control input (+) than at the negative (-).

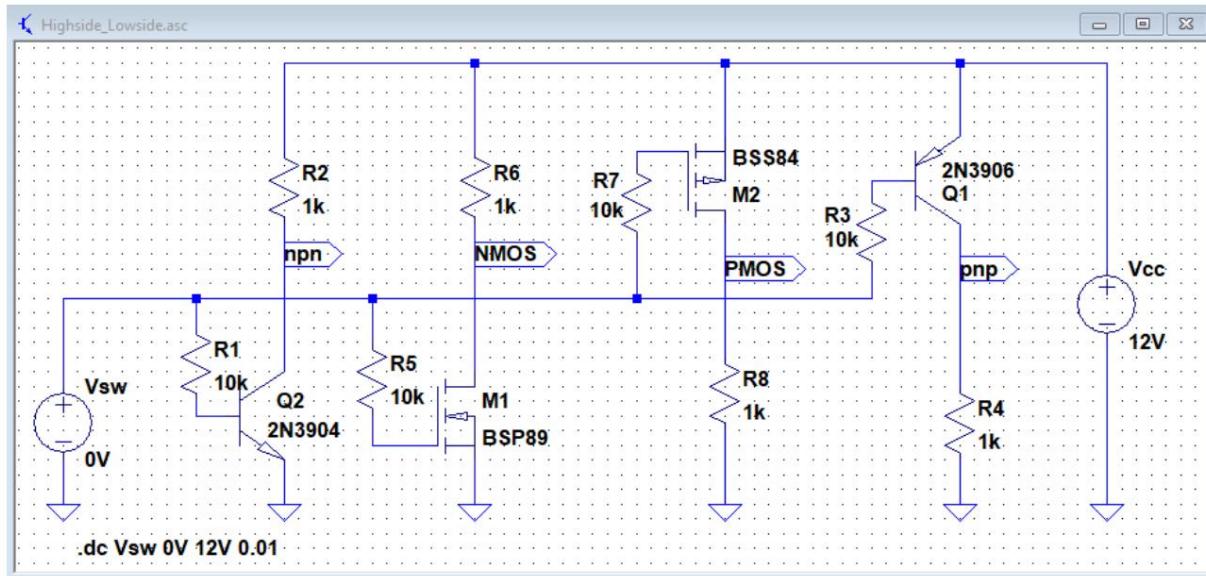
In the circuit diagram on the left, the arrow in the clamp meter symbol shows why the current through S2 is shown inverted (as $-I(S2)$) in the diagram on the right.

13.3.3 Real behavior of a changeover switch

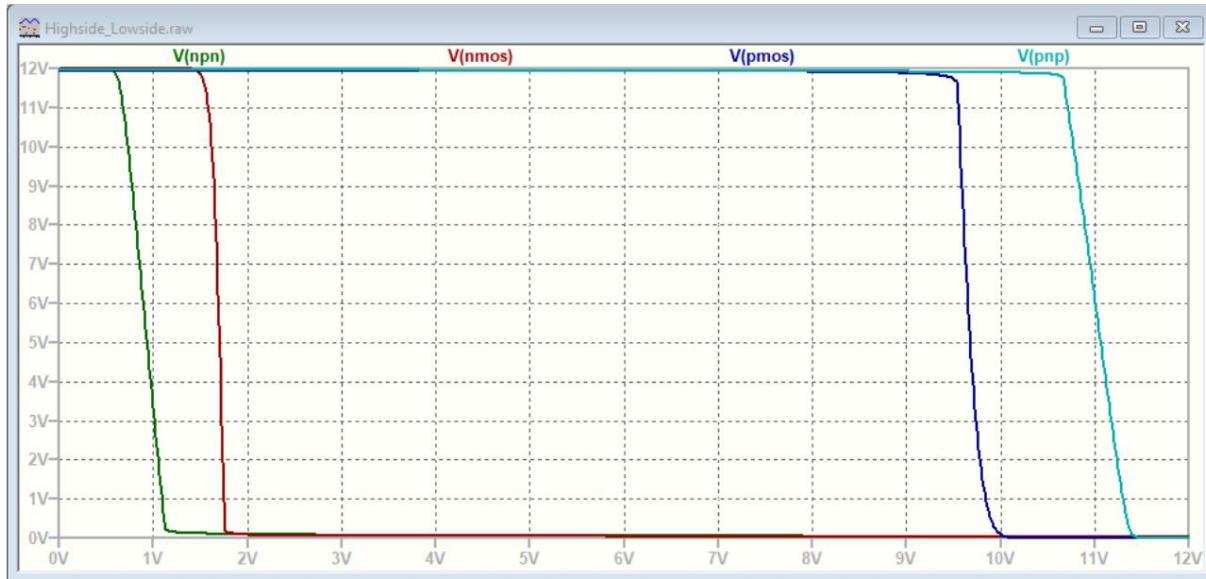
With a real switch, when switching, a state will occur in which no contact is closed. In the example above, this can be done using different models for S1 and S2 to reach. You have to raise the switching threshold for S2 slightly (e.g. $Vt=0.1V$).



13.4 Transistors as switches / High side – Low Side



Without...

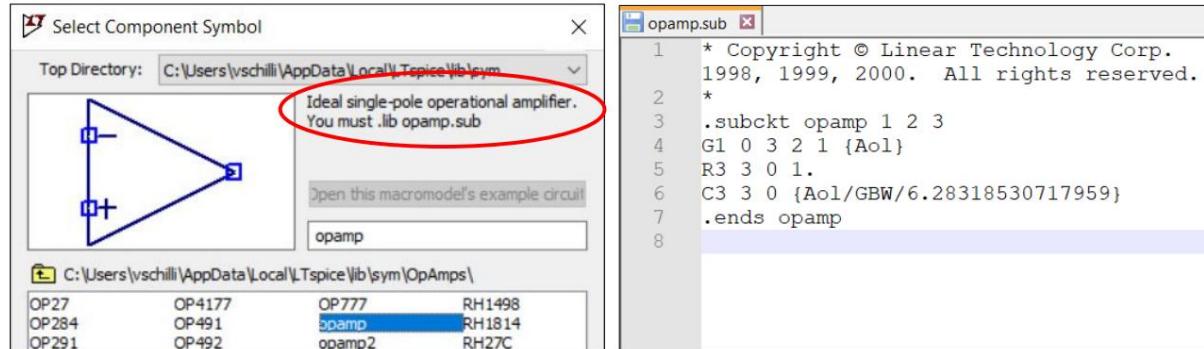


Without...

13.5 Operational Amplifiers

If you don't want to use a special model of a real operational amplifier for a simulation, LTspice offers various universal models, but there are a few things to keep in mind.

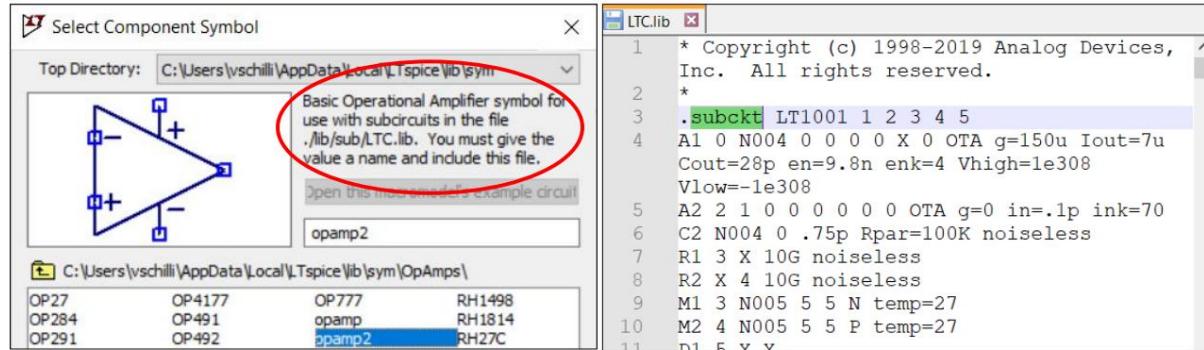
13.5.1 The “opamp” symbol



As stated in the information text next to it, the symbol **opamp** is an ideal amplifier. The circuit diagram **must** contain the Spicedirective “**.lib opamp.sub**” because no SpiceModel was assigned to the symbol in advance!

In the file linked by the **.lib** directive, a very simple subcircuit is then made up of a **Voltage Dependent Current Source (G)**, a **resistor (R)** and a **Capacitor (C)** defined.

13.5.2 The “opamp2” symbol



In contrast to the **opamp** symbol, the **opamp2** symbol is one of the almost 200 models of real operational amplifiers contained in the library file “**LTC.lib**”.

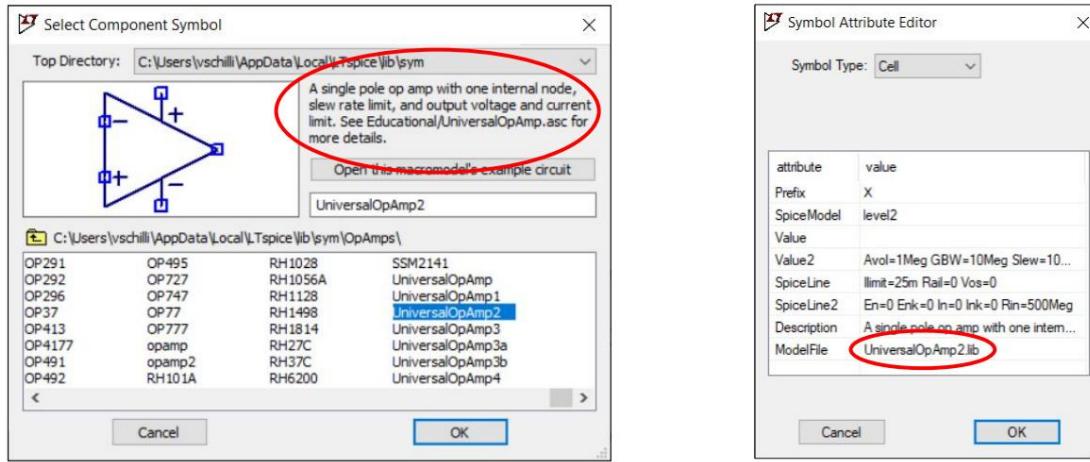
As stated in the description, you must specify one of the existing subcircuits (e.g. *LT1001*) in the **Value** (**opamp2**) parameter and integrate the library with the directive “**.inc LTC.lib**” or “**.lib LTC.lib**” .

13.5.2.1 Alternatives to opamp2 / LTC.lib

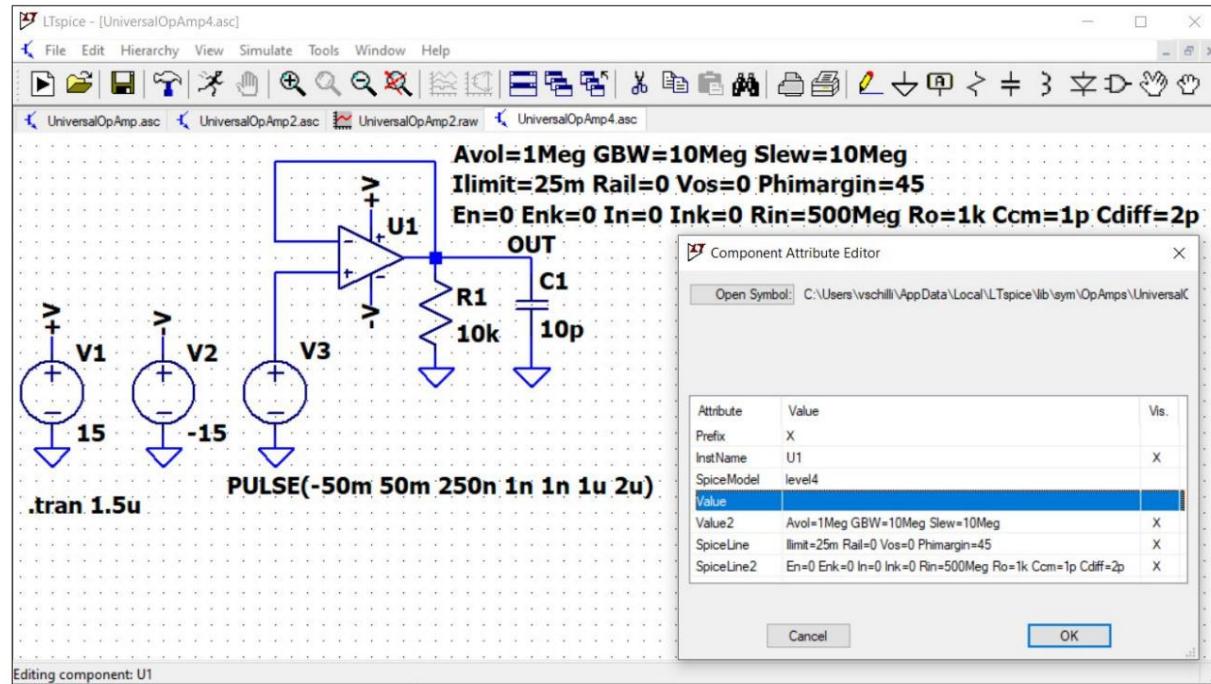
The operational amplifiers contained in the **LTC.lib** library actually have their own symbols, which have already entered the corresponding names in the **Value** parameter and also contain the link to the library.

Using opamp2 is not really recommended

13.5.3 The different UniversalOpAmp



The various **UniversalOpAmp** symbols all look exactly the same, but are linked to different model files in which the levels of complexity specified in the description are implemented. A simple example circuit is available for all UniversalOpAmps in the .../examples/Applications folder in which all available parameters have been specified.



13.6 Transformers / Transformators

13.7 Solar cells

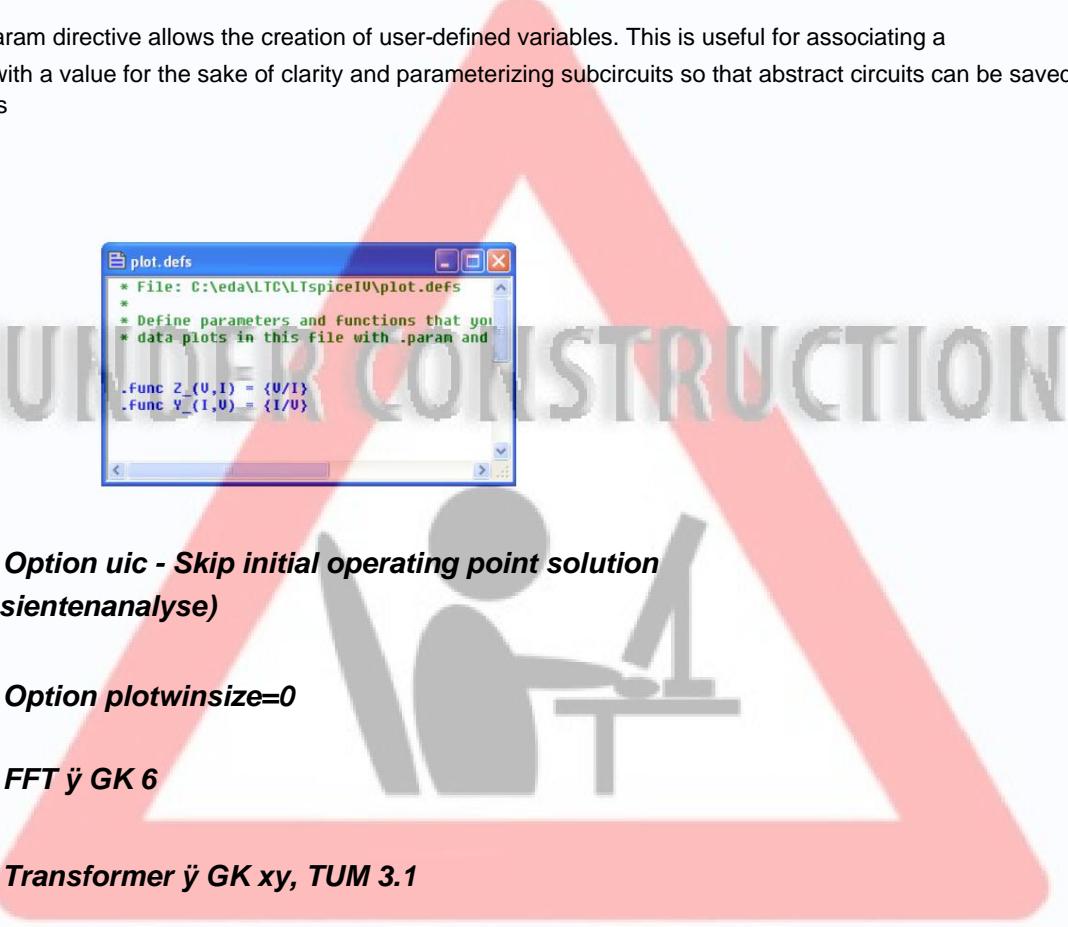
14 Additional information and techniques

14.1 User-defined functions and parameters / Plot Definitions File ...

Syntax: .func <name>([args]) {<expression>}

Example: .func Pythag(x,y) {sqrt(x*x+y*y)}

The .param directive allows the creation of user-defined variables. This is useful for associating a name with a value for the sake of clarity and parameterizing subcircuits so that abstract circuits can be saved in libraries



```
plot.defs
* File: C:\eda\LTC\LTspice10\plot.defs
*
* Define parameters and Functions that you
* want to use in your plots with .param and
* .Func definitions.
*
.Func Z_(V,I) = {V/I}
.Func Y_(I,V) = {I/V}
```

14.2 Option uic - Skip initial operating point solution (Transientenanalyse)

14.3 Option plotwinsize=0

14.4 FFT ѹ GK 6

14.5 Transformer ѹ GK xy, TUM 3.1

14.6 Controlled sources ѹ TUM 3.2