WÜRTH ELEKTRONIK

WÜRTH ELEKTRONIK

THE LTSPICE IV SIMULATOR

MANUAL, METHODS AND APPLICATIONS

WÜRTH ELEKTRONIK

THE LTSPICE IV SIMULATOR

GILLES BROCARD

MANUAL, METHODS AND APPLICATIONS

Preface by Mike Engelhardt

1st edition

1st edition

Swiridoff Verlag
ISBN: 978-3-89929-258-9

Preface

It is an honor to write a preface for Gilles Brocard. I appreciate his work writing this book and hope you benefit from his labors.

LTspice has been fun to write. It let me implement a number of numerical methods that make LTspice better than traditional SPICE programs: a new numerical integration method, node reduction, a native circuit element that behaves like a power MOSFET, and new time step size control to name a few.

The biggest recent advance in LTspice was when it went multi-threaded in 2008. We found it easy to distribute the computations over multiple cores but challenging to make the simulation actually run faster. The problem was that the LTspice object code had been so optimized (much had already been implemented in optimized assembly language) that it didn't take very many microseconds per timestep and that was a short time compared to how well one can synchronize multiple threads. That's when we developed a means to dynamically adjust each threads' cache size to stochastic cool the threads to keep the work load spread evenly. Another important technique introduced at that time was code generation that generates an assembly listing optimized for your circuit. Then that code is assembled and linked by LTspice for execution. This self-authoring code is generated typically every few seconds during the simulation to help your circuit execute close to the theoretical flop limit of a modern CPU. That's why LTspice IV is fast.

But all this is for a purpose. I believe SPICE has impacted mankind more than any other simulator. Writing a better SPICE is important. LTspice offers you the ability to rapidly prototype your designs so that you understand them better and even develop intuition.

Mike ENGELHARDT
Manager of Simulation Development
Linear Technology Corporation
April 2011

Index

1 LTSPICE IV: PRESENTATION AND HISTORIC

1.1 Circuit simulation with LTspice IV

1.1.1 Three basic steps

With LTspice IV, circuit simulation is easy as 1, 2, 3 (see fig.1.1):

- **First step: Create the schematic and choose the type of simulation.** With the **schematics editor**, you can draw your circuit and add your comments. LTspice IV offers a wide range of components, however, if one or more models of components are missing, you can download them online. (See chapter 12 for more information about this).
  Depending on your requirements (continuous, alternative or transient analysis), choose a simulation directive and add the relevant source (see chapters 9 to 11 and 13). You can also add other commands (simulation directives) and configure all the elements, components values, etc.
- **Second step: Run the simulation.** You are asking the computer to carry out all calculations required to perform the chosen simulation, this is called running the simulation. This step takes a little time for all calculations to be done.
- **Third step: Take the measurements.** You choose the measures you require (current, voltage, frequency, power, etc.). The measurements are displayed instantly in a screen looking like the display of a high-performance digital oscilloscope. You can configure the display parameters in the same manner as you would on a real oscilloscope (reframing, change of scale, zoom, colour choice, screen fractioning, traces multiplication).

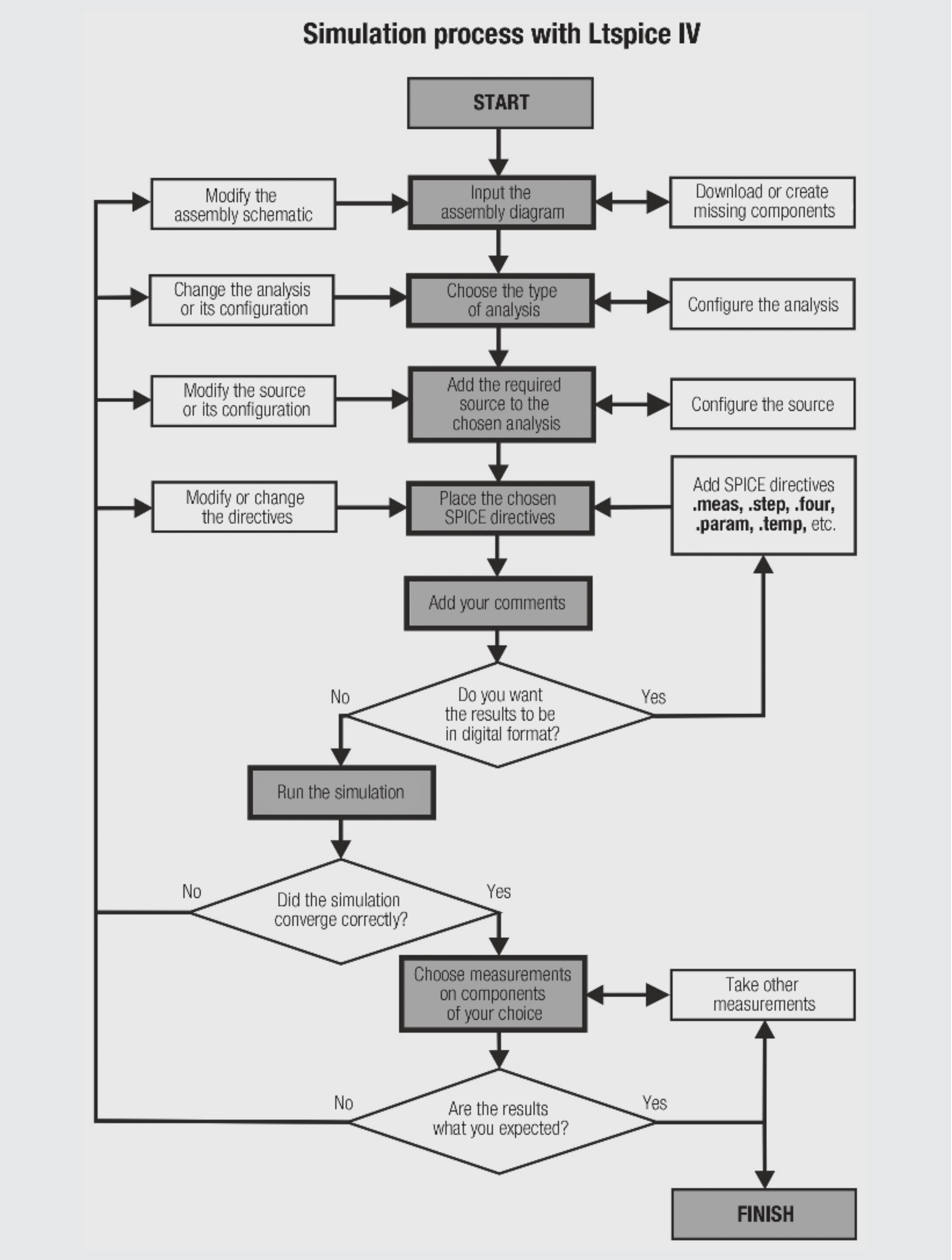
Simulation process with Ltspice IV
START
Modify the assembly schematic
Input the assembly diagram
Download or create missing components
Change the analysis or its configuration
Choose the type of analysis
Configure the analysis
Modify the source or its configuration
Add the required source to the chosen analysis
Configure the source
Modify or change the directives
Place the chosen SPICE directives
Add SPICE directives .meas, .step, .four, .param, .temp, etc.
Add your comments
No
Do you want the results to be in digital format?
Yes
Run the simulation
No
Did the simulation converge correctly?
Yes
Choose measurements on components of your choice
Take other measurements
No
Are the results what you expected?
Yes
FINISH

Figure 1.1

2. You create a new page for your schematic by clicking on the red icon on white background in the top left corner. You will see the background turn a lighter shade of grey and all 37 icons are now active.

The number of menus increases as well and their content is now relevant to the schematics editor you have just accessed.
LTspice IV is now waiting for you to either draw or open a schematic. The complete description of the new menus of the schematic editor can be found in chapter 4.

Open file	Pick visible traces
Save file	Autorange
Control panel	Tile windows
Run simulation	Cascade windows
Pause simulation	Close all
Zoom in	Cut
Pan	Search
Zoom out	Print setup
Zoom full extents	Print

Figure 3.4

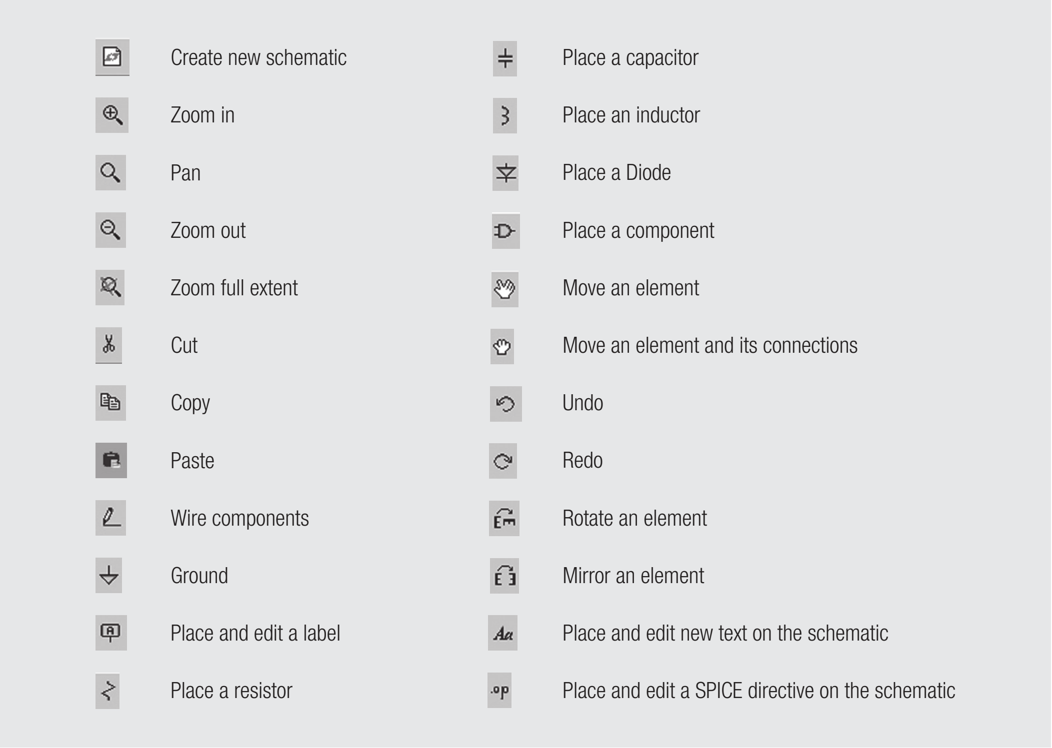

Figure 3.5

3.4.1 Drawing a schematic

3. We are not going to open a saved schematic, but we are going to create our own in order to go through all the creation steps. To start with, let's position on our schematic an operational amplifier symbol. To do so, click on the icon representing the **AND** logic gate symbol . Click once on this icon and another window appears. The left-hand column contains a series of words in brackets. In this column, double-click on **[Opamps]** to access the operational amplifier directory. (See figure 3.6).

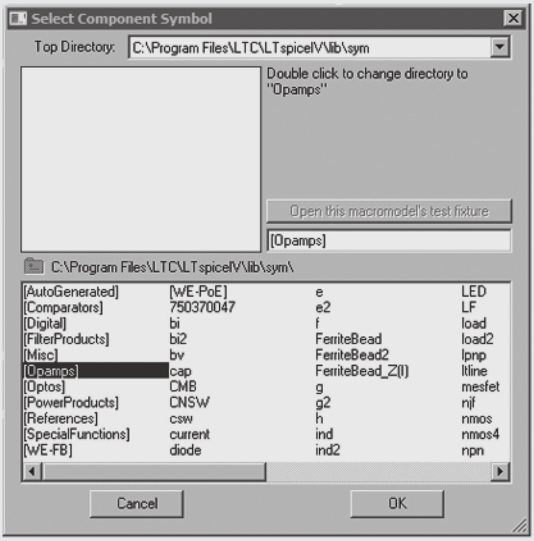

Figure 3.6

A list of operational amplifier appears, scroll to the very end of this list and double-click on **UniversalOpamp2,** which is currently the last item in the list. It is a standard operational amplifier with average performances. (Figure 3.7).

4. You automatically come back to the schematic page and your cursor is now shaped like an operational amplifier symbol. Go to the centre of the page and click left to drop the symbol on the schematic, figure 3.8. Another copy of this symbol appears straight away and follows the movement of your mouse.

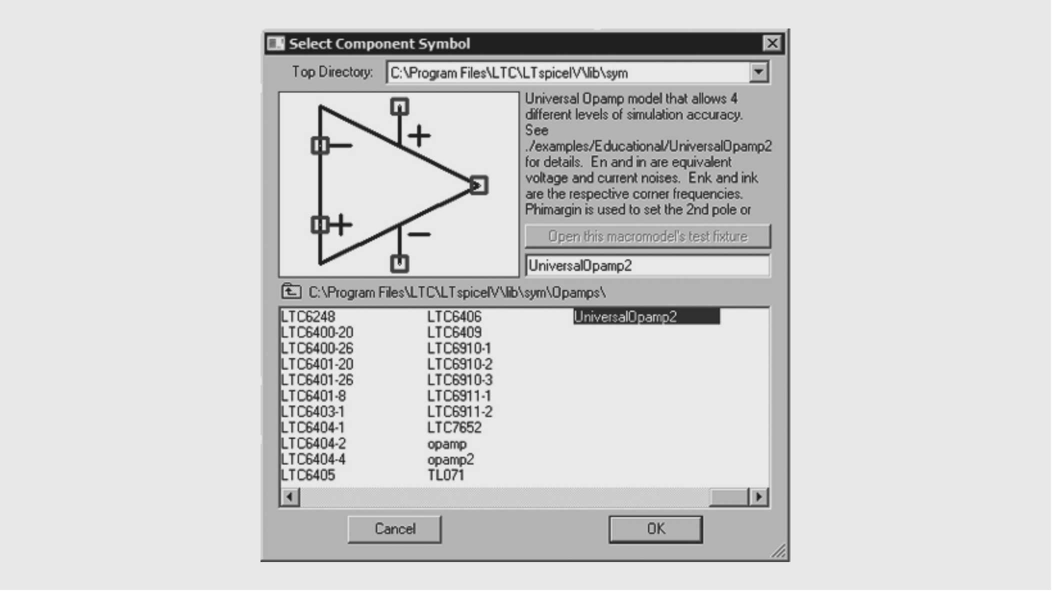

Figure 3.7

To get rid of it (we only want to use one operational amplifier on this schematic), right click anywhere, and the symbol disappears. So at the moment, the only item on our schematic is the operational amplifier in the centre.

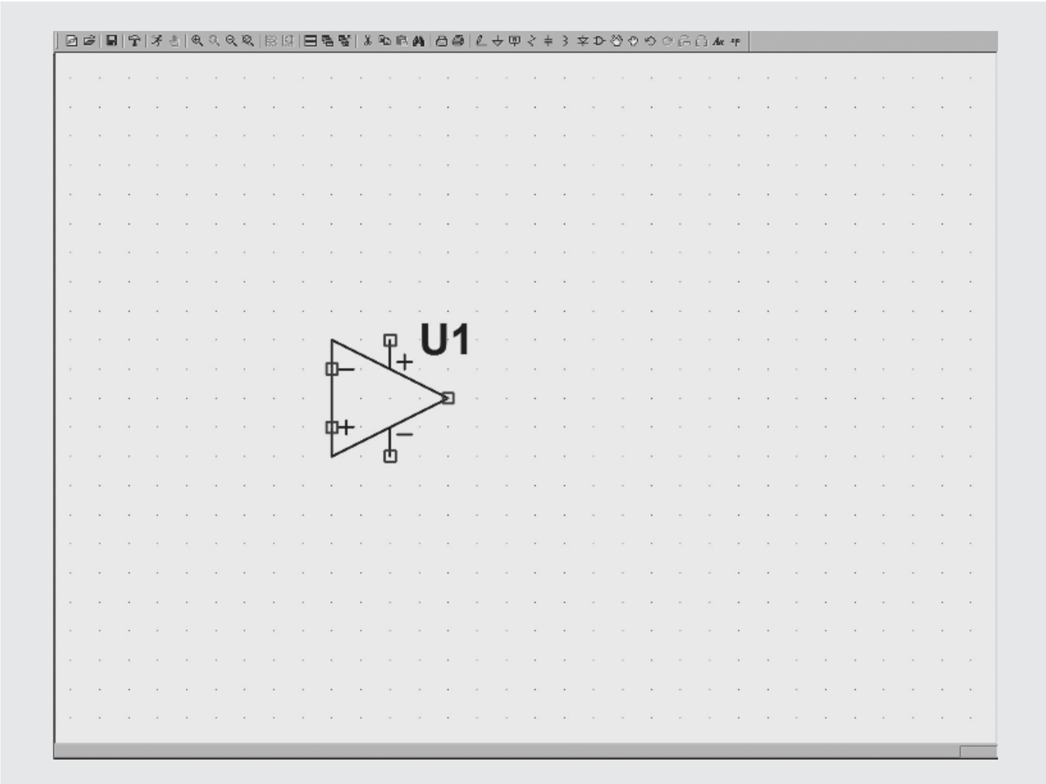

Figure 3.8

5. We are now going to add two transistors to our schematic. Once again, click on the **AND** icon , you can see that you are still in the operational amplifiers directory.
   So click on the yellow folder icon on the left to go back up one level in the directory.
   We are now in the standard component directory, in the 4th column, you'll see the word **npn**, double-click on it, see figure 3.9.
   As before, you find yourself straight back in the schematic page where you can place your NPN bipolar transistor by following the same procedure as for the amplifier: Click to drop, then, right-click to remove the copy.

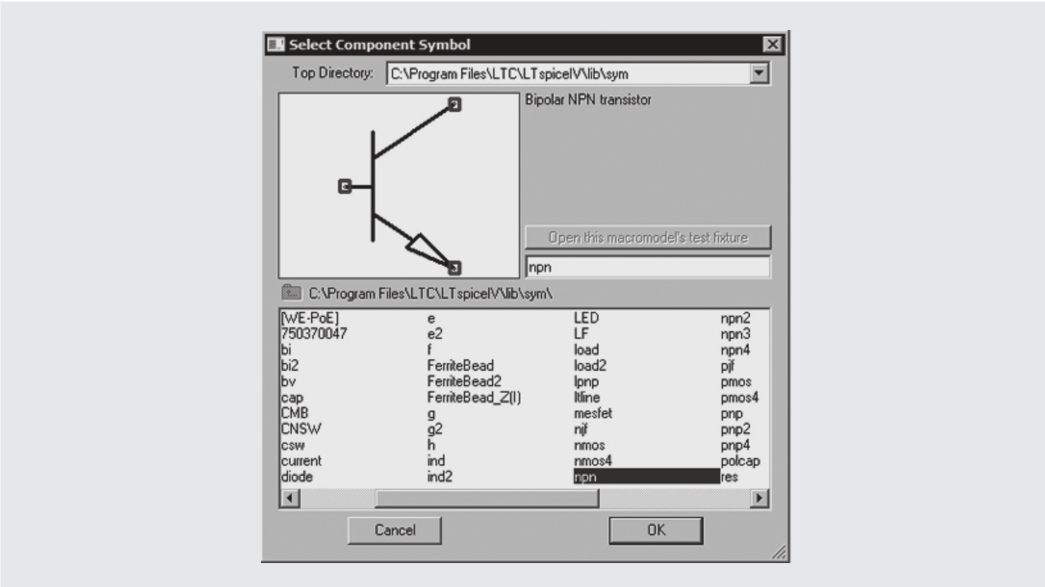

Figure 3.9

6. If you're not happy with the location of a component, you can remove it with . Click on the scissors, then click on the component you wish to remove, and lastly, right-click to exit the cut function. You can also use the open hand  to move a component. Click on the hand, then click on the component, it changes colour, move your mouse to place it where you want and click again to drop it, then right-click to exit the move function. You can also click and drag your mouse to draw a box around the section to move, then, move the box, click to drop it and right-click to exit.

7. We are now going to add another PNP bipolar transistor. Click on **AND** , and double-click on the word **pnp**, figure 3.10.
   Position the symbol as you see it on the schematic, but do not click to drop it, at this stage, you only have the outline of the symbol. Then to flip the symbol, you must use **twice** the rotation tool (command **Ctrl** + **R**) or  then once the symmetry tool (command **Ctrl** + **E**) or , figure 3.11. Once the symbol has the required orientation, click to drop it.