Introduction to LT-Spice

Anthony Mourliere, Patryk Strankowski and Jarosław Guziński

LTspice is based on standard software created to simulate electronic circuits called SPICE (Simulated program with Integrated Circuit Emphasis).

Spice was conceived in the 70s at the University of California (Berkeley) and has been improved over the years.

LTspice IV is a free high-performance simulator based on SPICE III, which has a schematic module, for editing electronic schematics as well as a visualization module of the results.

LTSpice contains more than 100 models of operational amplifiers, transistor models, logic gates etc ...

It is very useful for making a first draft of an electronic circuit and for understanding its operation.

LTspice is electronic simulation software that anticipates the characteristics and performance of an electronic circuit by assembling virtual components on the screen. From the spice core developed at Berkeley University but not very user-friendly, the company Linar Technology (LT) has developed a more visual version, easier to use, and free.
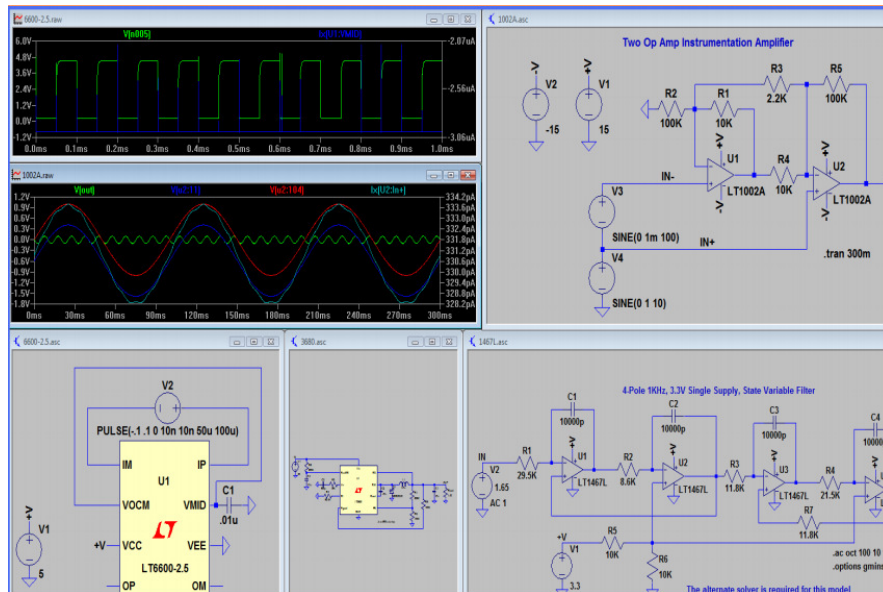
How does LTspice work?

LTspice produces an input file that contains the circuit description as well as the commands of the desired analysis type. The input file can be of two forms:

➢ Either we use the schematics schematic editor of LTspice which will automatically create an input file .asc
➢ Either we directly describe the circuit using lines of code in a text file called Netlist, extension .net
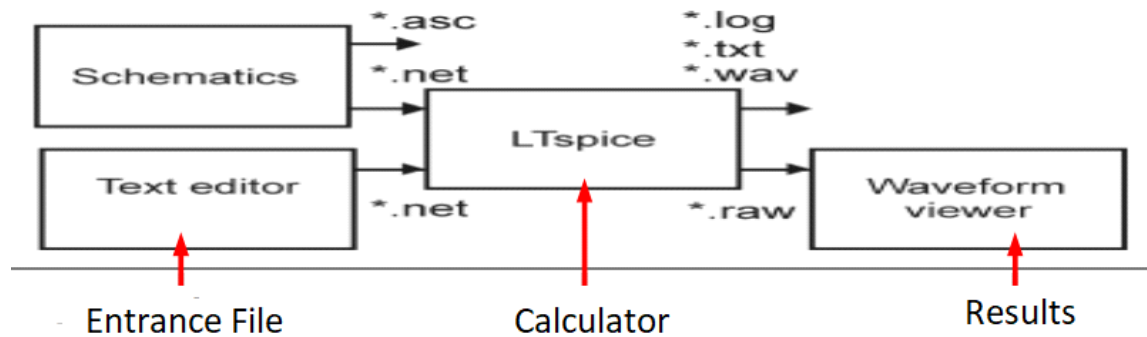

Drawing the circuit:

Adding a GND: This is very important. You cannot do any simulation on the circuit if you don't have a ground. To place a ground, you can

- press the 'g' key, or
- use the ground icon, ⏚, or
- get it from the 'Edit' menu.

If you aren't sure where to put it, place it near the bottom of the drawing.

Getting the other Parts:

- The next thing that you have to do is get some or all of the parts you need.
- This can be done by
  - clicking on the icon for a specific component (below picture)
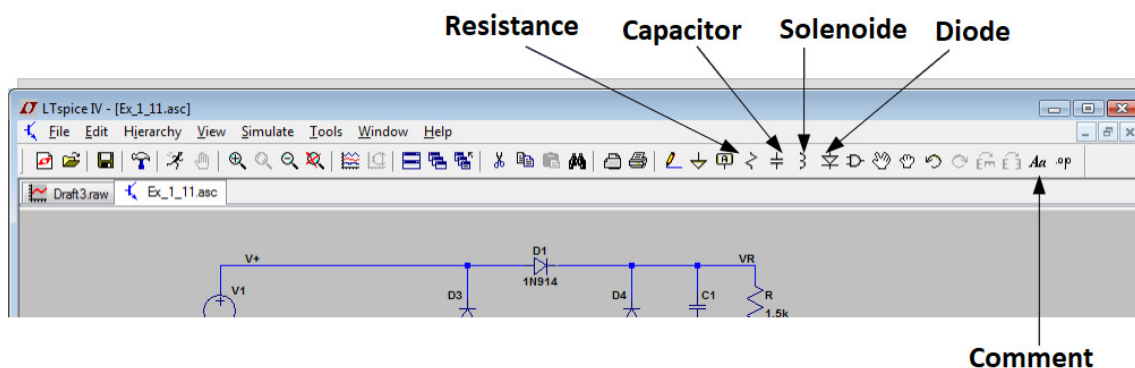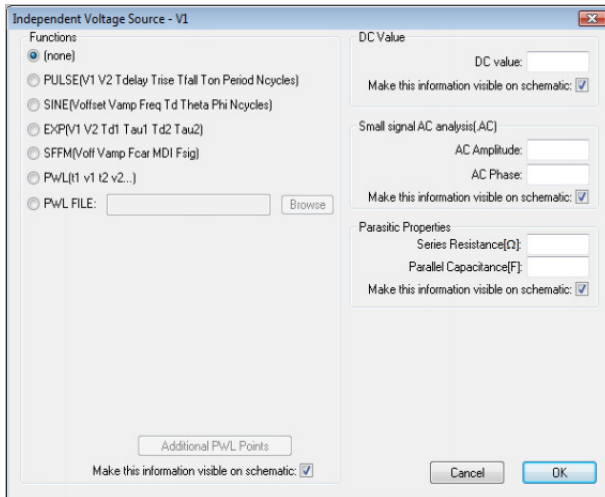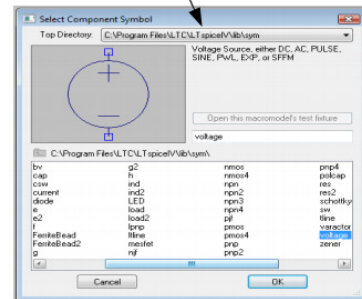

  - Left click on the desired symbol of the Schematic Edit bar

- Ctrl-R and Ctrl-M to rotate it or to get its symmetrical (mirror)
- Move the mouse where you want to position the element
- Left click to "fix" the item

Composants
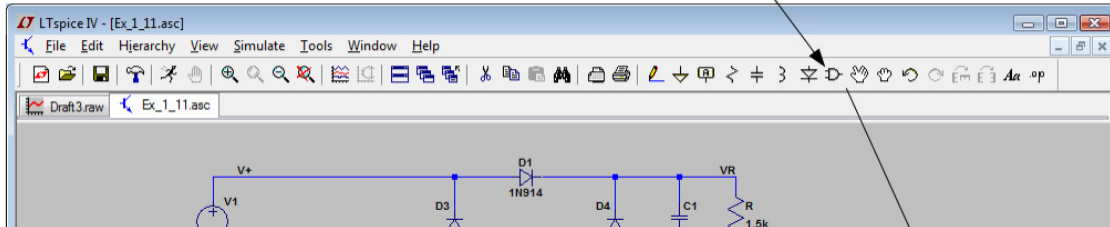
- Left click on the Component symbol of the Schematic Edit bar
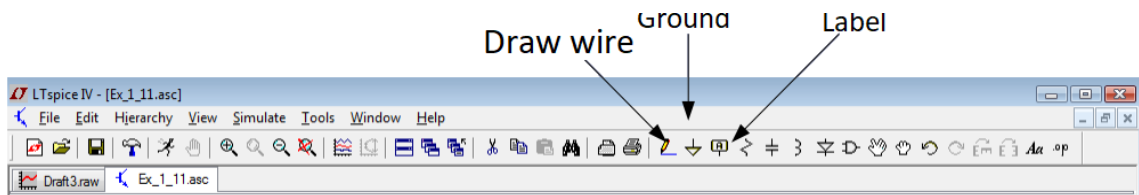- Select Voltage
- Left click OK

- Right click on this symbol then click on Advanced:
- The following symbol is obtained in Schematic for a voltage source
- Choice between different types of
- voltage sources:
- pulsed

For example we take a source of sinusoidal voltage with a frequency of 50Hz, voltage Voffset = 3V and Vamp = 10V
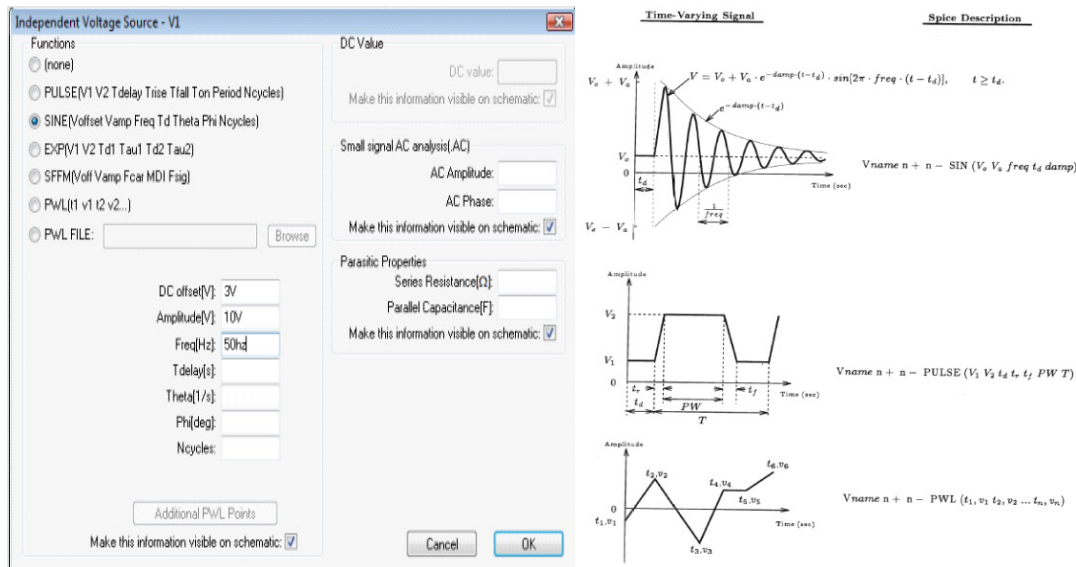
- *To create an electrical connection between the different elements, you must (Left):*
- *It is imperative to put a reference potential otherwise no simulation can work (Center).*
- *Labels are useful if you want to use your own node names (to find more easily a voltage or current in the circuit) instead of the numbering automatically provided by LTspice (Right)*

▪ Select the "Draw Wire" button	▪ Select the "ground" button	▪ To add a "Net label", press F4 or use the icon
▪ Left click on the starting node	▪ Left click where you want it	▪ A window will open:
▪ Left click on the arrival node	▪ to place	▪ Write the desired name
▪ Right click to stop the Draw option	▪ We can also use the "g" key as	▪ Click OK
▪ wire "	▪ shortens Right click to stop the Draw option	▪ Click once on a circuit wire where
	▪ wire "	▪ you want to get this nomenclaturekey as
		▪ shortensRight click to stop the Draw option
		▪ wire "
		▪ Name the positive potential of the source Vin, the reference potential Vref and the voltage of the resistance Vout

- *Polarization Analysis:* This is an extremely powerful analysis since it calculates the steady state of the circuit, therefore, the set of voltages and currents through the circuit components.
- *Transient Analysis*: It calculates all the variables of the circuit as a function of time.
- *Analysis Small AC signal:* With this analysis, LTSpice automatically calculates the polarization point of the circuit and then establishes the equivalent small-signal scheme of all the non-linear elements of the circuit (diodes, bipolar transistors, etc ...). She allows them to display the amplitude and phase of the different magnitudes of the circuit as a function of the frequency when an infinitesimal amplitude signal is applied to the circuit.
- *DC scan (DC sweep):* This analysis makes it possible to visualize the influence of a DC variable on the other parameters of the circuit (currents, voltages, ...)
- *Noise*
- *DC Transfer*

The advantages of LTSPICE:

- ➢ An unlimited number of nodes
- ➢ A Schematics editor to draw circuits
- ➢ A visualization editor of the results
- ➢ A large library of components
- ➢ Free