

Analog Electronic Circuits Tutorial - LTspice

Instructor: Prof. Abhishek Srivastava



About LTspice, Download and Install

- SPICE: Simulation Program with Integrated Circuit Emphasis
- LTspice is a high-performance SPICE simulation software, schematic capture and waveform viewer with enhancements and models for easing the simulation of analog circuits
- For SPICE, any circuit is described as an interconnection of various active, passive elements. This interconnection of elements is also called **Net-List**
- DC, transient, AC, pole-zero, noise - analysis can be performed using LTspice
- Result plots can be viewed and saved
- Download LTspice from following path and install:
<https://www.analog.com/en/design-center/design-tools-and-calculators/ltspice-simulator.html>
- LTspice documentation can be also downloaded from the same site

Getting Started

- Circuit schematic for design and analysis
- Models used to describe circuit elements are included
- Type of analysis to be done on the circuit to be mentioned in SPICE directive
- Results can be plotted with the help of probes and expressions

Getting Started-Creating New Schematic

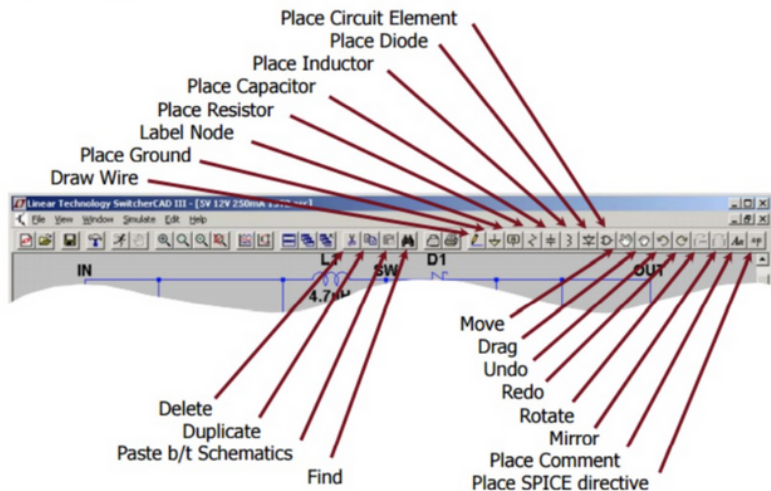
Start with a New Schematic

New Schematic



Getting Started-Toolbar

Summary of Schematic Editor Toolbar



LTspiceworkspace

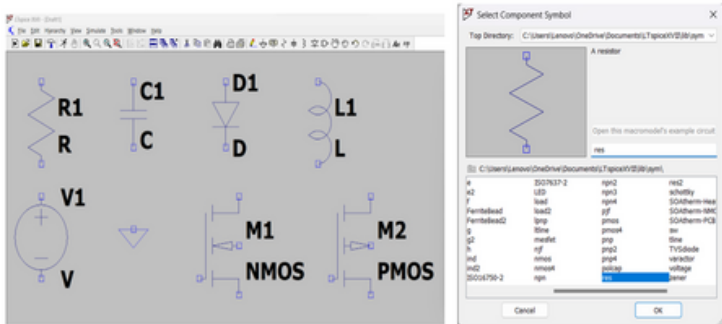


Figure: LTspice workspace and component window

- Gray region is a schematic workspace where you place components. Components can be instantiated from the components tab given in the toolbar.

You can search for required elements in the component window. 5/19

Units

Use the following labels to specify units for circuit element attributes

◆ **K** = k = kilo = 10^3

◆ **MEG** = meg = 10^6

◆ **G** = g = giga = 10^9

◆ **T** = t = terra = 10^{12}

◆ **m** = M = milli = 10^{-3}


◆ **u** = U = micro = 10^{-6}

◆ **n** = N = nano = 10^{-9}

◆ **p** = P = pico = 10^{-12}

◆ **f** = F = femto = 10^{-15}

Creating circuit schematic

- Place the required circuit element.
- To choose the resistance value, right-click on the resistor, which gives a dialogue box and enter the resistance value.
- To give the voltage value, right on the voltage source and enter your voltage value needed.
- Use **label function** from  the toolbar to label nodes.

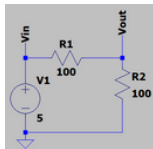


Figure: Voltage divider schematic

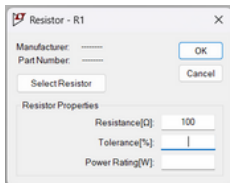


Figure: Resistance dialogue box

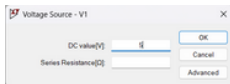


Figure: Voltage dialogue box

Simulation window

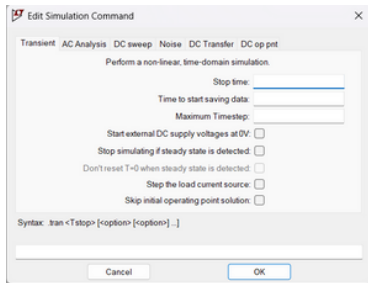



Figure: Simulation window

- Click on  in the toolbar to run the simulation. A window will appear, which gives different simulation options.
- Choose the necessary simulation option and enter the required setup. Click OK.
- This will create spice syntax corresponding to your simulation setup.

Operating Point

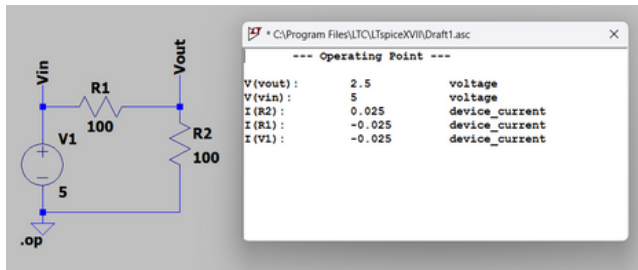


Figure: DC opt point

- In the simulation window, choose "DC opt pnt" and click OK.
- A dialogue box will appear which all the node voltages and branch currents. Be careful with the sign convention.
- You can also run the operating point by writing the .op command in SPICE directive and click on 'RUN' option in the toolbar.

DC sweep

- In the simulation window, choose "DC sweep" and enter the Voltage source you want to sweep(instance name should be exact).
- Enter the start and stop values with the increment needed. Choose the sweep type based on your range.
- After running the simulation, probe the required voltage and current by clicking on the appropriate node or branch.
- The graph shown in fig. corresponds to the circuit given in slide 9.

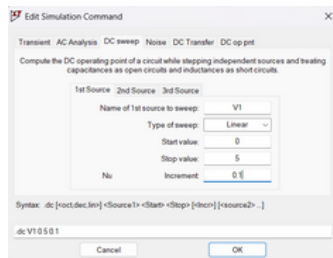


Figure: DC Sweep

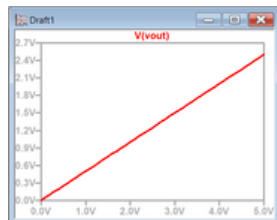


Figure: Vout vs Vin

Probing Circuit

- Left-click on any wire to plot the voltage on the waveform viewer



Voltage probe cursor

- Left-click on the body of the component to plot the current on the waveform viewer



Current probe cursor

- For voltage difference across two nodes, left click and hold on a node and drag the mouse to the other node

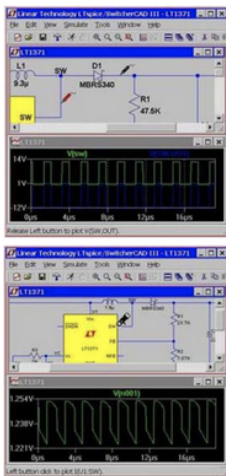


Figure: Edit simulation command window

Signal generator

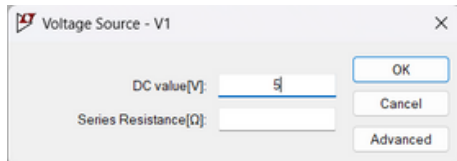
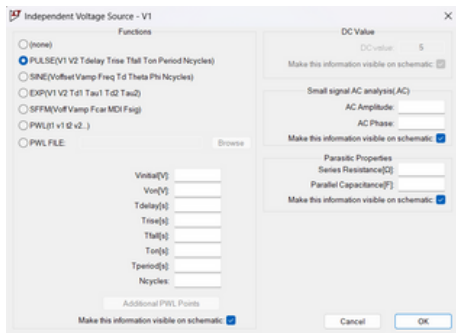


Figure: Voltage source window

Figure: Signal generator window

- To generate different voltage signals, right-click on the voltage source and click on advanced.
- This Independent sources window gives different voltage waveform options for use with the required signal parameter.
- Used to generate a signal for transient analysis.

Transient Analysis

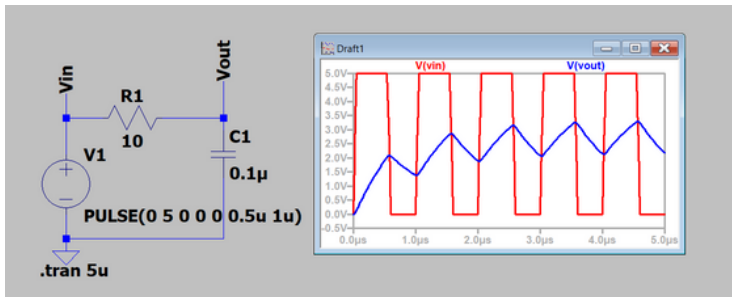
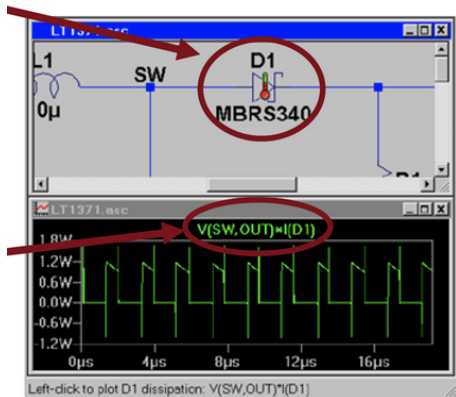


Figure: Transient analysis

- To run the transient simulation, select the appropriate voltage signal needed and run the simulation.
- Select the Transient simulation option and enter the stop time appropriately. Click OK
- After running the simulation, select the respective node/branch which you want to observe.

Instantaneous and average power dissipation

- Instantaneous power dissipation
 - Hold down ALT key and left click on the component
 - Pointer will change to a thermometer
- Average power dissipation
 - Hold down Ctrl key and left click on the trace label of the power dissipation waveform



AC Analysis - Low pass filter



Figure: Setup

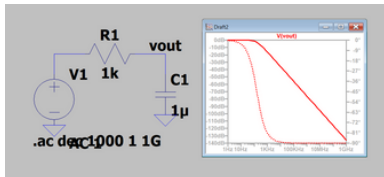


Figure: AC analysis

- To do AC analysis, first set the AC source magnitude to 1V in the advanced setting of the voltage source (given under small signal parameters).
- Select the AC analysis option in the simulation window and enter the required setting. After running the simulation, select the Output node you want to analyze.
- The solid line shows the magnitude plot, and the dotted line shows the phase plot.

Diode characteristics - Testbench

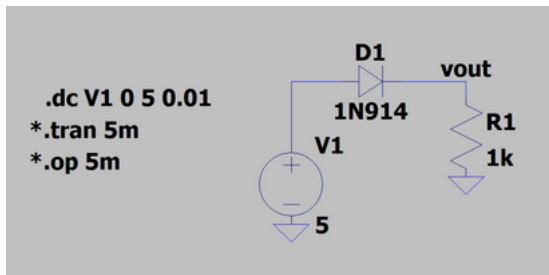


Figure: DC analysis Setup

- Creating a test bench is useful when running multiple analyses for the same schematic.
- Uncomment the analysis you want to run as shown in the figure.
- Very helpful in for the design of complex non-linear circuits.

Including model file

- To use pre-made models of devices such as opamps and transistors
- Right-click on the device symbol and edit properties to match the model of the device. For example, place an opamp and change the model name to LM741 for the given LM741 model file.
- Include model file in spice directive. Example: *.include lm741.lib* (Please mention the correct path)

The image shows a SPICE circuit diagram with three MOSFETs labeled M1, M2, and M3. M1 and M2 are NMOS transistors, and M3 is a PMOS transistor. The circuit includes voltage sources V1, V2, V3, V4, V6 and current sources I1, I2, I3, I4, I5, I6. The MOSFETs are connected to a common source node labeled VGS. The MOSFET models are defined in the circuit file as follows:

```
.include TSMC_180nm.txt
.lib 'ts18sl_scl.lib' ff_18
.include IBM130nm.txt
```

The MOSFET properties are shown in the bottom right corner of the image:

- M1 NMOS**:
 - Model: CMOSN
 - Length(L): 1u
 - Width(W): 1u
 - Source Area(AS): 0
 - Drain Area(AD): 0
 - Source Perimeter(PS): 0
 - Drain Perimeter(PD): 0
 - No. Parallel Devices(M): 1
 - CMOSN in: 0.18u w: 1u
- M2 CMOSN**:
 - Model: CMOSN
 - Length(L): 1u
 - Width(W): 1u
 - Source Area(AS): 0
 - Drain Area(AD): 0
 - Source Perimeter(PS): 0
 - Drain Perimeter(PD): 0
 - No. Parallel Devices(M): 1
 - CMOSN in: 0.18u w: 1u
- M3 CMOSN130**:
 - Model: CMOSN130
 - Length(L): 1u
 - Width(W): 1u
 - Source Area(AS): 0
 - Drain Area(AD): 0
 - Source Perimeter(PS): 0
 - Drain Perimeter(PD): 0
 - No. Parallel Devices(M): 1
 - CMOSN130 in: 0.18u w: 1u

The circuit file also includes the following SPICE directives:

```
.dc v1 0 1.8 0.1 v2 0 1.8 0.3
*.dc v2 0 1.8 0.1
*.dc v2 0.5 1.8 0.01
*.dc i1 10u 1m 10u
*.dc v2 0 1.8 0.1 v3 LIST -0.9 0 0.9
*.op
*.tran 20m
*.ac dec 10 1m 1G
```

Step command

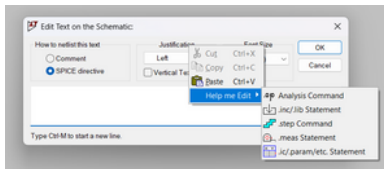


Figure: Resistance sweep

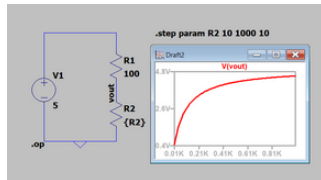


Figure: Resistance sweep

- Select the component value you want to sweep and give a name to the parameter in curly braces.
- Go to spice directive -> right-click on the syntax window -> Select help-in editor -> Select .step command option.
- The window will appear where you have to enter the parameter you want to sweep with the start and initial values followed by the increment value.
- The above figure shows an example of sweeping resistance value and plotting the Vout, respectively.

LTspice shortcuts on OS X

■ G	GROUND
■ S	ADD SPICE DIRECTIVE
■ CMD+N	NEW SCHEMATIC
■ CMD+O	OPEN
■ CMD+S	SAVE
■ CMD+Z	UNDO
■ CMD+SHIFT+Z	REDO
■ F2	COMPONENT
■ F3	WIRE
■ F4	NET NAME
■ F5	DELETE
■ F6	DUPLICATE
■ F7	MOVE
■ F8	DRAG
■ SPACE BAR	ZOOM TO FIT