

See discussions, stats, and author profiles for this publication at: <https://www.researchgate.net/publication/303565325>

# Circuit Simulation Examples using LTspice

Article · May 2016

---

CITATIONS

0

READS

1,123

1 author:



Guru Prasad

Manipal University

18 PUBLICATIONS 0 CITATIONS

SEE PROFILE

Some of the authors of this publication are also working on these related projects:



I am currently working on Layout of LDO in 180 nm technology [View project](#)



Tutorials on analog circuit design [View project](#)

All content following this page was uploaded by [Guru Prasad](#) on 27 May 2016.

The user has requested enhancement of the downloaded file.

# Circuit Simulation Examples using LTspice

Guru Prasad  
Assistant Professor  
Department of ECE  
MIT, Manipal

## Abstract

This article gives a brief introduction on how to use LTspice for simulating electronic circuits. The step by step procedure for drawing analog circuits and performing analyses like dc, ac, transient etc is explained. This article is not substitute for LTspice manual however it reduces the learning time.

## Introduction

SPICE stands for Simulation Program with Integrated Circuit Emphasis. Originally, SPICE was implemented at the University of Berkeley. Now many SPICE variants are available however they are basically derived from the above. Among SPICE tools Orcad Pspice, Texas TINA-TI spice, Ngspice, Winspice, LTspice are widely used. In the above list Ngspice and LTspice are freely available without any limitations. NG spice is mainly used in Linux operating system. LTspice is mainly used in windows operating system although it can be installed in linux space using wine tool.

LTspice has been created and supported by linear technology which can be downloaded from the linear technology website. Install the application file in the required folder like any windows application program.

In the next section the various simulation examples are illustrated which will give the basic idea about the usage of the tool.

## Simulation Examples

### 1. Simulation of a half wave rectifier

First circuit diagram is drawn as shown in Figure 1. Right clicking on the diode symbol and then *pick new diode* we see many preloaded diode models. I have taken 1N914 model. Similarly right clicking on voltage source symbol and then *advanced option*, any source type among pulse, sine, exponential, piecewise linear or from data file can be selected.

After saving the file as name.sch, navigate the menu bar as following

*Simulate — Edit Simulation cmd*

We can perform following analysis

- Transient : The timing waveforms of the circuit variables can plotted.
- AC analysis : Compute the small signal AC behaviour of the circuit linearised about its DC operating point
- DC sweep : Compute the DC operating point of a circuit while sweeping independent sources.

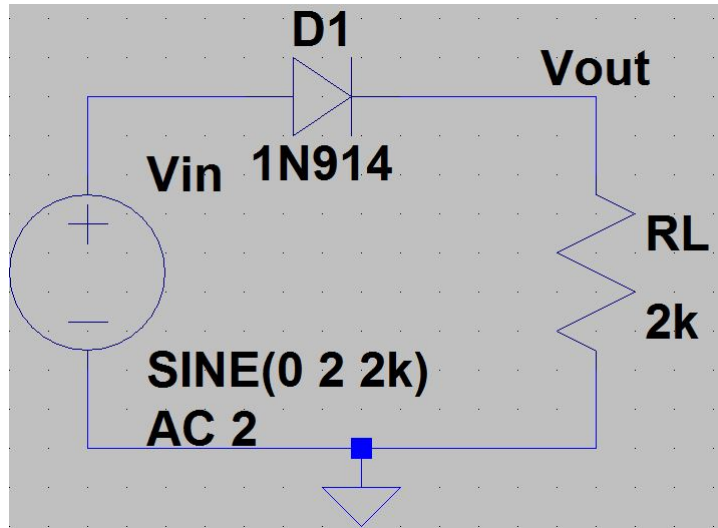


Figure 1: Half wave rectifier schematic diagram

- Noise : Perform a stochastic noise analysis
- DC Transfer : Find the DC small signal transfer function
- DC op pnt : Compute DC operating point.

Transient simulation is the suitable option for verifying half wave rectifier. Stop time is taken as 1 ms. After simulation run and clicking on output node in the schematic window, the waveform is plotted as shown in Figure 2.

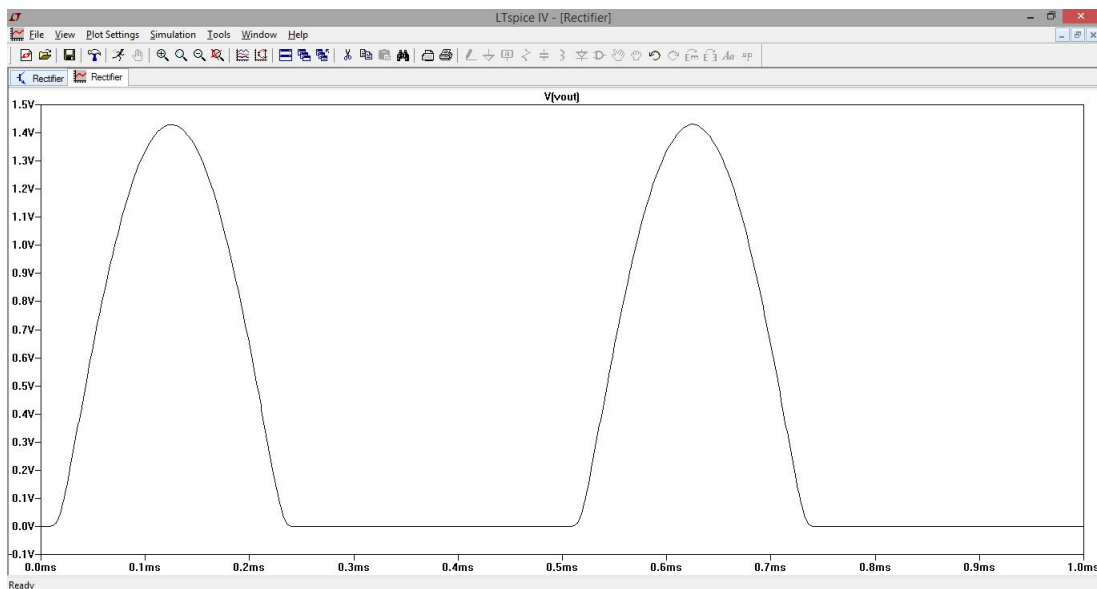


Figure 2: Transient response of Half wave rectifier

Since input is a sinusoidal of 2 kHz, the output obtained is half sine wave as expected.

## 2. Plotting the I/O characteristics of common Emitter amplifier

Draw the common emitter amplifier as shown in Figure 3. 2N2222 npn bjt transistor has been selected.

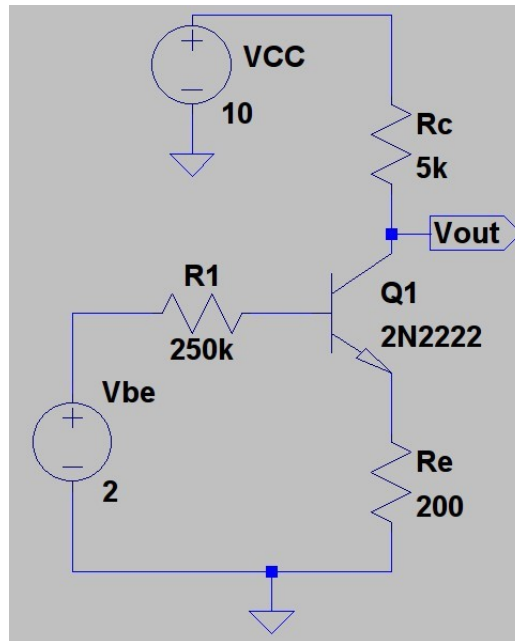


Figure 3: Common Emitter amplifier

To find the DC operating point, in the *Edit simulation cmd*, *DC op pnt* analysis is chosen. After simulation, the window as shown in Figure 4 pops up.

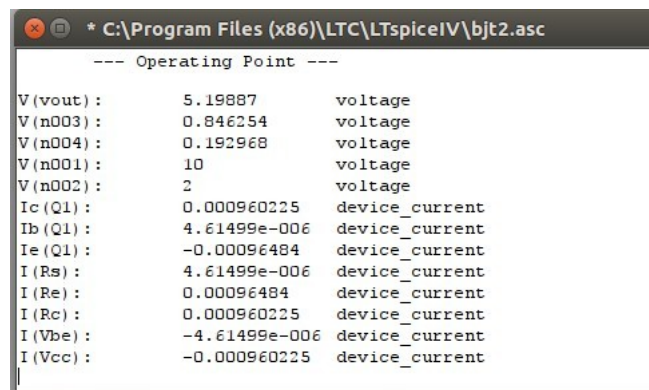


Figure 4: DC operating point of Common Emitter amplifier

Voltage at the collector node is 5.19 V and current through collector resistor is 0.96 mA.

To plot the input characteristics, in the *Edit simulation cmd*, DC sweep option is chosen. Sweep source, start value, final value and increment step are chosen as  $V_{be}$ , 0, 5 and 0.5 respectively. After simulation, input characteristic appears as shown in Figure 5.

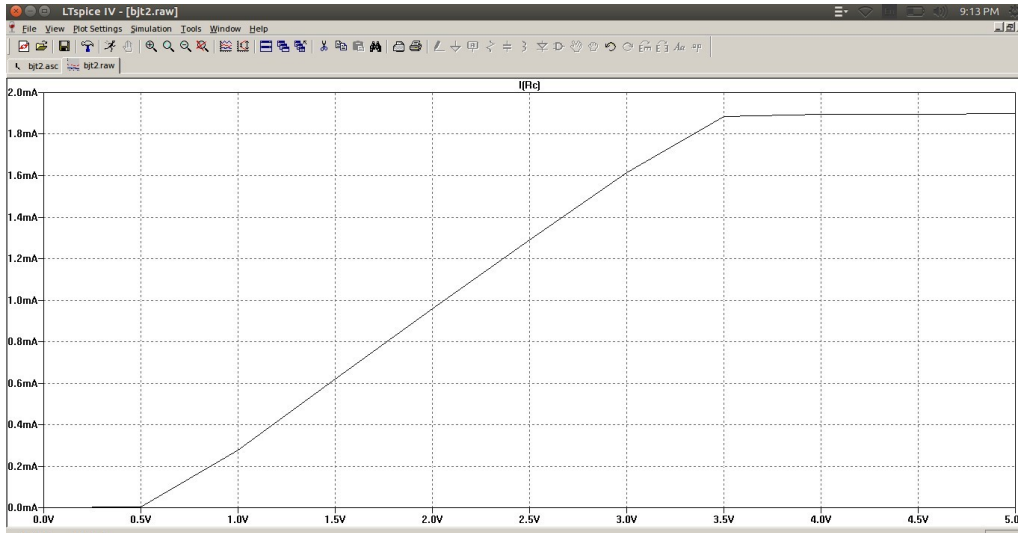


Figure 5: Input Characteristics of Common Emitter amplifier

To plot the output characteristics, in the *Edit simulation cmd*, DC sweep option is chosen.  $V_{CC}$  is taken as 1st sweep source and  $V_{be}$  is taken as 2nd sweep source. The sweep range for  $V_{CC}$  is from 0 V to 20V with the step size of 1V. The sweep range for  $V_{be}$  is from 0 V to 5V with step size of 0.5V. The simulation result is shown in Figure 6.

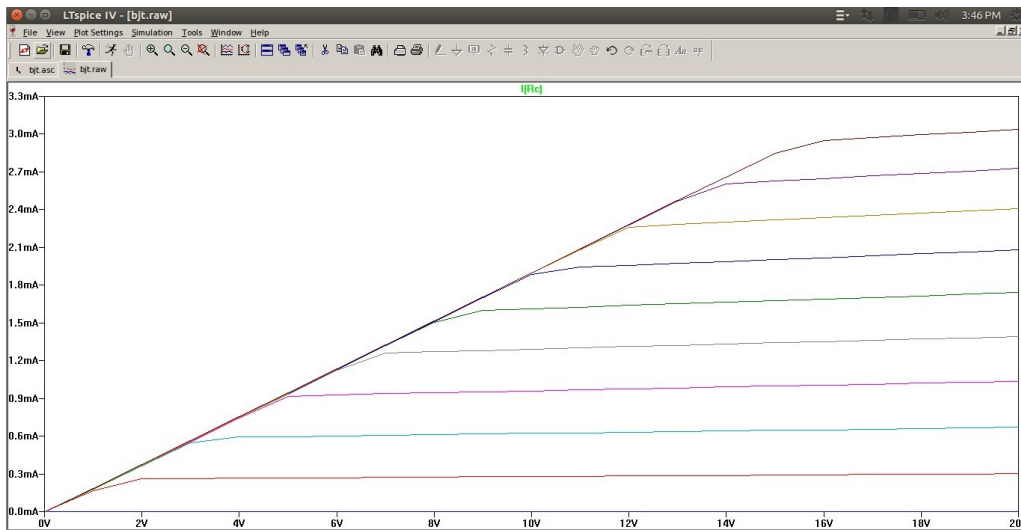


Figure 6: Input Characteristics of Common Emitter amplifier

### 3. Frequency response plot of RC coupled amplifier

Draw the RC coupled amplifier as shown in Figure 7.

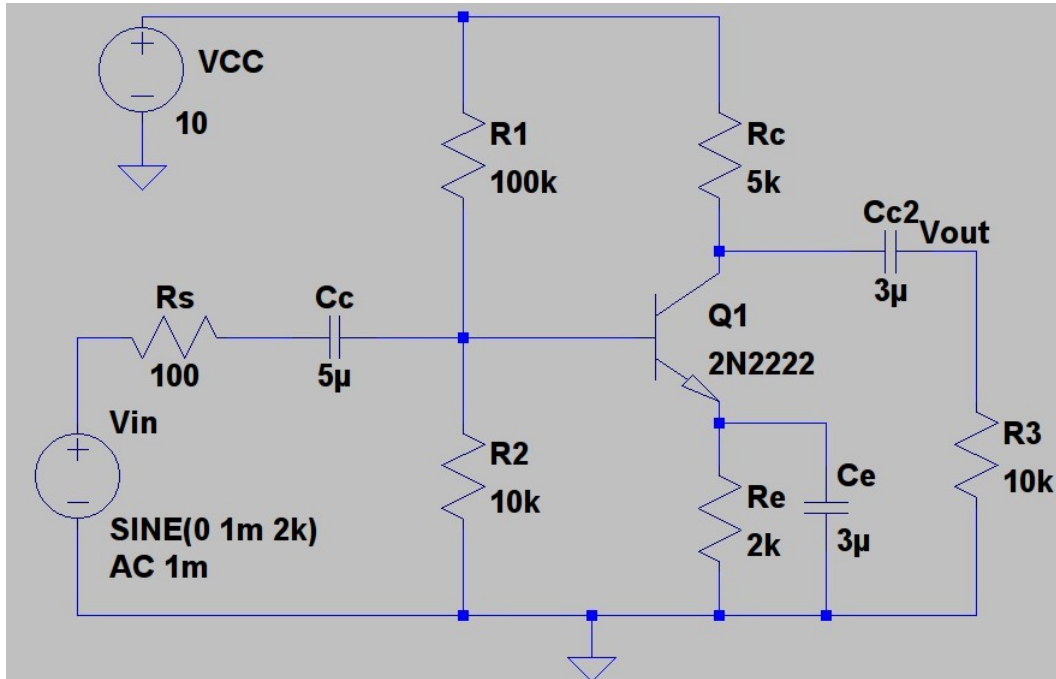


Figure 7: RC coupled amplifier

To plot the frequency response, in the *Edit simulation cmd*, AC Analysis option is chosen. Type of sweep is taken as decade. Starting frequency is set as 1 and final frequency as 10 MHz. The number of points per decade is set as 100. The simulation result is shown in Figure 8.

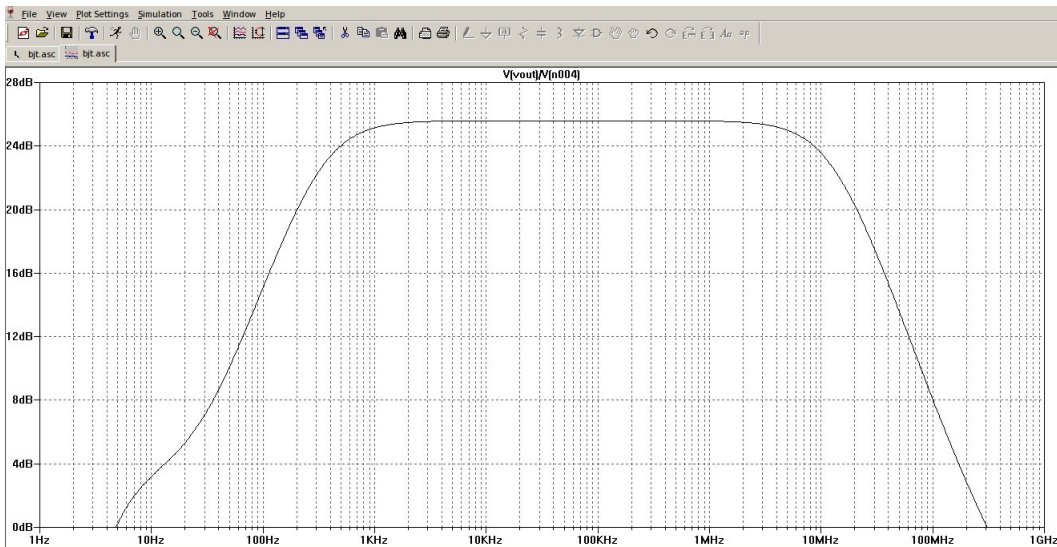


Figure 8: Frequency response of RC coupled amplifier

#### 4. Simulation of common source amplifier using SPICE LEVEL=1 model for MOSFET.

In this example, how to add model file for the circuit component is shown. It is very useful when designing Integrated circuit. For example, if we would like to design a two stage OTA using 180 nm technology then we have to add 180 nm technology file while simulating the circuit.

In the present example we need SPICE LEVEL=1 parameters for MOSFET. We can get that either from internet source or from any microelectronics text book. A typical  $0.5\mu\text{m}$  technology SPICE LEVEL=1 model is shown below

```
.MODEL MODN NMOS LEVEL=1 VTO=1 KP=50U LAMBDA=0.033  
GAMMA=0.6 PHI=0.8 TOX=1.5E-10 CGDO=5E-10 CGSO= 5e-10  
CJ=1E-4 CJSW=5E-10 MJ=0.5 PB=0.95
```

Where MODN is the name of model, .MODEL is spice directive and NMOS is the device type. To attach this model to MOSFET, click on *.op* icon in the tool bar then following window pops up.

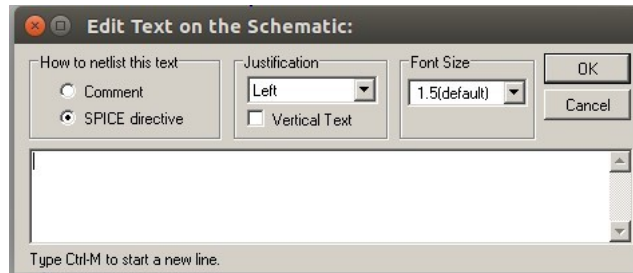


Figure 9: SPICE directive

Copy the parameter list and paste in the blank field provided in SPICE directive window. Click OK and place in the schematic window. Then holding ctrl key, right click on MOSFET symbol then following window pops up.

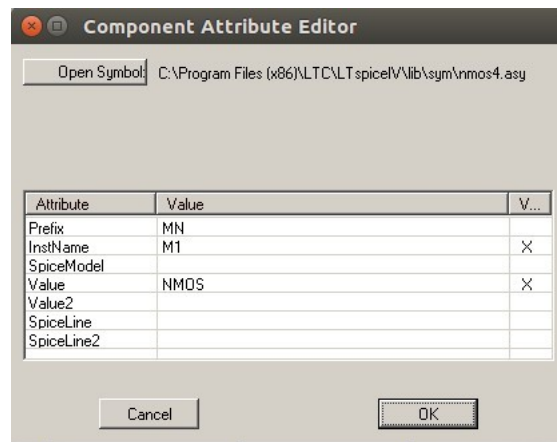


Figure 10: Component attribute editor

In the above window change *Value* form NMOS to MODN. Click OK. Again right click on MOSFET symbol, then following window pops up.

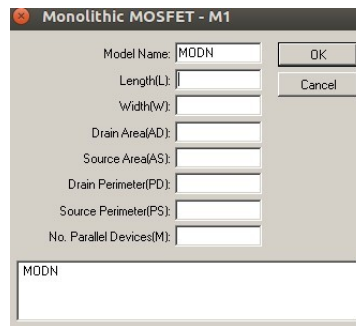


Figure 11: MOSFET parameter editor

Enter the width, length and other values.

The final circuit is shown in Figure 12

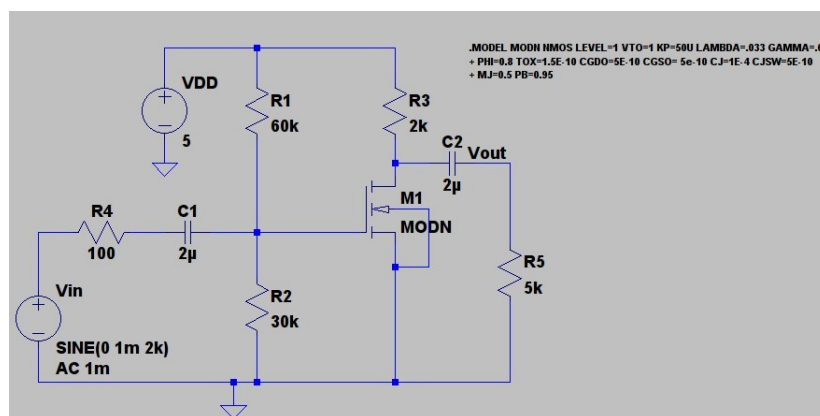


Figure 12: Common Source Amplifier

The transient response of Common source amplifier is shown in Figure 13. Both input and output timing wave forms are plotted.



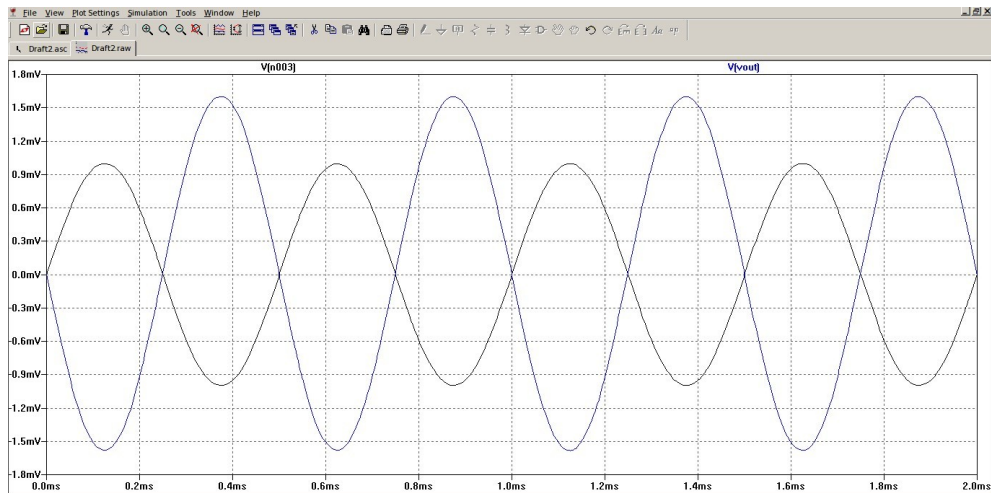


Figure 13: Transient response of Common Source Amplifier

## Conclusion

In this brief article I have explained how to use LTspice tool for drawing the circuit diagram and performing DC, AC and Transient analysis. I have also given one example in which how to attach technology file to circuit components is explained. I hope this article gives you an initial idea about how to use the tool and if you spend some time you can really feel comfortable using it.

## References

- [1] LTspice user Manual
- [2] Sedra and Smith *Micro Electronics*, 5th edition, Prentice Hall, 2010.