



# **LT SPICE SIMULATION**

## **PROJECT REPORT**

SUBMITTED IN PARTIAL FULLFILLMENT OF THE  
REQUIREMENT FOR THE AWARDS OF THE DEGREE OF

***DIPLOMA IN ENGINEERING***

***IN***

***ELECTRONICS ENGINEERING***

**BY**

**MUNAZIR REZA**

**MOHD. AMAAN**

**SHADAB ALI**

**MOHD. SHAHID**

UNDER THE SUPERVISSION OF

**MR. TEHZEEB AHMED ABBASI**

**(Associate professor)**

**ELECTRICAL ENGINEERING SECTION**

**UNIVERSITY POLYTECHNIC**

**ALIGARH MUSLIM UNIVERSITY**

**ALIGARH (INDIA)**

**2019-20**

## **Candidate's Declaration**

I hereby declare that the work, which is being presented in this dissertation, entitled “**LT SPICE SIMULATION**”, In partial fulfillment for the award of the degree of Diploma in Electronics Engineering in the Electrical Engineering Section, University Polytechnic, , Aligarh Muslim University Aligarh, is an authentic record of my own work carried out under the supervision of Mr.Mr.Tehzeeb Ahmad Abbasi, Electrical Engineering Section, University Polytechnic, A.M.U., Aligarh.

**(MUNAZIR REZA)**

Faculty No. 17DP EE215

Enrollment No.-GK1533

This is to certify that the above statement made by candidate is correct to the best of our knowledge.

**(Mr.Tehzeeb Ahmad Abbasi)**

Associate Professor

Electrical Engineering Section,

University Polytechnic, A.M.U, ALIGARH

## **ACKNOWLEDGEMENT**

I wish to express my heartfelt gratitude to those who provided me guidance and shared their time and experiences to give me full understanding and moral support to complete my dissertation. I would like to express my special thanks to Mr.Tehzeeb Ahmad Abbasi, as a supervisor for his invaluable suggestion, guidance and encouragement provided in carrying out this investigation.

I am also highly grateful to Mr.Mohd Zihaib Khan, Assistant Professor, Electrical Engineering Section, University Polytechnic for providing me facilities to complete my task with utmost satisfaction. I am thankful to Mr.Jamshed, Mr.Shahabuddin for their technical support throughout the experimental work. Finally, my endless thanks goes to my parents, my family members and relatives for being a constant source of motivation, inspiration and support.

MUNAZIR REZA

## **ABSTRACT**

This project report presents a LT SPICE simulation. LT SPICE is a SPICE based analog electronic circuit simulator computer software, produced by semiconductor manufacturer Analog Device. It is the widely distributed and used SPICE software in the industry. It is freeware.

LTspice provides schematics capture to enter an electronic schematic for an electronics circuit, an enhanced SPICE type analog electronic circuit simulator, and waveform viewer to show the results of the simulation. Circuit simulation analysis based on transient, noise, AC, DC, transfer function, DC operating point can be performed and plotted as well as fourier analysis.

LTspice does not generates printed circuit board (PCB), but netlist can be exported to PCB layout software.

LTspice used in many users in the field including radio frequency electronics, power electronics, audio electronics, digital electronics , and other disciplines.

LTspice is easy to use circuit designing software and can be used for analysis for any type of circuit. It is less time consuming and all circuits can be executed accuratelyIt's cheaper than buying a lot of real parts and a real signal generator and oscilloscope.

In simulation, the values of components can be change. It can save time by designing a prototype of a circuit, by simulation we can check the behaviours of the circuits and then performed on the hardware.

<b>SERIAL NO</b>	<b>CONTENTS</b>	<b>PAGE NO</b>
<b>1</b>	INTRODUCTION	2
<b>2</b>	WHY LT SPICE	19
<b>3</b>	EXPERIMENTS	26
<b>4</b>	CONCLUSION	53

# CHAPTER 1

## INTRODUCTION

**LTspice** (Linear Technology Simulation Program with Integrated Circuit Emphasis) is a free circuit simulator from the manufacturer Analog Devices that uses a mixture of Spice commands and circuit diagrams with a sizable library of passive and active components. The software also allows sub-circuits and Hierarchical circuits of any size, even from third-party sources to add components that are not currently available in the library. There is already a lot of support for the program and it always helps to have more if questions are answered vaguely online. Also, this tool is useful for testing out ideas that often use high current which requires plenty of safety factors when testing.

**LTspice** is a SPICE-based analog electronic circuit simulator computer software, produced by semiconductor manufacturer Analog Devices (originally by Linear Technology). It is the most widely distributed and used SPICE software in the industry. Though it is freeware, LTspice is not artificially crippled to limit its capabilities (no node limits, no component limits, no subcircuit limits).

**LTspice** provides schematic capture to enter an electronic schematic for an electronic circuit, an enhanced SPICE type analog electronic circuit simulator, and a waveform viewer to show the results of the simulation. Circuit simulation analysis based on transient, noise, AC, DC, DC transfer function, DC operating point can be performed and plotted as well as fourier analysis. Heat dissipation of components can be calculated and efficiency reports can also be generated. It has enhancements and specialized models to speed the simulation of switched-mode power supplies (SMPS) in DC-to-DC converters.

**LTspice** does not generate printed circuit board (PCB) layouts, but netlists can be exported to PCB layout software. While LTspice does support simple logic gate simulation, it is not designed specifically for simulating logic circuits.

It is used by many users in fields including radio frequency electronics, power electronics, audio electronics, digital electronics, and other disciplines.

### LT SPICE XVII

In 2016, LTspice XVII was released, and is currently the latest version. It is designed to run on 32-bit or 64-bit editions of Windows 7, 8, 8.1, 10, and macOS 10.9.

Summary of major changes from LTspice IV to LTspice XVII are:

- Add 64-bit executables.
- Add Unicode characters in schematics, netlists, plot.
- Add device equations for IGBT (Insulate gate bipolar transistor), diode soft recovery, arbitrary state machine.
- Add user-defined symbol and library directory search path settings to the LTspice control panel.

- Add schematic thumbnail and preview support on Microsoft Windows.
- Add editors for most SPICE commands.
- Add multi-monitor support.

## Device models

---

LTspiceships with thousands of third-party models. (capacitors, diodes, inductors, resistors, transistors, ferrite beads, opto-isolators, 555 timer, and more), as well as macro models for Analog Devices and Linear Technology parts (ADCs, analog switches, comparators, DACs, filters, opamps, timers, voltage references, voltage supervisors, voltage regulators, 0.01% quad resistor networks, and more). In the device library, Analog Devices part numbers start with "AD", and Linear Technology parts start with "LT".

LTspice allows a user to choose from device models that ship with LTspice, as well as allows the user to define their own device model, or use 3rd party models from numerous electronic component manufacturers, or use a model from a 3rd party device library. Starting with LTspice XVII, control panel settings were added to allow the user to specify search directories for 3rd party device symbols and libraries. See option setting at LTspice -> Tools -> Control Panel -> Sym. & Lib. Search Paths

The text that describes intrinsic SPICE models can be placed directly on an LTspice schematic by using the spice directive `.op` button. The advantage of this method is the 3rd party model is self-contained as part of the schematic when you distribute the schematic file. The same `.model` can also be copied to an ASCII text file on your computer too but it won't "travel" with a schematic when you copy it to another computer. For example, the following diode part numbers aren't included in the current LTspice device library:

```
.model 1N4004_WIKI D(Is=500p Rs=0.12 N=1.6 Tt=4u Cjo=40p M=0.35 BV=400 Ibv=5.00u
Mfg=BobCordellBook Type=Silicon).
```

```
.model 1N4007_WIKI D(Is=7.02767n Rs=0.0341512 N=1.80803 Tt=1e-07 Cjo=1e-11 Vj=0.7
M=0.5 Eg=1.05743 Xti=5 Fc=0.5 BV=1000 Ibv=5e-08 Mfg=OnSemiconductor Type=Silicon)
```

```
.model 1N5408_WIKI D(Is=63.0n Rs=14.1m N=1.70 Tt=4.32u Cjo=53.0p M=0.333 BV=1000
Ibv=10.0u Mfg=DiodesInc Type=Silicon).
```

## Number conventions

---

In LTspice, numeric values can be expressed in four different ways: integer (i.e. 1000), real (i.e. 1000.0), scientific e-notation (i.e. 1e3, 1.0e3), scale factor notation (i.e. 1K, 1K0).

If the first character after a number is not the letter "e" for scientific e-notation or a scale factor suffix (left column of table), then trailing characters are ignored.<sup>[26]</sup> For example, 5 is treated the same as 5V / 5Volt / 5Volts / 5 Hz / 5Hertz.

## Scale factors.

Integer and real numbers supports a scale factor (multiplier) suffix. These are based mostly on metric conventions.

The suffix (left column) can be upper / lower / mixed case, known as case insensitive.<sup>[26]</sup> For example, 1MEG / 1meg / 1Meg represents 1000000; 1k / 1K represents 1000.

Any appended text after the suffix (left column) is ignored. For example, 2MegHz / 2MegaOhm represents 2000000; 3mV / 3mOhm represents 0.003; 4uF / 4uHenry represents 0.000004.

In LTspice, any suffix (left column) can replace the decimal point of a real number, a common format for printed schematics. For example, 4K7 represents 4700, 1u8 represents 0.0000018.

## Node name conventions

---

In LTspice, a node/net (connection point) on the schematic can be labeled by using the **Label Net** tool button or **F4** key. The "Label Net" wizard has three choices for a label, two predefined graphical symbols (GND, COM), or a user-defined node/net name.

The two graphical symbols represent:

- **GND** - The ground symbol assigns a node with a special global net name of "0".
- **COM** - The COM symbol assigns a node with a net name of "COM", which doesn't have any special significance.

Historically, SPICE and older version of LTspice software only supported printable ASCII characters for node/net names, then LTspice XVII added support for Unicode characters.<sup>[3]</sup>

A user-defined name supports two optional features that can be prepended to the text name:

- An underscore causes an overbar to be placed above the entire name, which commonly means an active low signal. For example, "\_RESET" is shown on the schematic as "RESET".
- **\$G\_** - This means a node is global, no matter where the name occurs in the circuit hierarchy. For example, "\$G\_ENABLE" / "\$G\_ERROR". The ground symbol is treated in a similar way, but it does not have "\$G\_" prepended to it.

When a node/net name is placed on a schematic, it will have one of five different visual representations. Two are automatically determined, while three others are chosen by the "Port Type" field in the "Label Net" wizard.

- **None** - Bare text. This is the default..
- **Global** - "Rectangle" around the text. This is automatically shown for a global net name that starts with "\$G\_".
- **Input** - "Rectangle with triangle end" around the text. This is chosen by the "Port Type" field in the "Label Net" wizard.



- `Output` - "Rectangle with triangle on other end" around the text. This is chosen by the "Port Type" field in the "Label Net" wizard.
- `Bidirectional` - "Rectangle with triangle on two ends" around the text. This is chosen by the "Port Type" field in the "Label Net" wizard.

## File format.

---

Many of the LTspice files are stored as an ASCII text file, which can be viewed or edited with any ASCII text editor programs. One of the side benefits of an ASCII file format is that a schematic can be listed in a printed document / book / magazine / datasheet / research paper / homework assignment, which allows the reader to recreate LTspice files without electronic file distribution.

LTspice filename extensions:

- `asc` - schematic. It consists of a netlist based on SPICE text-based commands.
- `asy` - electronic symbol shown in a schematic.
- `cir` - external netlist input.
- `fft` - FFT binary output.
- `lib` - model library sub-circuits.
- `plt` - waveform viewer plot settings.
- `raw` - binary output, optional ASCII output.
- `sub` - subcircuit.
- `lib` / `sub` / `mod` / `model` - device model. While any file extension is allowed, users tend to gravitate towards common ones.

## Example

The following example can be viewed by copying each into two different text files. For each, copy the text in the gray box from this article, paste into an ASCII text editor, saving as a text file. Both files must have the same "base name" and sit in the same directory. To see it, opening the "asc" file with LTspice then click the "Run" button inside LTspice software.

- LTspice\_RC.asc
- LTspice\_RC.plt

## Schematic file.

LTspice schematics are stored as an ASCII text file with a filename extension of "`asc`".

The following example shows the contents from a small LTspice schematic file for a simple RC circuit with four schematic symbols: V1 is 10 volt DC voltage source, R1 is 1K ohm resistor, C1 is 1 uF capacitor, ground. The bottom three TEXT lines are:

- 1) a transient simulation directive with a stop time parameter of 10 ms (`.tran 10mS`).

- 2) a SPICE directive to set the initial condition of RC "out" net to zero volts (`.ic v(OUT)=0V`), and.
- 3) a text comment (title).

```
Version 4
SHEET 1 880 680
WIRE 224 96 128 96
WIRE 128 160 128 96
WIRE 224 192 224 176
WIRE 288 192 224 192
WIRE 224 208 224 192
WIRE 128 288 128 240
WIRE 224 288 224 272
WIRE 224 288 128 288
WIRE 224 304 224 288
FLAG 224 304 0
FLAG 288 192 OUT
IOPIN 288 192 Out
SYMBOL res 208 80 R0
SYMATTR InstName R1
SYMATTR Value 1K
SYMBOL cap 208 208 R0
SYMATTR InstName C1
SYMATTR Value 1uF
SYMATTR SpiceLine V=50
SYMBOL voltage 128 144 R0
WINDOW 123 0 0 Left 0
WINDOW 39 0 0 Left 0
WINDOW 0 7 10 Left 2
WINDOW 3 -20 57 Left 2
SYMATTR InstName V1
SYMATTR Value 10V
TEXT 120 344 Left 2 !.tran 10mS
TEXT 120 376 Left 2 !.ic v(OUT)=0V
```

### Plot file

LTspice waveform viewer plot settings are stored as an ASCII text file with a filename extension of "**plt**".<sup>[30]</sup> If this optional plot file is present, then all plot planes will automatically be displayed after the "Run" button is pressed, otherwise the user will need to click on each net to see the waveform(s). To create a plot file on Windows, after a plot graph is displayed, right-click on it and choose "File", then choose "Save Plot Settings".

The following example for the above schematic shows settings for a "transient analysis" simulation with two waveforms on one plot plane consisting of the RC voltage at "out" net and current through resistor R1, which are labeled V(out) and I(R1) at the top of the plot graph.

```

[Transient Analysis]
{
Npanes: 1
{
traces: 2 {524290,0,"V(out)"} {34603011,1,"I(R1)"}
    X: ('m',0,0,0.001,0.01)
Y[0]: ('',0,0,1,10)
Y[1]: ('m',0,0,0.001,0.01)
    Volts: ('',0,0,0,0,1,10)
    Amps: ('m',0,0,0,0,0.001,0.01)
    Log: 0 0 0
GridStyle: 1
}
}

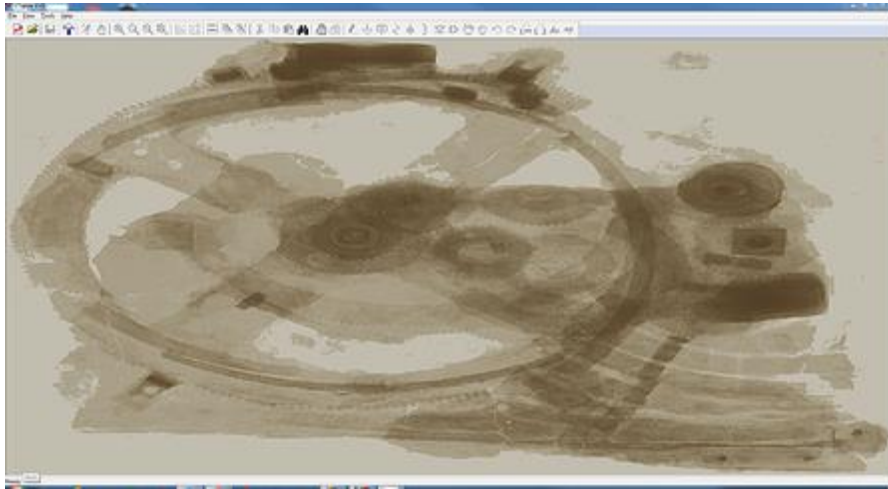
```

## Installation

This project report will talk about the Windows version as it is a little more user-friendly, there is a Macintosh version, but we found that version very difficult to use when we tried it. The installation is a typical executable installation process, follow the recommended Windows default installation process. When this is finished, there will be a new folder located at C:\Program\Files\LTC\LTspiceXVII\ where the default program is. we would recommend making a desktop shortcut by right-clicking the program named XVIIx64 (this might be different based on the operating system). Rename the program to something easier once the shortcut is on the Desktop. This is a type of program that needs to stay inside the installation folder for it to function properly because of how the libraries and other important running files work.

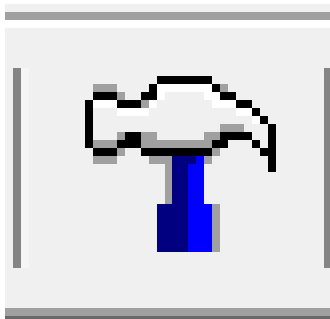
## Basic User Interface

When the program is opened, the main interface will be seen:



The software uses an embedded window for the circuit space, drawings for new parts, and the embedded “oscilloscope”. The windows can be cascaded/rearranged and the program can run several circuits (limits window space) at once. There will be no schematic loaded from starting the program, so a new one has to be made by clicking the icon under File to start a new circuit draft or using the File drop-down menu. The main tools used in this program are listed under File, View, Tools, and Help. One can hover over the toolbars to see what function they provide. The ones I use the most are Zoom full extents, Run, Auto-range, and the basic components such as wire, ground, label net, capacitors, inductors, diodes, and other components (the one that looks like an AND gate). There is a list of shortcuts that can be used for moving around the parts, copying, pasting, deleting, mirroring, rotating listed in the control panel:

1. Click the hammer for the control panel.



- Click the tab that says Drafting Options.

Compression Save Defaults SPICE Drafting Options

Allow direct component pin shorts[\*] ☐

Automatically scroll the view[\*] ☒

Mark text justification anchor points ☐

Mark unconnected pins ☒

Show schematic grid points[\*] ☐

Orthogonal snap wires ☒

Ortho drag mode[\*] ☐

Cut angled wires during drags ☒

Un-do history size: 500

Pen thickness[\*]: 1

Show Title Blocks[\*] ☒

Reverse Mouse Wheel Scroll[\*] ☐

Font Properties[\*]

Arial

Size[\*]: 28

Bold[\*] ☒

Color Scheme[\*]

Hot Keys[\*]

[\*] Setting remembered between program invocations.

Reset to Default Values

- Click the Hot Keys[\*] button for all the shortcuts, these can be edited too.

Schematic Symbol WaveForm Netlist

Place Component: F2

Draw Wire Mode: F3

Place Netname: F4

Delete Mode: F5

Duplicate Mode: F6

Move Mode: F7

Drag Mode: F8

Undo: F9

Redo: Shift+F9

Place Comment Text: T

Place SPICE Directive: S

Unconn. Pin Marks: U

Text Anchor Marks: A

Place Ground: G

Place Resistor: R

Place Capacitor: C

Place Inductor: L

Place Diode: D

Draw Lines: (none)

Draw Rectangles: (none)

Draw Circles: (none)

Draw Arcs: (none)

Reset T=0: 0

Schematic Grid: Ctrl+G

Rotate: Ctrl+R

Mirror: Ctrl+E

Zoom Area: Ctrl+Z

Zoom Back: Ctrl+B

Halt Simulation: Ctrl+H

Run Simulation: (none)

Place BUS Tap: (none)

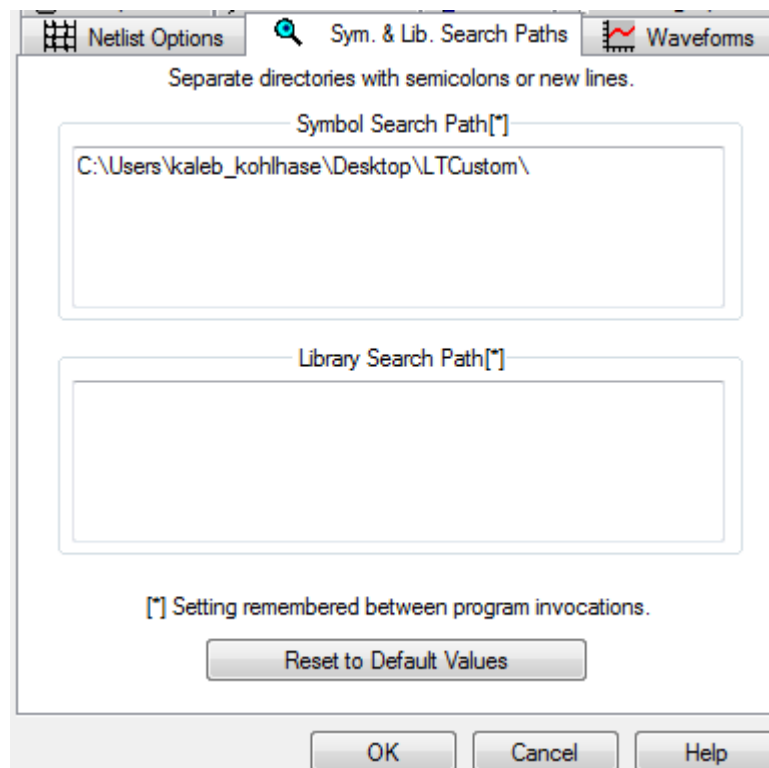
Zoom to Fit: Space

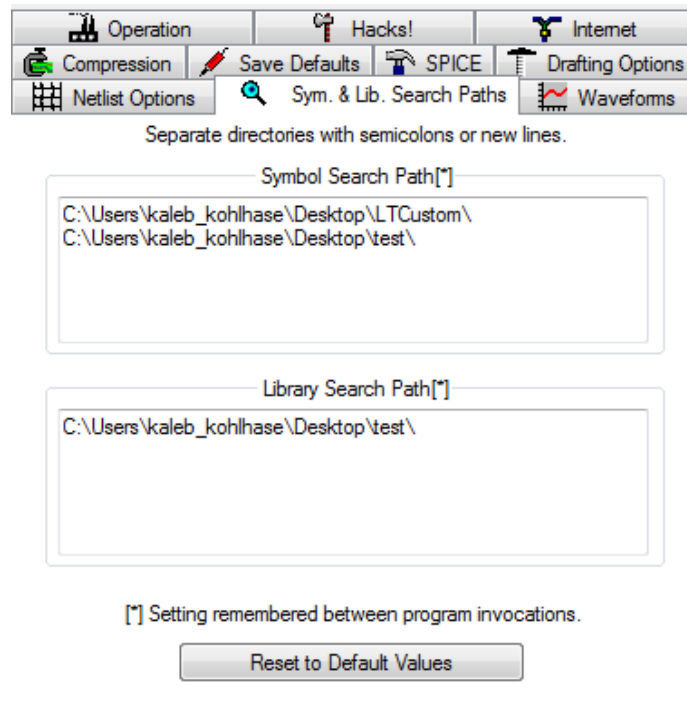
View SPICE Error Log: Ctrl+L

Cancel Reset to Default Values OK

Before I cover the last section for this first post, there is one other suggestion. Make a local folder on the Desktop or wherever to save schematics, symbols, sub-circuits, and plots. This makes it easier to save circuits in one place and then be able to reference the same folder for sub-circuits or Hierarchical circuits instead of trying to set up different folders and becoming confused later on. Linking this folder to the directories LTSpice searches will make other topics easier by having one master folder. The first step is to simply make a folder in the desired location, I will use the Desktop as my example. The destination address has to be known for this to make this work, my path to the folder I am using will be "C:\Users\kaleb\_kohlhase\Desktop\test\ ". All the parts must be written down or copied in the address, including the working directory letter (mine is C:) After the path is copied or in a separate window, click the control panel button (hammer) again and click the Sym& Lib. Search Paths tab:

1. Search paths tab:



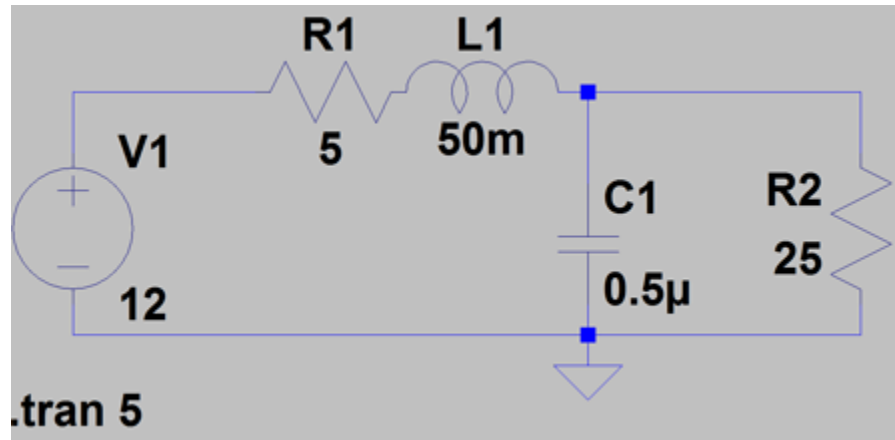


2. Paste or type in the path for Symbol Search Path and Library Search Path.

Each new path is separated by a return or semicolon, disregard my LTCustom path. After that, click Ok at the bottom of this prompt. These should be conserved between sessions according to the note at the bottom; if a sub-circuit breaks in the future: this is likely the source if library paths are de-referenced.

## Basic Circuit Simulation Example

This section of this report will be a basic RLC circuit with one DC voltage source and how to use the proper tools for getting started on analysis. For this example, there will be a resistor and inductor in series, one capacitor in parallel, and one further resistor in parallel to that capacitor.

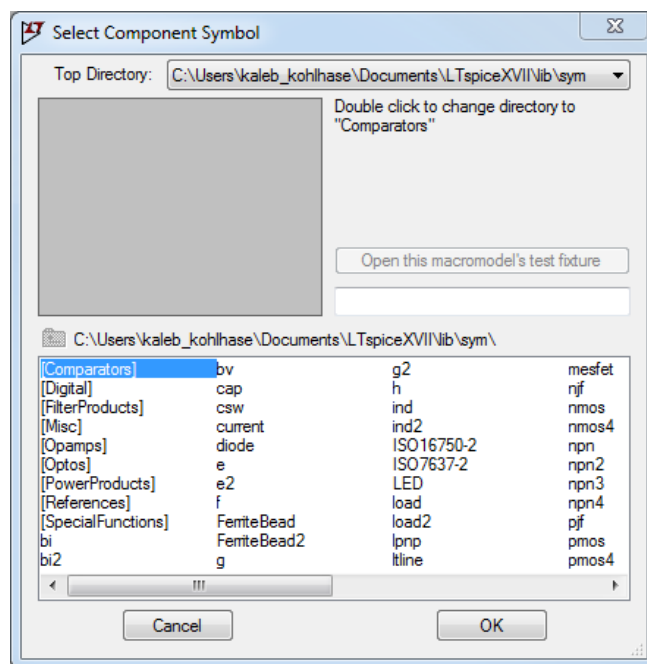


Any values in mind can be substituted for the following parts in the circuit above if desired. LTSpice requires a ground reference in all circuit schematics to function properly.

1. Place the voltage source, wires, ground, and passive components into the schematic.
  - a. The voltage source is in the list of pre-loaded library files located in the Components list among many others.

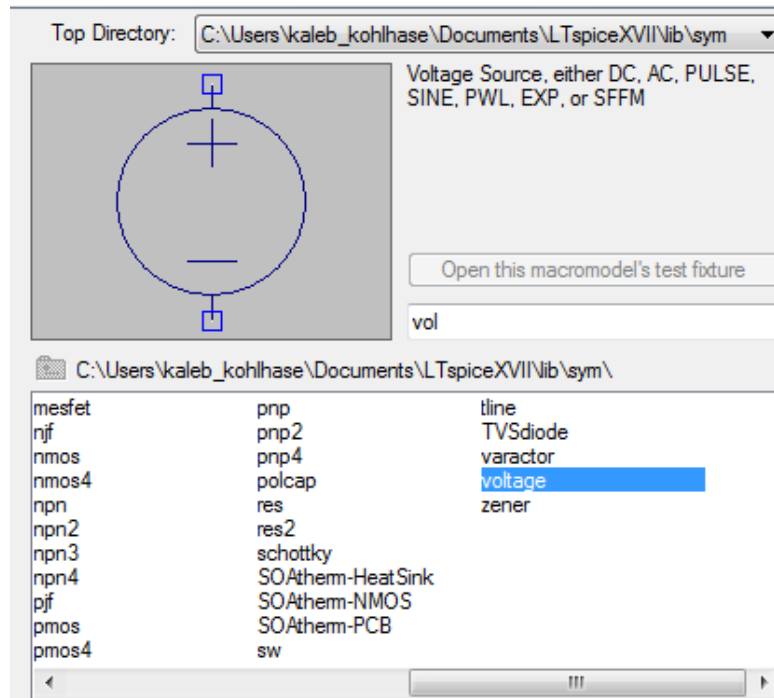


- b. After clicking the above symbol, the Library should show:



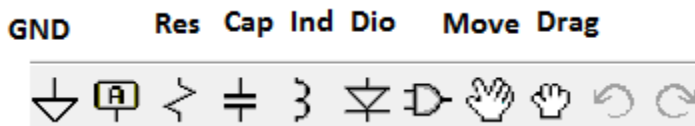


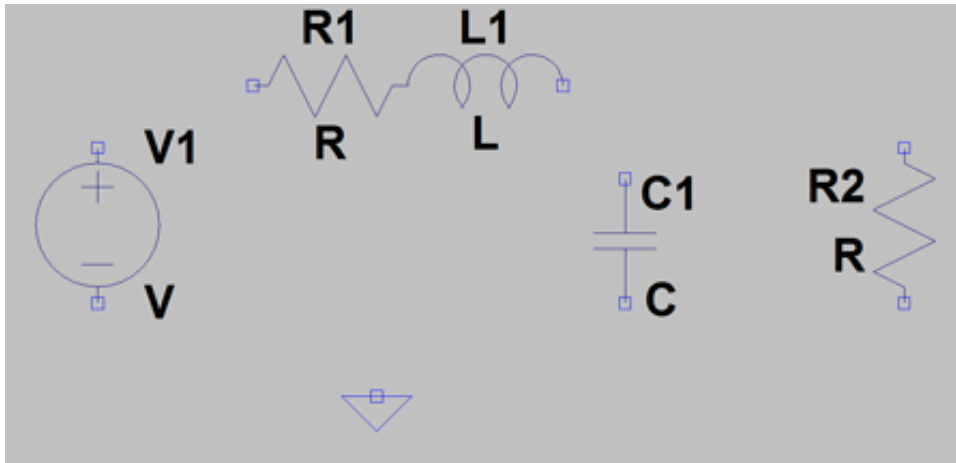
c. There is a search box under “Open this macro model’s test fixture” where a part can be searched for.



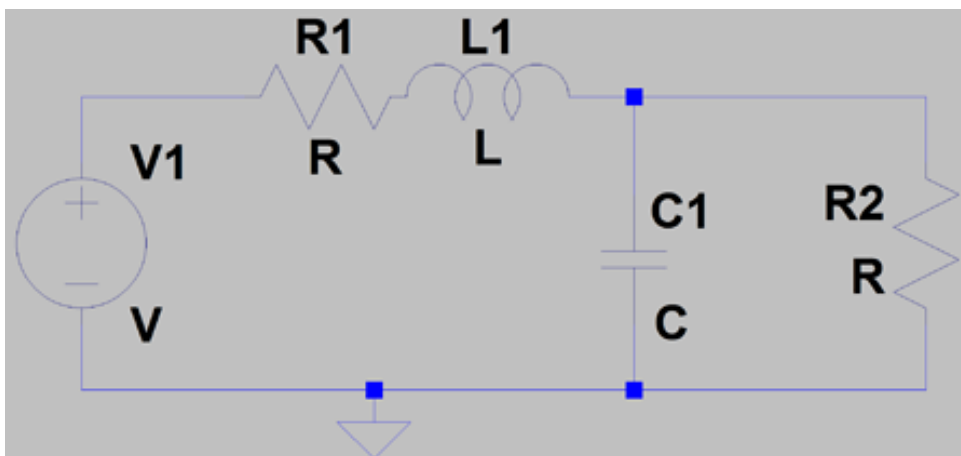
We usually type “vol” for a basic voltage source (AC, DC, and other options available for the model). Click OK and place one source anywhere on the document (Control + R to rotate component, Control + E to mirror component). After a source is placed, press Esc on the keyboard as the editor thinks multiple components are to be placed. This happens for all basic components and commands such as move or drag, so remember to press escape. We have been frustrated many times when the window starts moving really far from the part because I forgot to press escape. If the schematic is hard to find again, press ZoomToExtents:

d. Add all the other basic components by clicking the appropriate symbols.



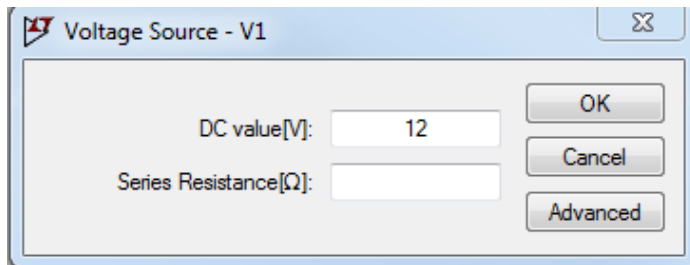


Parts can be connected together by placing the pins on top of each other in the grid to reduce wiring. Move around the parts using either move or drag (drag will create a lengthened wire instead of breaking connections). Wire the schematic using the wire tool which is to the left of the GND symbol. The tool will automatically end without pressing Esc when drawing node to node of each pin. Change the direction of the wire by clicking anywhere on the grid. Escape is only needed if a mistake is made. The undo function is to the right of the drag function (Ctrl + Z is assigned [changeable] to the Zoom command in this program). Nodes (blue solid squares) can be made if clicking the end of a wire directly on a different wire. If clicking past another wire and overlapping an existing wire, no nodes will be created. This is the same as “jumping” over an existing wire in other editors.

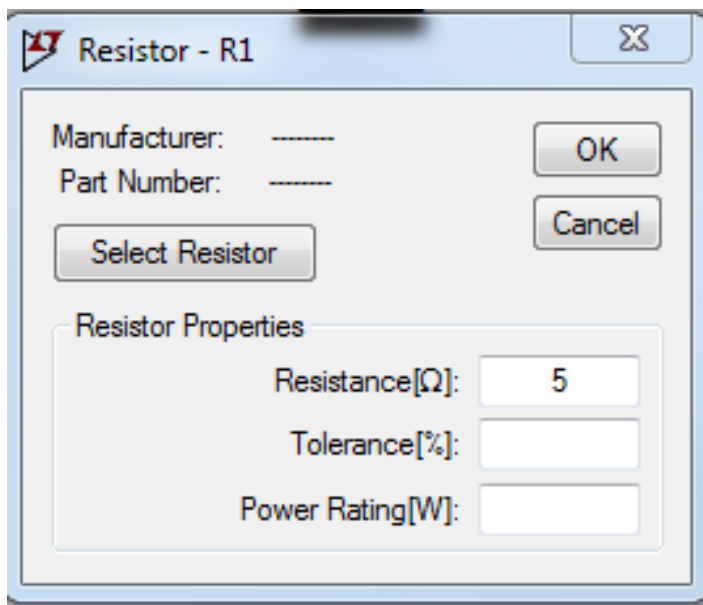


The program doesn't care how the parts are wired and length of wire doesn't apply in basic analysis. If there is continuity with no missing connections to the positive and/or negative side of the circuit, it will simulate in most cases. There are some instances where small series resistors may need to be added if the components don't have parasitic properties.

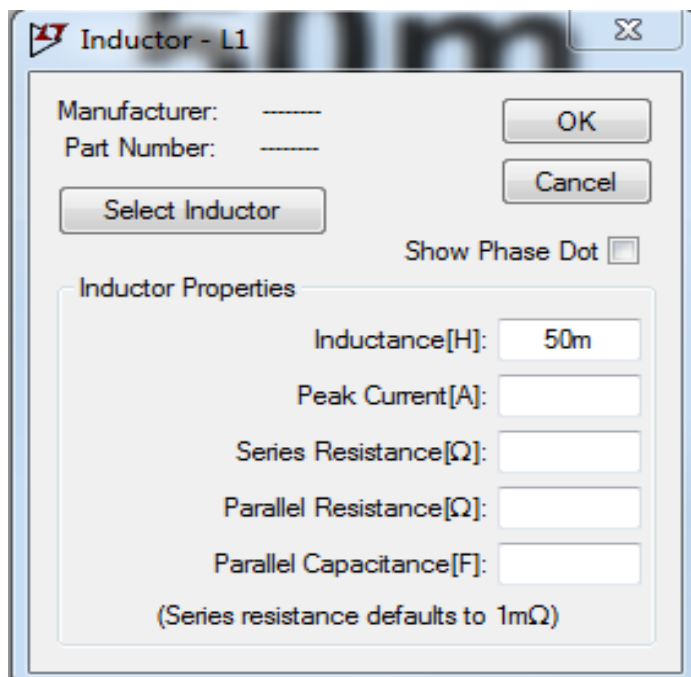
2. Edit the values of the components by right-clicking each one, there will be a custom dialogue box based on the type of component.



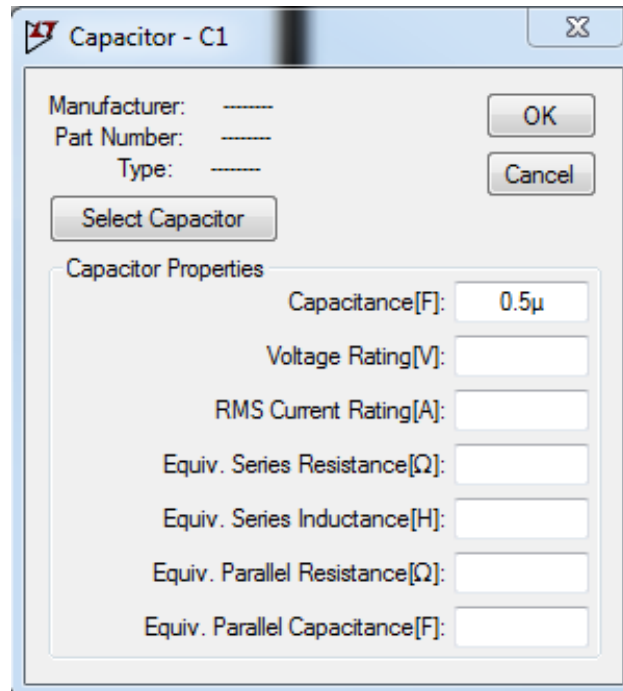
A dialog box titled "Voltage Source - V1" with a close button (X) in the top right corner. It contains two input fields: "DC value[V]" with the value "12" and "Series Resistance[Ω]" which is empty. To the right of these fields are three buttons: "OK", "Cancel", and "Advanced".



A dialog box titled "Resistor - R1" with a close button (X) in the top right corner. It contains two input fields: "Manufacturer: -----" and "Part Number: -----". Below these is a "Select Resistor" button. To the right are "OK" and "Cancel" buttons. A section titled "Resistor Properties" contains three input fields: "Resistance[Ω]" with the value "5", "Tolerance[%]" which is empty, and "Power Rating[W]" which is empty.



A dialog box titled "Inductor - L1" with a close button (X) in the top right corner. It contains two input fields: "Manufacturer: -----" and "Part Number: -----". Below these is a "Select Inductor" button. To the right are "OK" and "Cancel" buttons. A checkbox labeled "Show Phase Dot" is checked. A section titled "Inductor Properties" contains five input fields: "Inductance[H]" with the value "50m", "Peak Current[A]" which is empty, "Series Resistance[Ω]" which is empty, "Parallel Resistance[Ω]" which is empty, and "Parallel Capacitance[F]" which is empty. At the bottom of this section is the text "(Series resistance defaults to 1mΩ)".

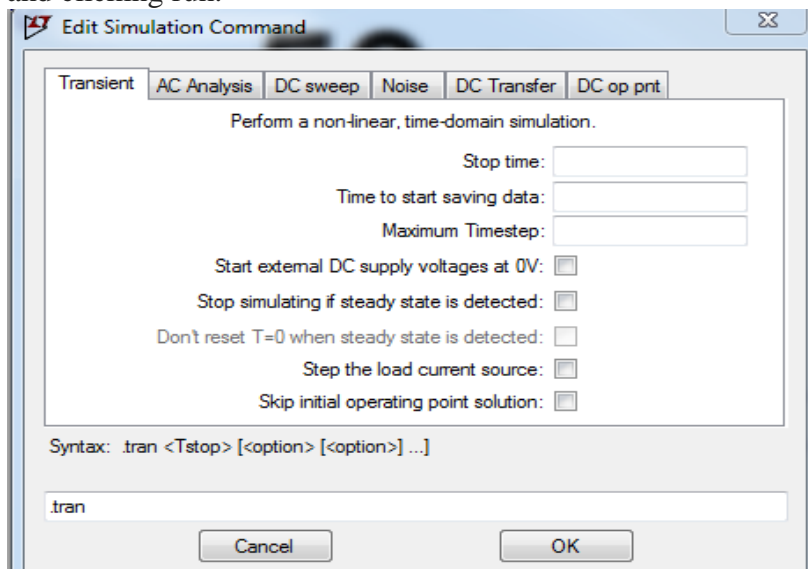


Most components can be edited (there are a few library items that can't be edited, but these are rare). The basic components need a basic primary value. The other values are optional for parasitic effects or more accurate models based on real-world equivalents.

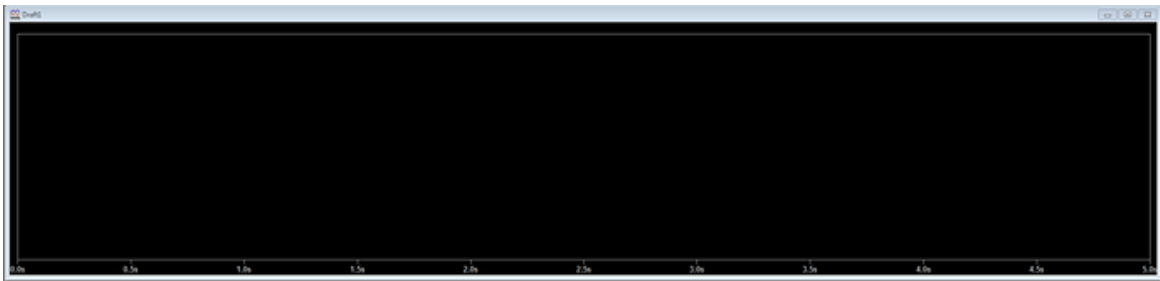
3. Run the simulation based on the type of analysis desired.



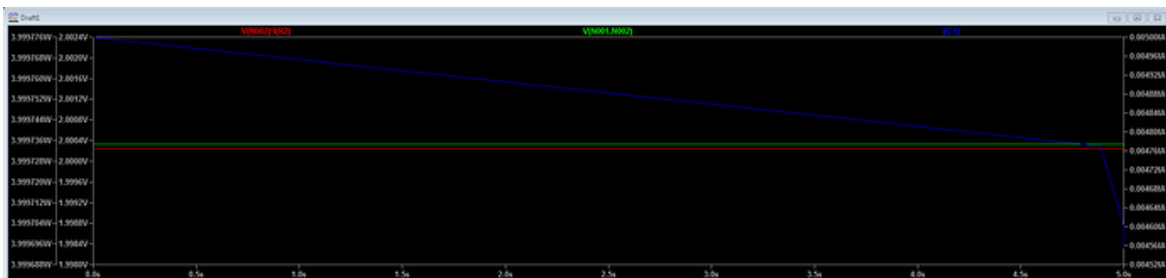
There is also a second way to choose a simulation by clicking the menu Simulate at the top and clicking run.



There are many options for simulation from transient analysis to finding the DC operating point. I tend to use the first three the most. In this example, I wanted to run a basic simulation for 5 seconds and save data as soon as it starts. This only requires one input of Stop time: 5 (everything starts off in base SI units). When “OK” is clicked, the Spice directive will show up on the schematic (if it is in the way, the grab/drag tool can move it). A black graph will appear above the circuit. This is the simulated O-Scope.



Voltage can be measured in two ways, clicking anywhere but ground reference will give a single node voltage output. The test lead can also be clicked and dragged to measure across several/single components to view voltage drop. The polarity will change based on the direction the mouse is dragged. Current can be tested by hovering over a component with a test lead and clicking or holding the Alt key on the keyboard over a section of wire. Resistors will often give a negative current which will be the exact current calculated in opposite polarity. It is recommended to measure current on the wires or components that give the expected current direction. Power is measured by holding the Alt key and clicking a component. Here is the waveform for the voltage across R1 and L1, current on the path for C1, and power consumed by R2:



Each new value measured will show in a new color (there is a limit and the colors will cycle), but there are unique names for the test points at the top of the graph:

The scales on the left and right of the graph will begin auto-adjusted to the most recent measurement. The units are also listed in the measurement scale. The scales can also be adjusted to Logarithmic by right-clicking the appropriate list of values. RMS can be calculated by pressing Control + Clicking the unique test point names. There are more advanced settings to do different calculations on the graphs, but that can be covered later.

This concludes this post by introducing the basics of the user interface and simulation with some personal tips to make using the program easier.

## CHAPTER 2

### WHY USE LTSPICE?

- Stable SPICE circuit simulation with
  - Unlimited number of nodes.
  - Schematics/symbol editor.
  - Waveform viewer.
  - Library of passive devices.
- Fast simulation of switch mode power supplies
  - Steady state detection.
  - Turn on transient.
  - Step response.
  - Efficiency / power consumption
- Advanced analysis and simulation options
  - Not covered in this lab class.
- Outperforms or as powerful as pay- for tools.

#### 2. Analysis setup

LTspice includes all the analyses available in most Spice-based simulation tools. For a better understanding of the capabilities of the Spice analyses,

Table 2 shows the most common applications for each analysis.

**Table 2.** Main applications for each type of Spice analysis

Analysis	Application
DC operation point	Determine the DC conditions of a biased transistor (i. e. the operation region). DC node voltages and loop currents of an electric network.
Transient	The time response of any circuit.
DC sweep	Transfer function of an amplifier, DC characteristic curves of a transistor.
AC	Frequency response (gain and phase) of a passive or active filter. Bandwidth of an amplifier.
Noise	Test the response of an audio amplifier under noise conditions.
DC transfer	Input and output resistance of an electric network. Output impedance of an amplifier. Output voltage of a network given for a determined input value.

A short description of each Spice analysis and its corresponding Spice syntax is presented next.

## 2.1 DC operation point

The `.OP` command replaces all capacitors with an open circuit and all the inductors with a short circuit and calculates the DC solution for the circuit. The results are displayed in a dialog box that pops up after the simulation is complete. Figure 2.1 shows the analysis setup where there is no need to enter any argument. AC sources are disconnected.

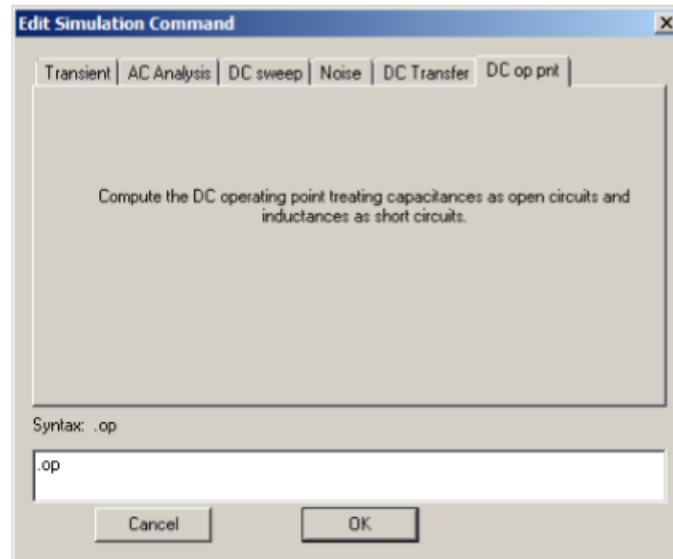


Fig. 2.1

## 2.2. Transient analysis

A non-linear, time domain analysis is performed, so results are displayed in such a way that the independent axis is timed in seconds, and the dependent axis can be any of the electrical variables interpreted by Spice or a mathematical expression of them.

The syntax of the transient analysis is as follows:

`.TRAN <Tstep><Tstop> [Tstart [dTmax]] [modifiers]`

where `<Tstep>` is the plotting increment of the waveforms and `<Tstop>` is the duration of the simulation. You can specify `Tstart` if you wish to start the simulation at a time different from zero.



Figure 2.2 shows the transient analysis setup window, where you have the option of writing the Spice directive to set such analysis at the bottom of the window.

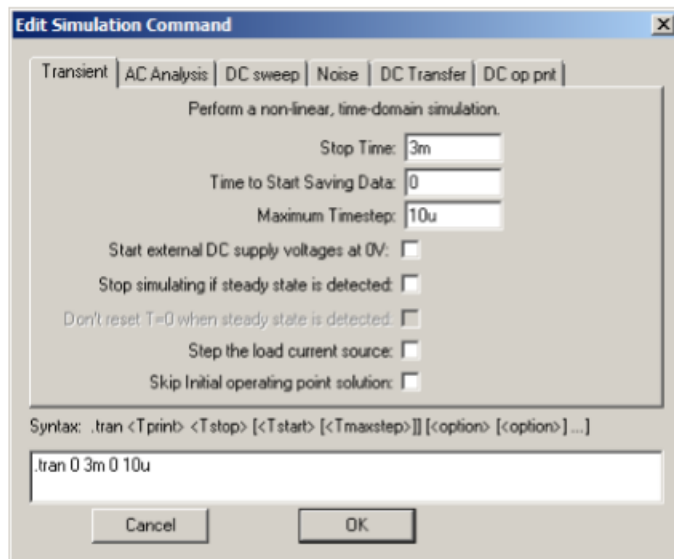


Fig.2.2

## 2.3 DC sweep

With this analysis, a DC source is swept in a determined voltage range, so the response of a variable as function of such voltage sweep can be plotted. This means that we can find the transfer function from the input source to some output variable.

The syntax of the DC sweep analysis is as follows:

```
.dc <srcnam><Vstart><Vstop><Vincr> + [<srcnam2><Vstart2><Vstop2><Vincr2>]
```

where<srcnam> is the name of the DC source to be swept starting from <Vstart> and stoping at <Vstop> by increments of <Vincr>.

Figure 2.3 shows the setup window of this type of analysis. As you can see, you can sweep up to three sources, which is useful when plotting parametric curves.

Figure 2.3 DC sweep analysis' setup

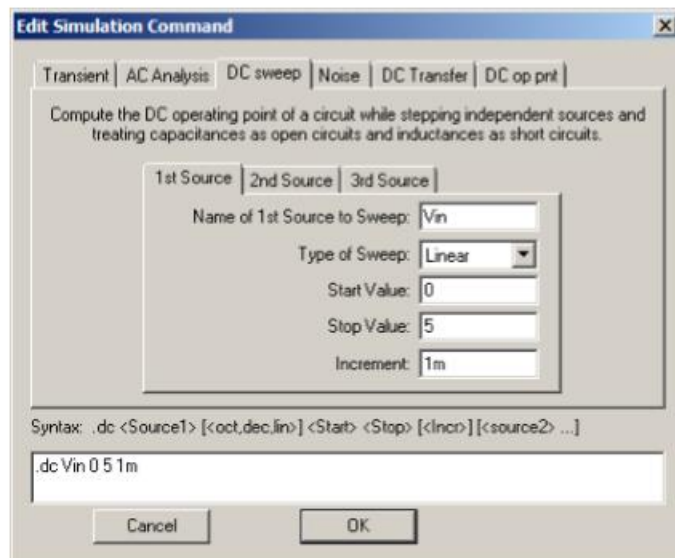


Fig. 2.3

## 2.4 DC transfer function

With this analysis you are able to find the small-signal DC transfer function of a voltage node or loop current, due to small variations of an independent source. Results shown includes the input voltage and resistance and output voltage or current and output resistance.

The syntax of the transfer function analysis is as follows:

`.TF V(<node>[, <ref>]) <source> .TF I(<voltage source>) <source>`

where `V(<node>[, <ref>])` and `I(<voltage source>)` are the voltage node and current through the source, respectively, and those values will be displayed as a function of the voltage source `<source>`.

Figure 2.4 shows the setup of a DC transfer function analysis.

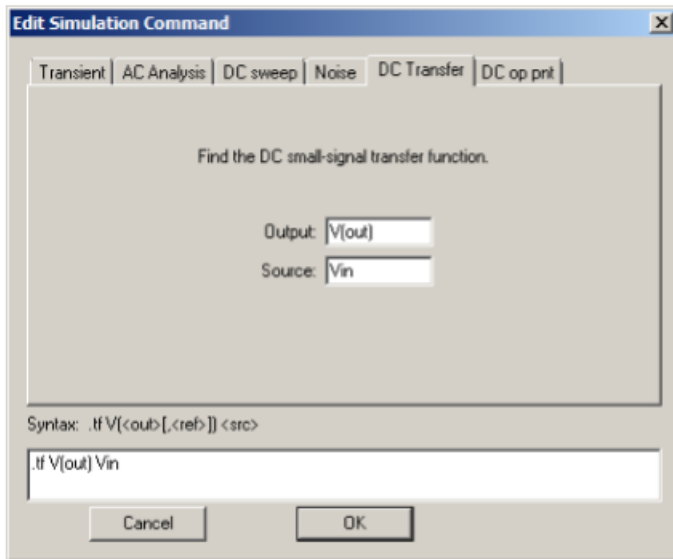


Fig. 2.4

## 2.5 AC analysis

This analysis computes the small-signal AC response of a circuit as a function of frequency. This analysis finds the DC operation point of the circuit first, so you can use DC sources to bias your circuit to find the small signal response under a DC bias condition.

The syntax of the AC analysis is as follows:

```
.ac <oct, dec, lin><Nsteps><StartFreq><EndFreq>
```

You can plot your results using octal, decade or linear frequency ranges using the keywords oct, dec or lin, respectively. With <Nsteps> you define the number of steps for each octave or decade of the range from <StartFreq> to <EndFreq>.

Figure 2.5 shows the setup of an AC analysis.

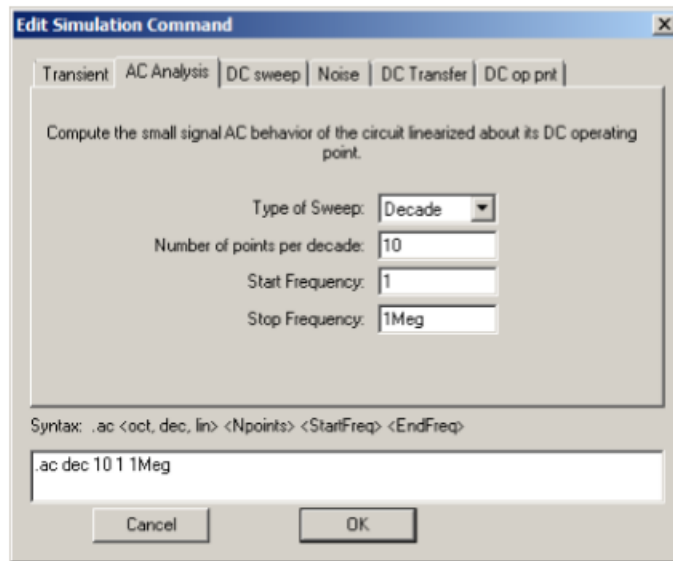


Figure 2.5

## 2.6 Noise analysis

This analysis performs a frequency domain analysis to compute Johnson, shot and flicker noise types. The output is noise spectral density per unit square root bandwidth.

The syntax of the noise analysis is

`.noise V(<out>[,<ref>]) <src><oct, dec, lin> + <Nsteps><StartFreq><EndFreq>`

`V(<out>[,<ref>])` is the node at which the output noise is calculated. `<src>` is the name of an independent source to which input noise is referred. `<src>` is the noiseless input signal. The parameters `<oct, dec, lin>`, `<Nsteps>`, `<StartFreq>`, and `<EndFreq>` define the frequency range of interest and resolution in the manner used in the `.ac` directive.

Figure 2.6 shows the setup of a noise analysis.

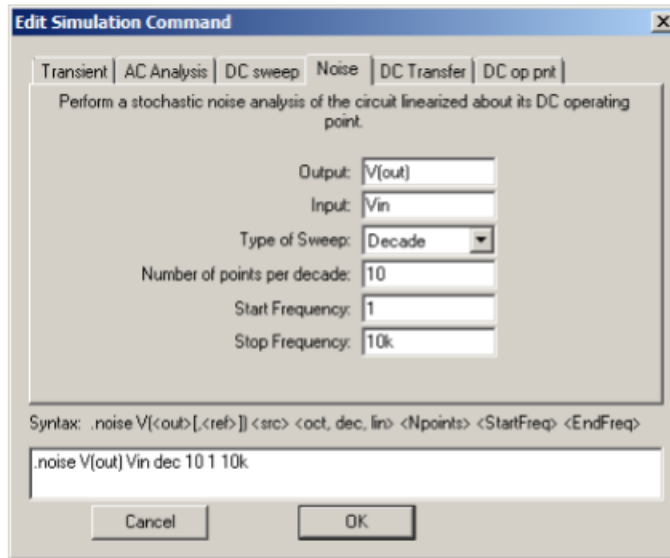


Figure 2.6

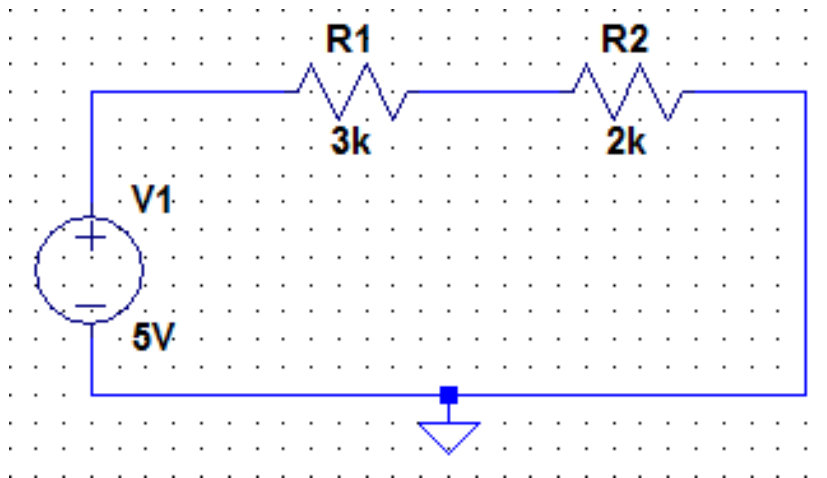
## 2.7 Parametric analysis

With a parametric analysis you are able to sweep the value of a component while performing any type of analysis available on LTspice IV such as transient, DC bias point, AC analysis, DC sweep. Basically, a parametric analysis is a multi-run process where you set a main analysis and specify a series of values to be swept for a component. When you run the analysis, LTspice IV sets the first value of the parametric variable and performs the simulation. When finished, the next value is set automatically on the circuit and the simulations run again. This process is repeated until the list of values for the component that you selected is completed and the results are plotted.

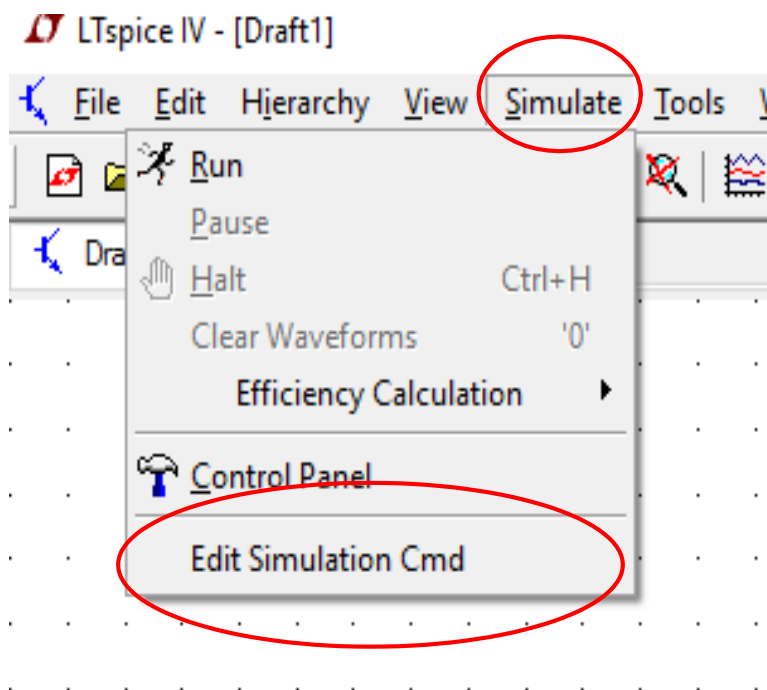
## CHAPTER 3

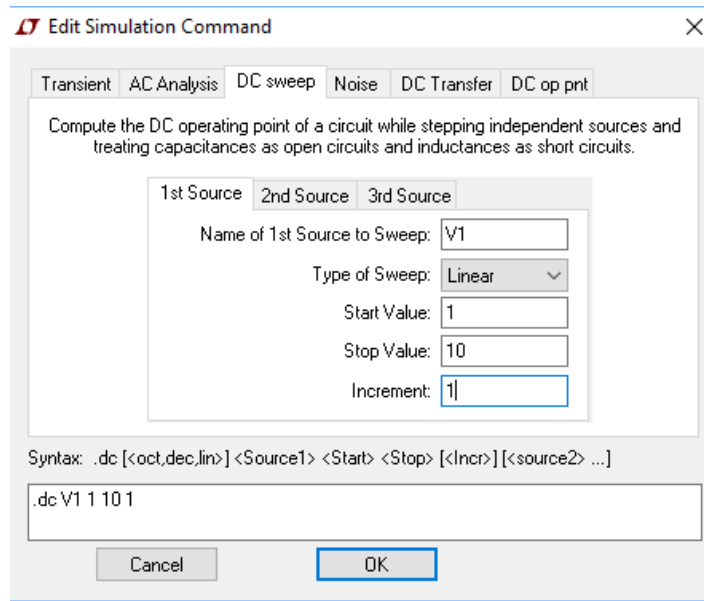
### EXPERIMENTS EXAMPLE:

Build the following circuit in LTspice and simulate it:



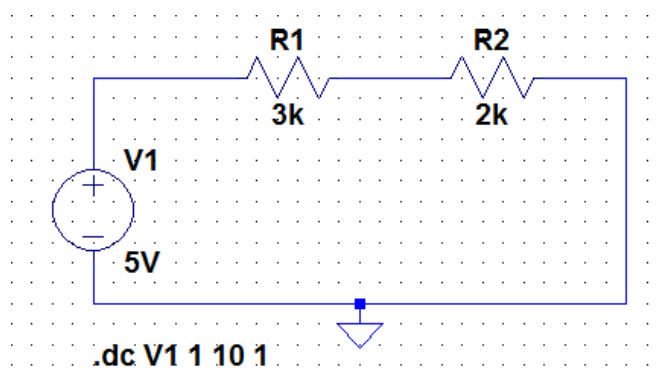
- DC Sweep is used when we want to vary the value of one of the sources or resistances in the circuit
  - Use DC sweep to figure out the response of the circuit over a range of values.
- In the dialog box click the DC Sweep tab and set the following parameters and click OK.

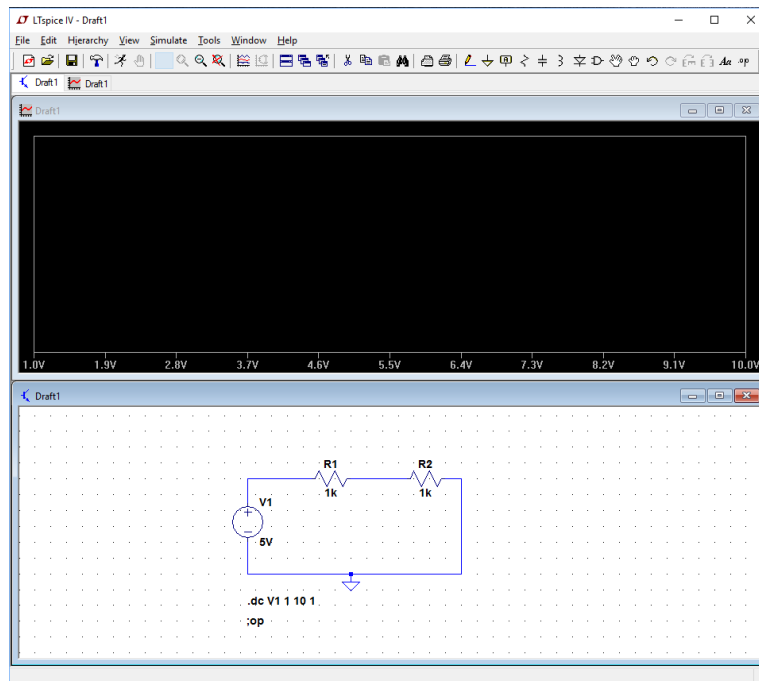




- Sweep type is linear. We will try other options later.
- For the Name text box, enter the name of your voltage source in the schematic. **NOT** the type of source it is (e.g. voltage) but what we call it: “V1,” or “VSource1”. Enter the desired start, end and increment values. We choose, 1, 10 and 1 respectively. Note that you do NOT have to enter the units
- Running a DC Sweep implies that the static values supplied on the schematic for the voltage supply does not matter.

## Simulate

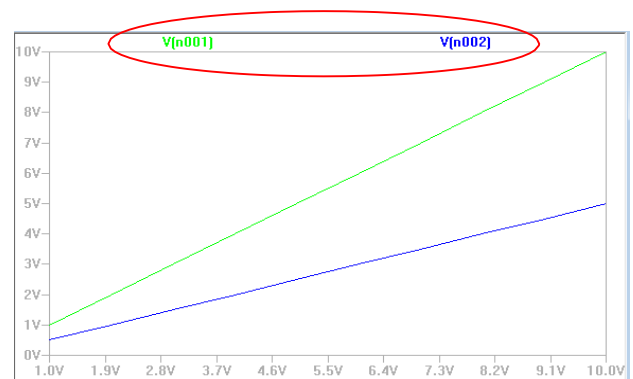
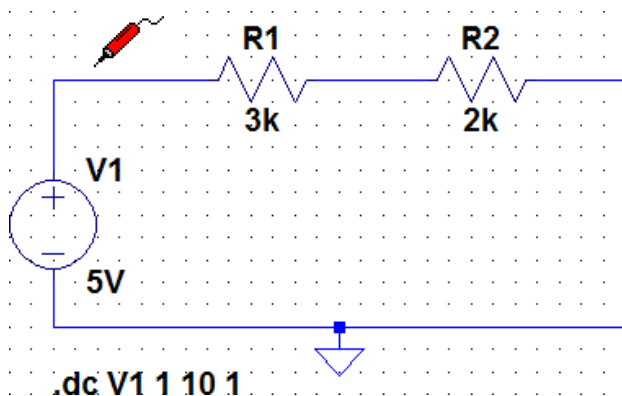




Once you have set the DC Sweep, simulate the circuit. The Probe window will pop up with a blank blackgraph

### Adding a Trace

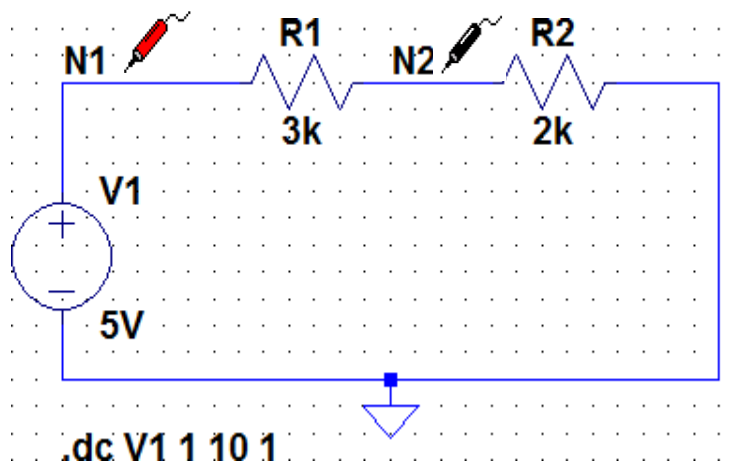
- Left-click on a wire to plot the voltage for that node. This will measure the voltage with respect to ground.

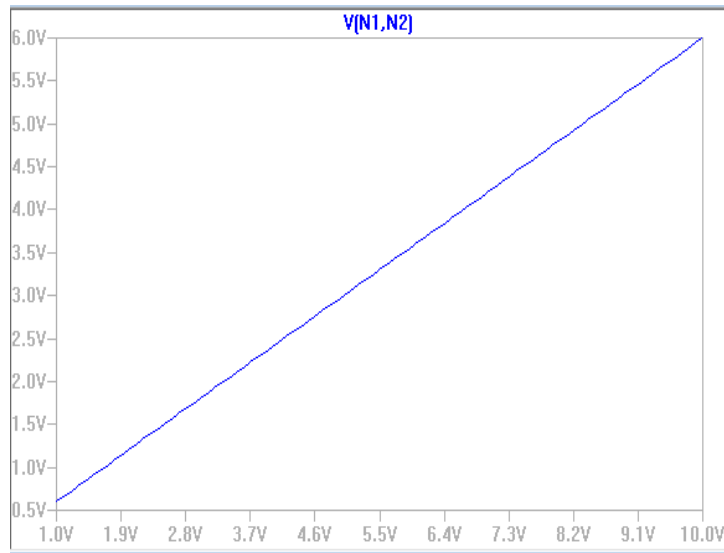




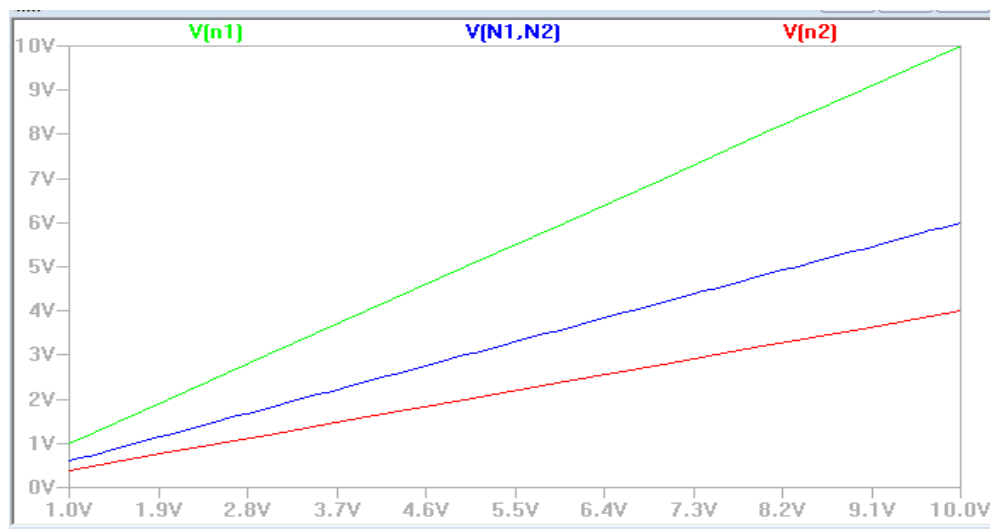
- We will plot the voltage drops across all three components using node voltages. Here you can see the two voltages are V(n001) and V(n002).
- For more complicated circuits, it can be a problem to recognize the nodes without labels.
- It is important to label nodes when adding traces for the homework problems.
- Go back to the schematic, rename the wires N1 and N2 and simulate your schematic again.
- Plot the three voltage drops across the components. For example, the voltage difference across nodes N1 and N2 will provide the voltage across R1.

**Voltage difference across nodes:** Left-click and hold on the node and drag the probe to another node





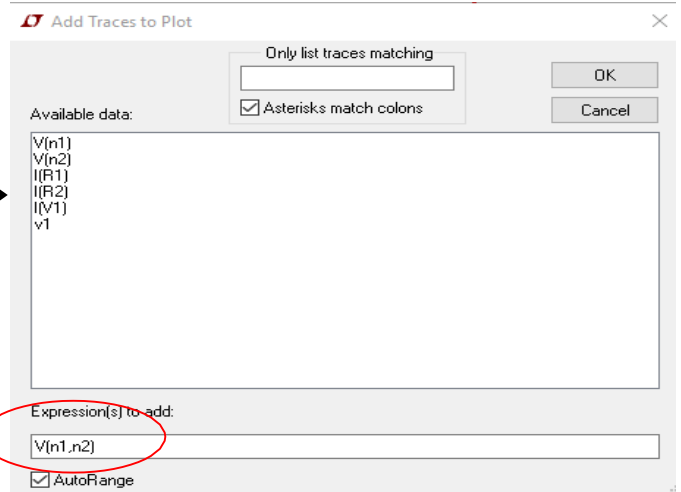
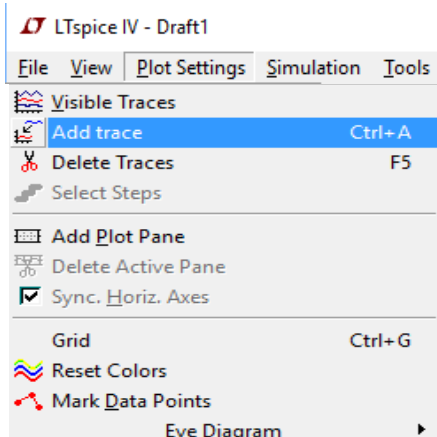
- You should see the following plot.




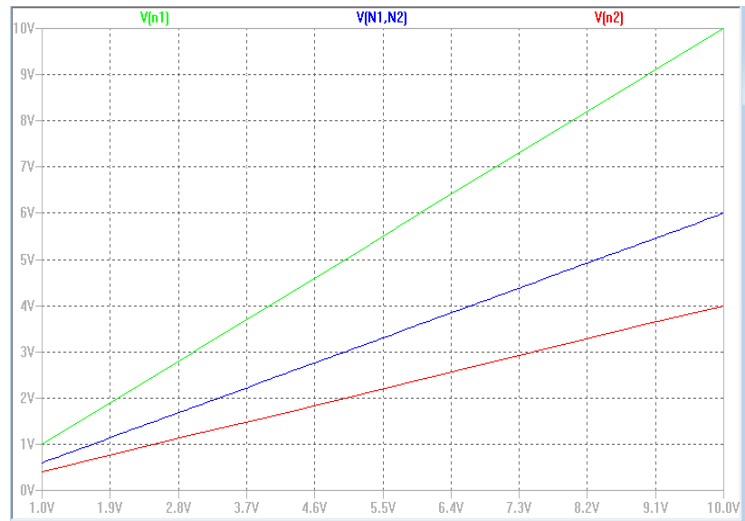
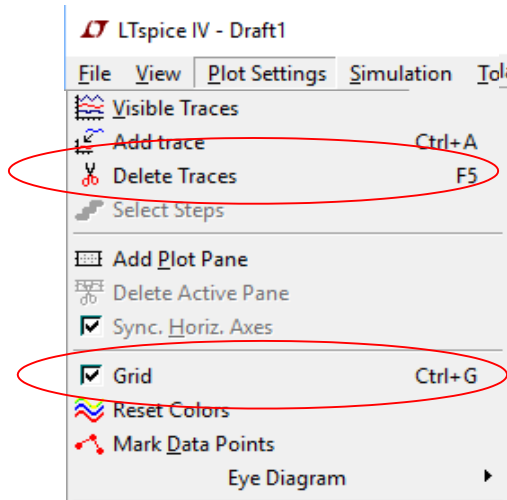
Note that the variable for the x-axis has automatically been chosen to be the voltage source. This is because it is the swept variable.

In the *Tools* drop down menu, choose *Copy bitmap to Clipboard* to paste the graph in word processing applications.

- To add a plot, after selecting the plot window, click on the *Plot Settings* menu and select *Add trace* or you can press Ctrl+A.



- To modify a specific trace, right-click the legend, and change it.
- To turn the grid on, under *Plot Settings* click on *Grid* (Ctrl+G).
- To delete a trace, under *Plot Settings* click *Delete Traces* (F5) or click  and delete the legend.

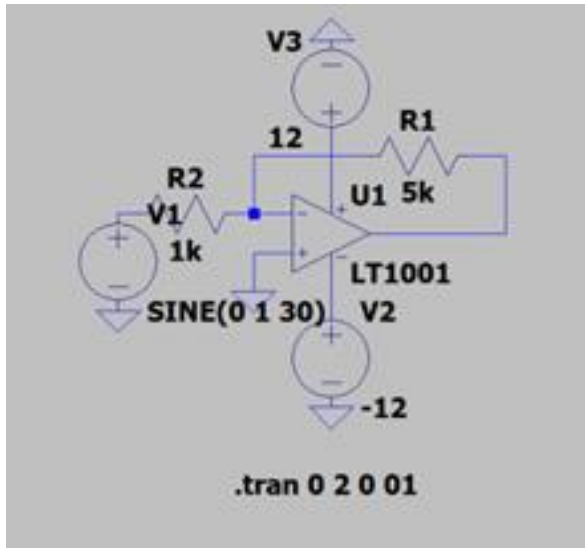


- Note that the voltage dropped across the resistance R2 is exactly half of the voltage supplied by the source at any point on the trace

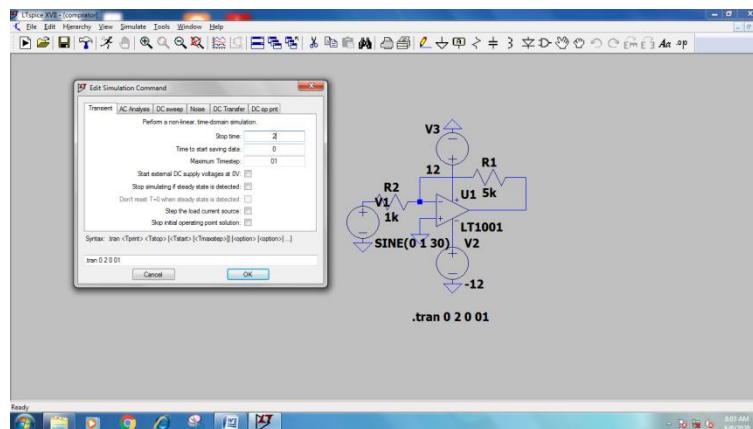
## EXPERIMENT 1

OBJECTIVE:-Write a program in LT spice for input and output characteristics of inverting amplifier.

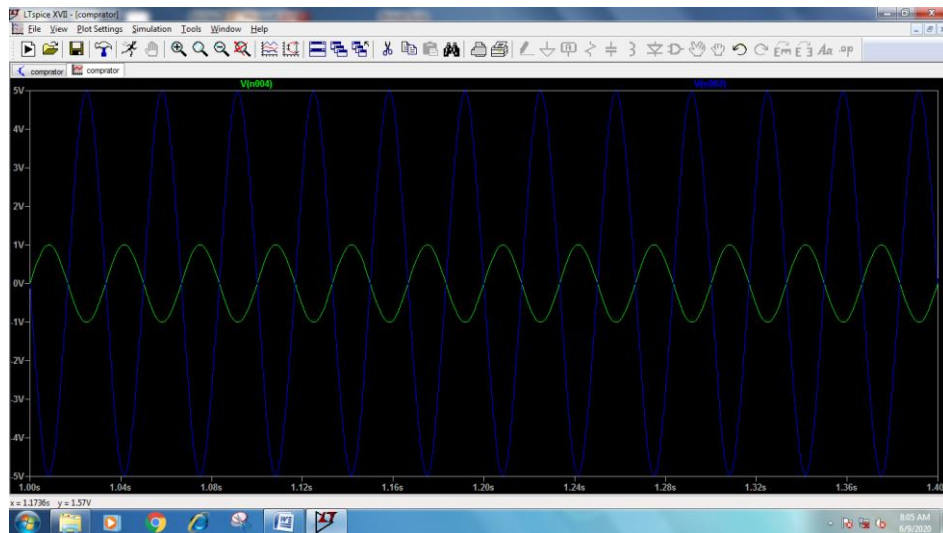
CIRCUIT DIAGRAM:



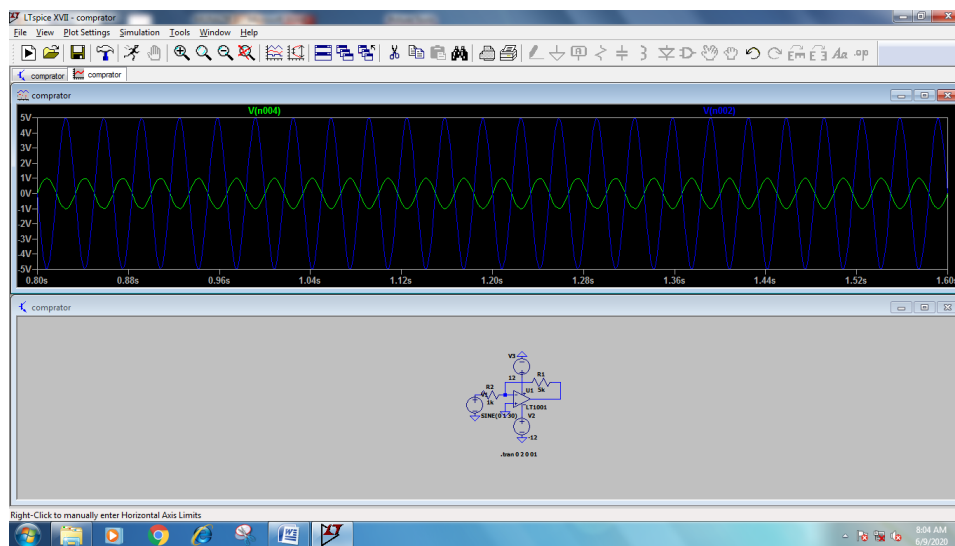
Simulation command:



OUTPUT:



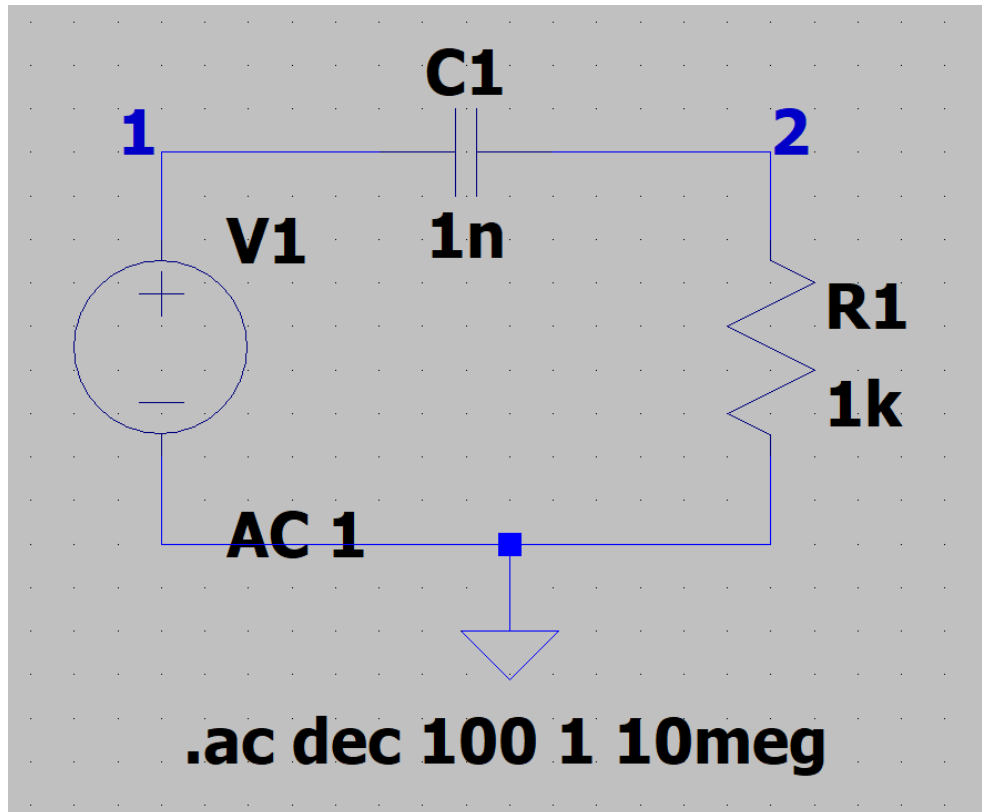
Where Blue colour trace shows the output waveform and green colour trace shows input waveform.



## EXPERIMENT 2

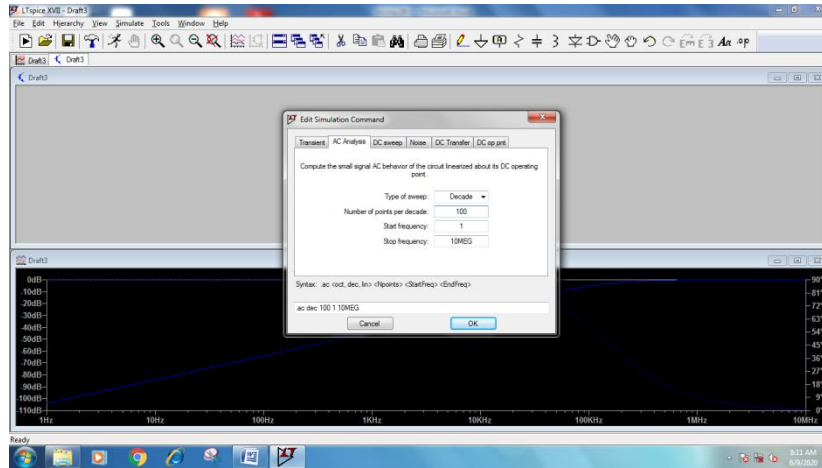
OBJECTIVE:- Write a program in LT spice for output characteristics of High pass filter.

CIRCUIT DUAGRAM:



Simulation command:

AC simulation: It computes the small-signal AC response of a circuit as a function of frequency. Syntax of ac simulation is discussed above.



**Code:**

\* R-C HIGH PASS FILTER

V1 1 0 ac 1v

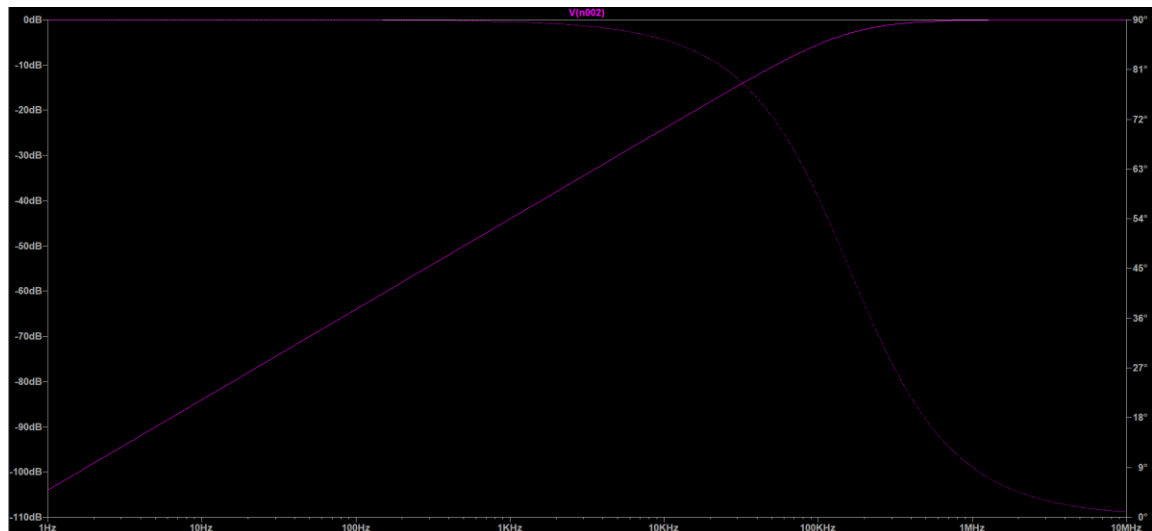
C1 2 1 1n

R1 2 0 1k

.ac dec 100 1 10meg

.end

**OUTPUT:**



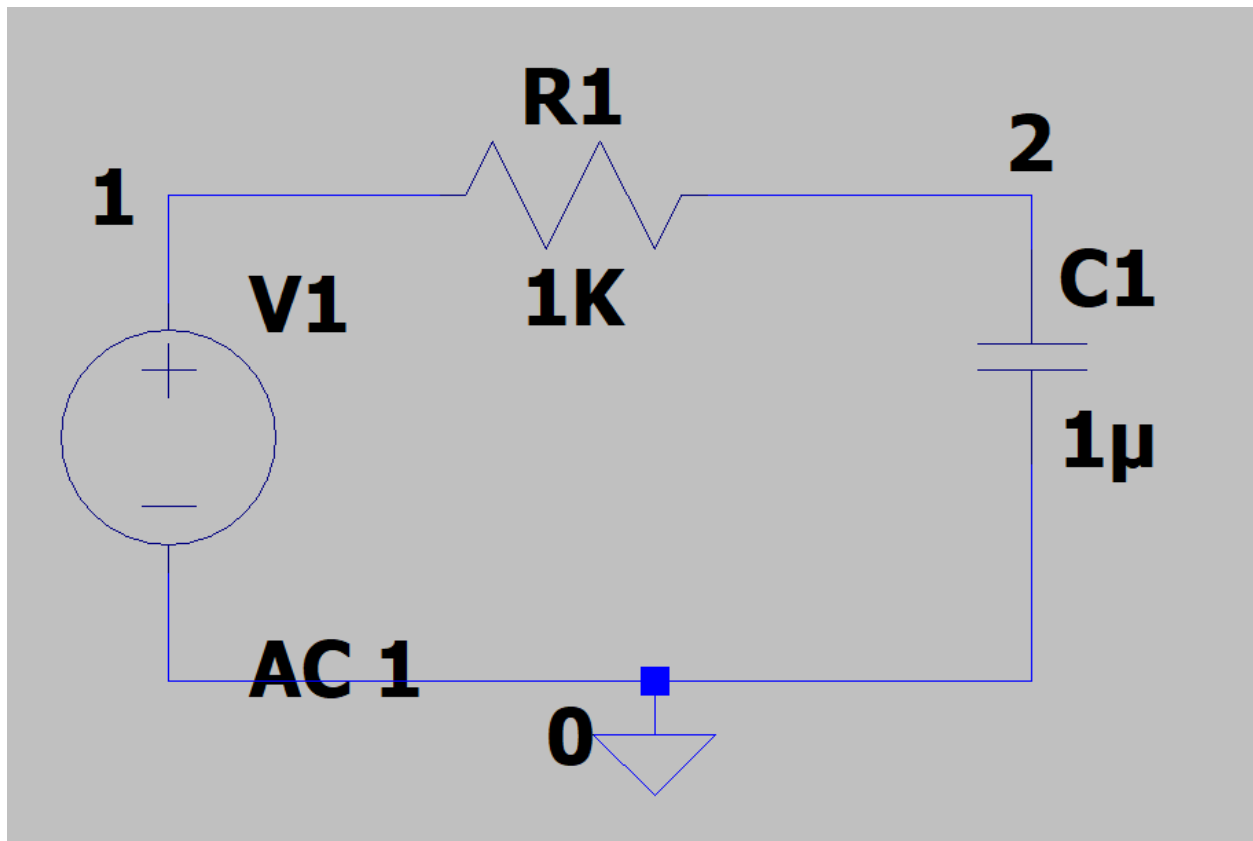
A High pass filter (HPF) is an electronic filter that passes signal with a frequency higher than a

certain cut-off frequency and attenuates signal with frequency lower than cut-off frequency.

### EXPERIMENT 3

**OBJECTIVE:-**Write a program in LT spice for output characteristics of R-C LOW PASS filter.

**CIRCUIT DIAGRAM:**



**Code:**

\* RC low pass filter simulation

V1 0 1 ac 1v

R1 1 2 1k

C1 2 0 1uF

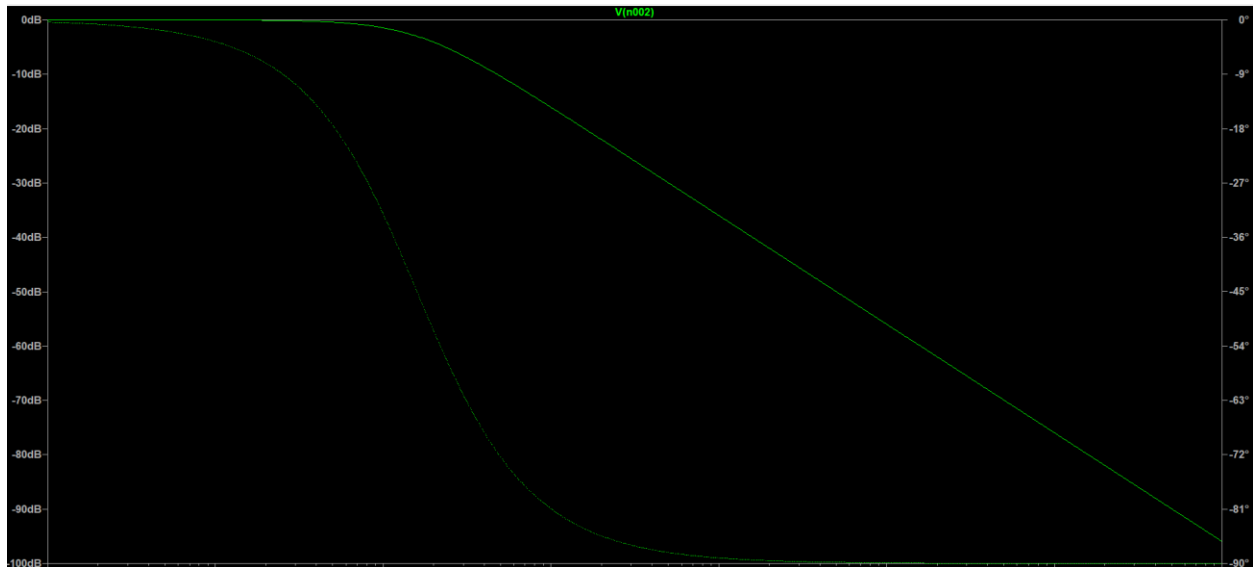


```
.ac dec 100 1hz 10Mhz
```

```
.probe
```

```
.end
```

OUTPUT:

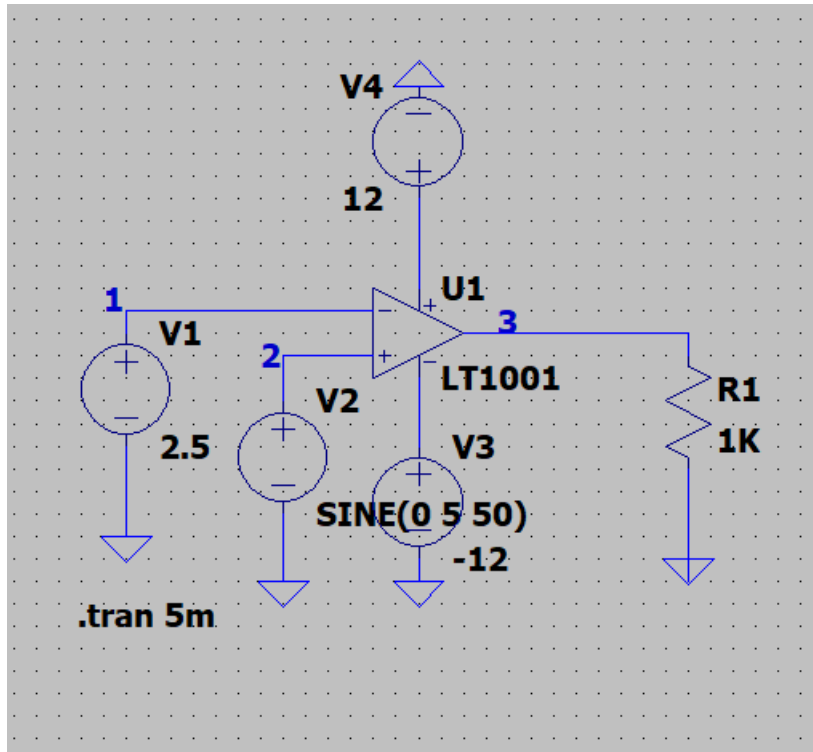


A low pass filter ( LPF ) is an electronics filter that passes signal with a frequency lower than a certain cut-off frequency and attenuates signal with frequency higher than the cut-off frequency. The amount of attenuation for each frequency depends on the filter design.

## EXPERIMENT 4

OBJECTIVE:-Write a program in LTspice for input and output **characteristics Non inverting op-amp comparator**.

CIRCUIT DIAGRAM:



**Code:**

\*non inverting comparator

```
v1 1 0 2.5v
```

```
v2 2 0 sin(0 5 50)
```

```
r1 3 0 1k
```

```
vddvdd 0 12
```

```
vssvss 0 -12
```

```
X1 2 1 vddvss 3 ua741
```

\* Model for uA741 Op Amp (from EVAL library in PSpice)

\* connections: non-inverting input

\* | inverting input

\* | | positive power supply

\* | | | negative power supply

\* | | | output

\* | | | |

\*

.subckt uA741 1 2 3 4 5

\*

c1 11 12 8.661E-12

c2 6 7 30.00E-12

dc 5 53 dy

de 54 5 dy

dlp 90 91 dx

dln 92 90 dx

dp 4 3 dx

egnd 99 0 poly(2),(3,0),(4,0) 0 .5 .5

fb 7 99 poly(5) vbvcvevlpvln 0 10.61E6 -1E3 1E3 10E6 -10E6

ga 6 0 11 12 188.5E-6

gcm 0 6 10 99 5.961E-9

iee 10 4 dc 15.16E-6

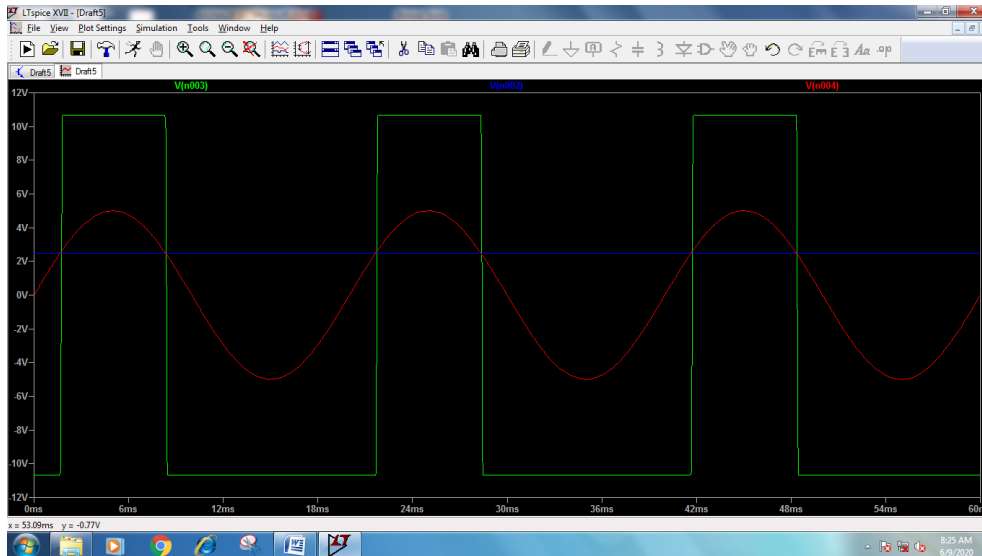
hlim 90 0 vlim 1K

q1 11 2 13 qx

q2 12 1 14 qx

```
r2 6 9 100.0E3
rc1 3 11 5.305E3
rc2 3 12 5.305E3
re1 13 10 1.836E3
re2 14 10 1.836E3
ree 10 99 13.19E6
ro1 8 5 50
ro2 7 99 100
rp 3 4 18.16E3
vb 9 0 dc 0
vc 3 53 dc 1
ve 54 4 dc 1
vlim 7 8 dc 0
vlp 91 0 dc 40
vln 0 92 dc 40
.model dx D(Is=800.0E-18 Rs=1)
.model dyD(Is=800.00E-18 Rs=1m Cjo=10p)
.model qxNPN(Is=800.0E-18 Bf=93.75)
.ends
.tran 5m
.end
```

OUTPUT:

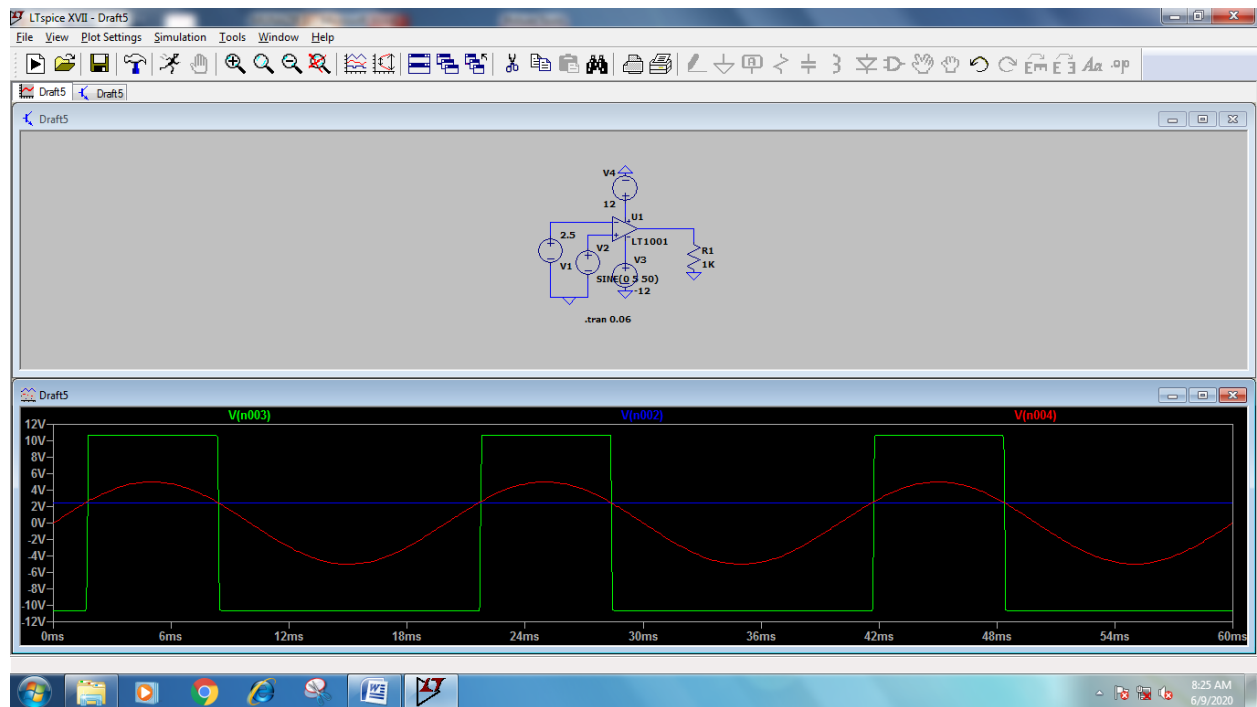


Here, the straight line is reference voltage which is applied at inverting terminal, sine wave is voltage at non-inverting terminal and square wave is output wave of comparator.

A comparator is a circuit with two input and single output. The two input can be compare with each other i.e one of them can be considered as reference.

When the voltage non-inverting terminal is greater than inverting terminal ( reference terminal ) voltage. The comparator output is HIGH.

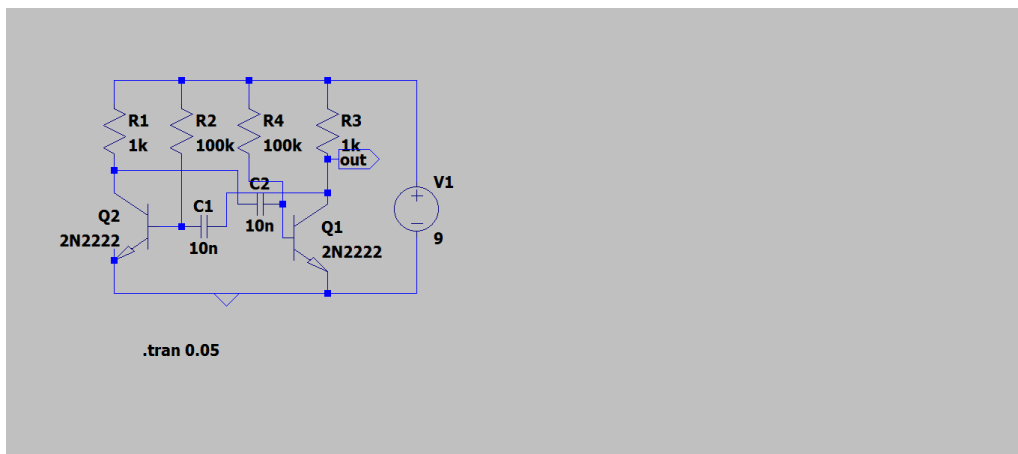
When the voltage non-inverting terminal is less than inverting terminal ( reference terminal ) voltage. The comparator output is LOW.



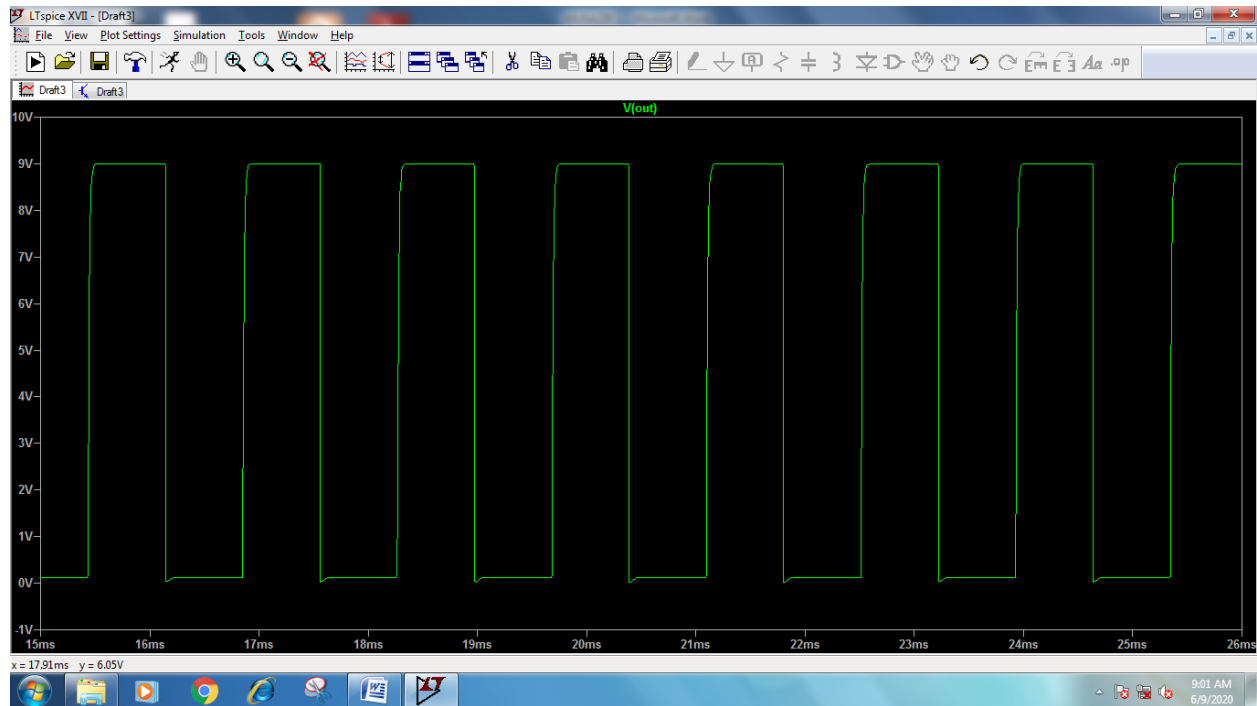
## EXPERIMENT 5

OBJECTIVE:-Write a program in LT spice for the characteristics of ASTABLE MULTIVIBRATOR.

CIRCUIT DIAGRAM



OUTPUT:



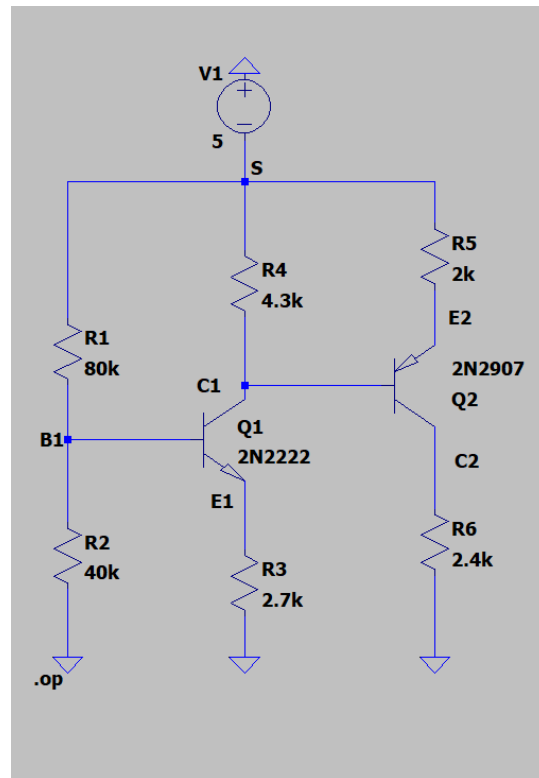
A multivibrator is an electronic device that generates square wave of its own ( i.e. without any external triggering pulse ) is known as an astable or free running multivibrator.

Astable multivibrator has no stable states. It switches back and forth from one states to the other, remaining in each states for the time determined by circuit constants.

## EXPERIMENT 6

**OBJECTIVE:-** Write a program in LT spice for Characteristics of BJT at DC operating points.

**CIRCUIT DIAGRAM:**



**CODE:**

\* A BJT AT DC

R1 S B1 80K

R2 B1 0 40K

R3 E1 0 2.7K

R4 S C1 4.3K

R5 S E2 2K

R6 C2 0 2.4K

v1 s 0 5



Q1 C1 B1 E1 qmodn

Q2 C2 C1 E2 qmodp

.model qmodnnpn

.model qmodppnp

.op

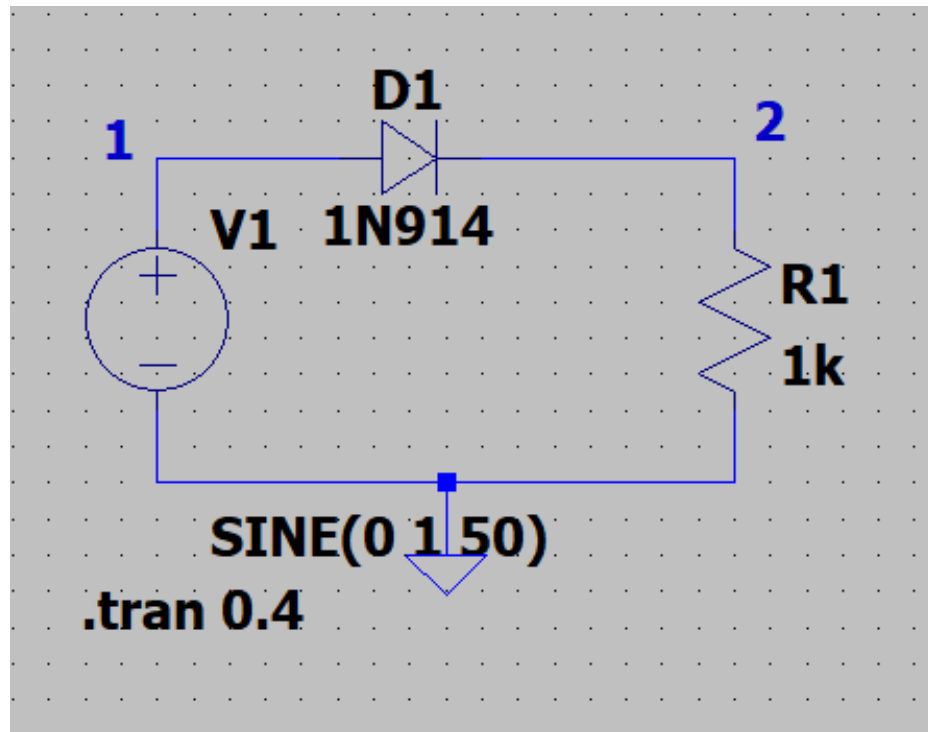
OUTPUT:

```
--- Operating Point ---  
V(s) :          5          voltage  
V(b1) :      1.58446      voltage  
V(e1) :      0.840665     voltage  
V(c1) :      3.68659     voltage  
V(e2) :      4.42818     voltage  
V(c2) :      0.679389     voltage  
Ic(Q2) :     -0.000283085 device_current  
Ib(Q2) :     -2.83085e-006 device_current  
Ie(Q2) :      0.000285916 device_current  
Ic(Q1) :      0.000308281 device_current  
Ib(Q1) :      3.08281e-006 device_current  
Ie(Q1) :     -0.000311364 device_current  
I(R6) :      0.000283079 device_current  
I(R5) :      0.00028591 device_current  
I(R4) :      0.000305444 device_current  
I(R3) :      0.000311357 device_current  
I(R2) :      3.96115e-005 device_current  
I(R1) :      4.26942e-005 device_current  
I(V1) :     -0.000634048 device_current
```

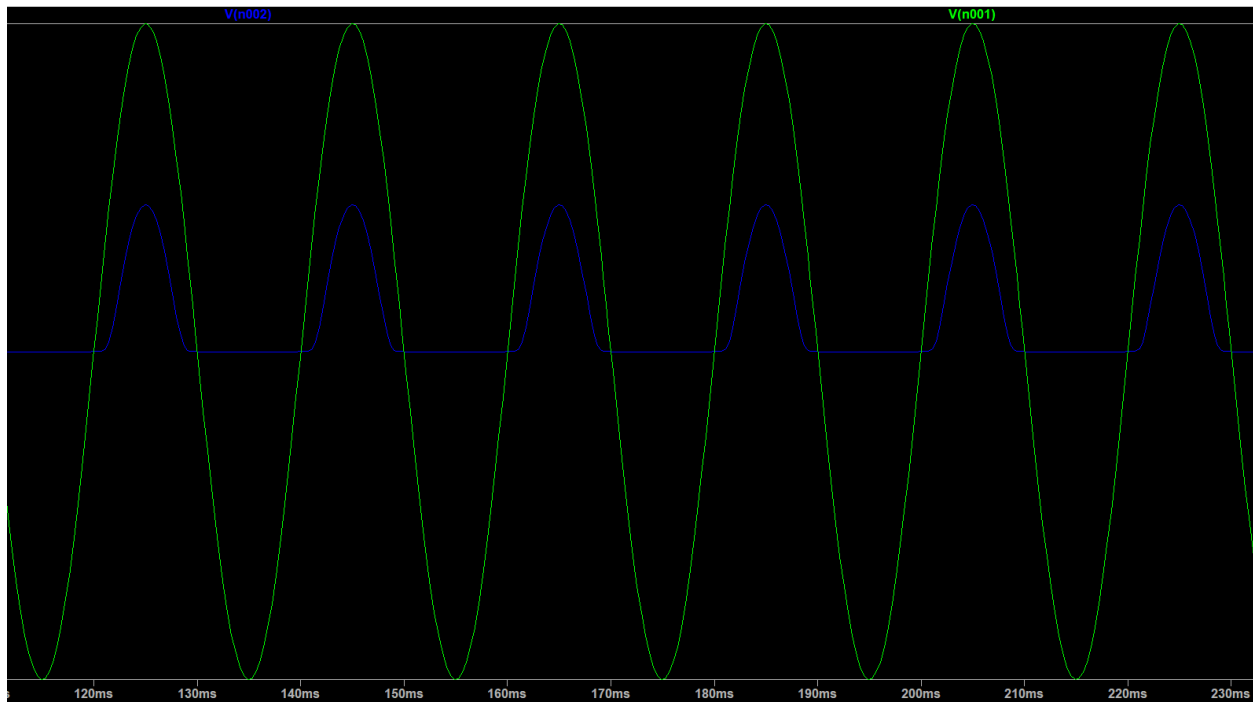
## EXPERIMENT 7

**OBJECTIVE:-** Write a program in LT spice for the input and output waveforms of half wave rectifier.

**CIRCUIT DIAGRAM:**



## OUTPUT

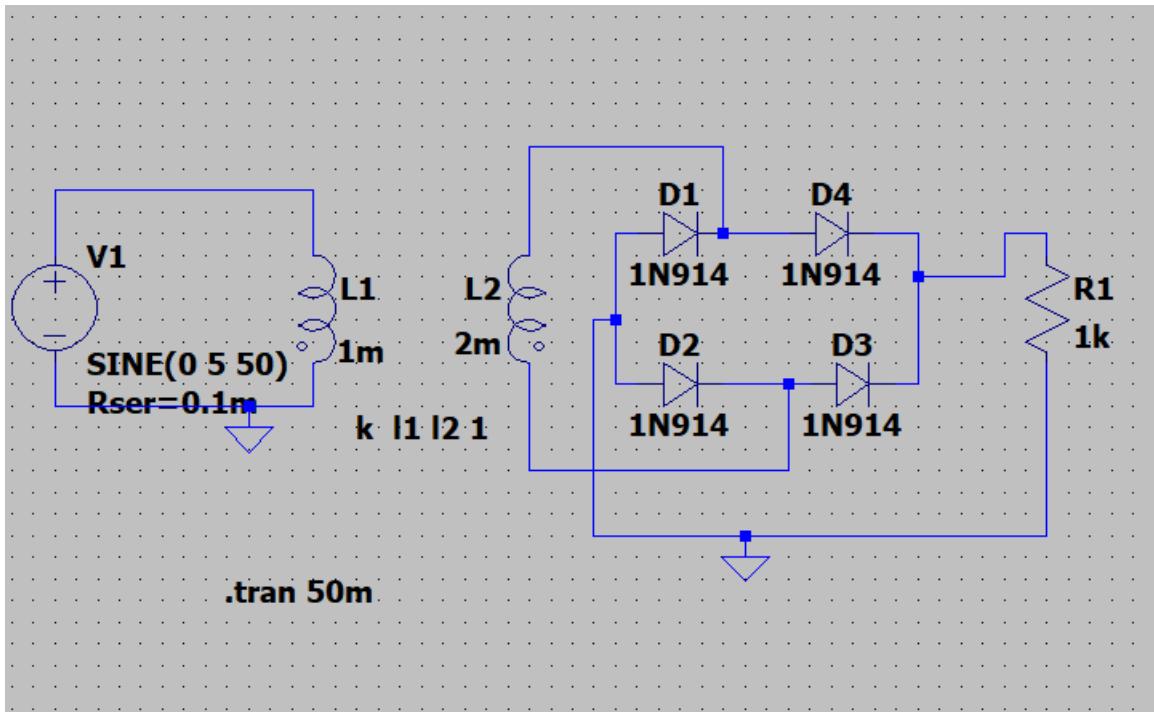


A Half wave rectifier is defined as a type of rectifier that allows one half cycle of an AC voltage waveform to pass, blocking the other half-cycle. Half wave rectifier used to convert AC to DC voltage, and require a single diode to construct.

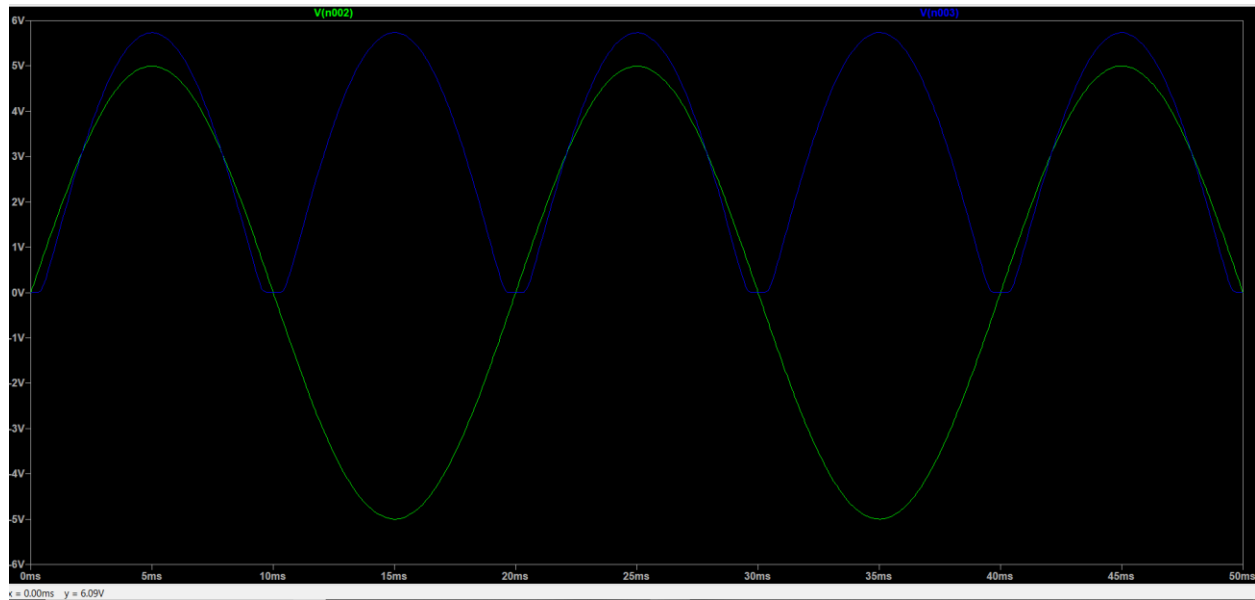
## EXPERIMENT 8

**OBJECTIVE :-** Write a program in LT spice for the output waveforms of full wave rectifier.

**CIRCUIT DIAGRAM:**



## OUTPUT

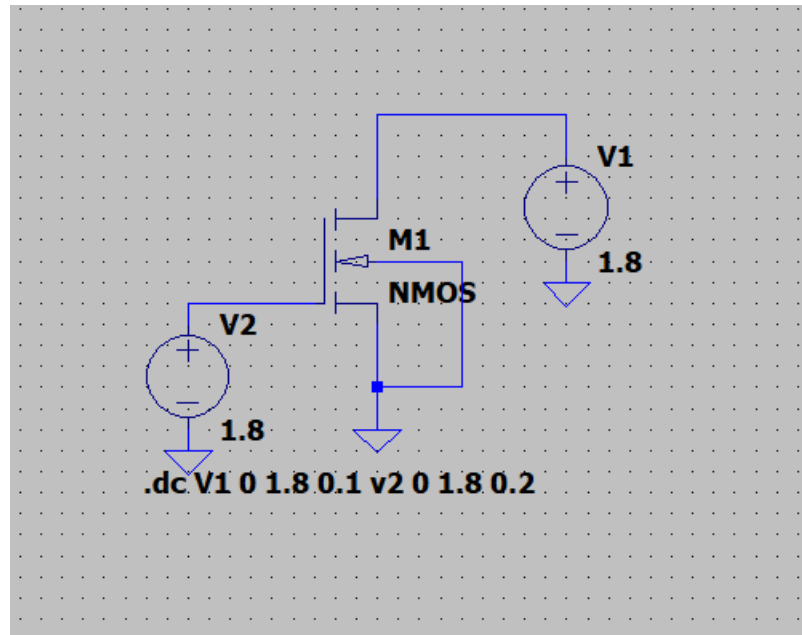


A full wave rectifier convert both halves of each cycle of an alternating wave ( AC signal ) into pulsating DC signal. In full wave bridge rectifier four diodes are used.

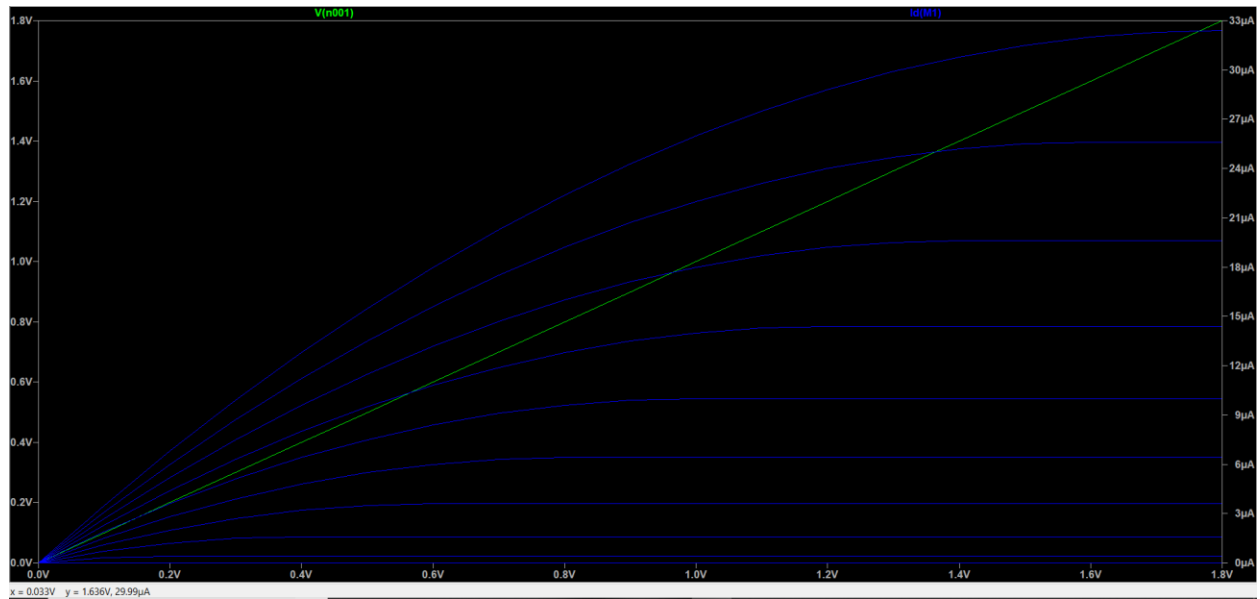
## EXPERIMENT 9

**Objective:** Write a program in LT spice for the output characteristics of NMOS Transistor.

**CIRCUIT DIAGRAM:**



**OUTPUT:**

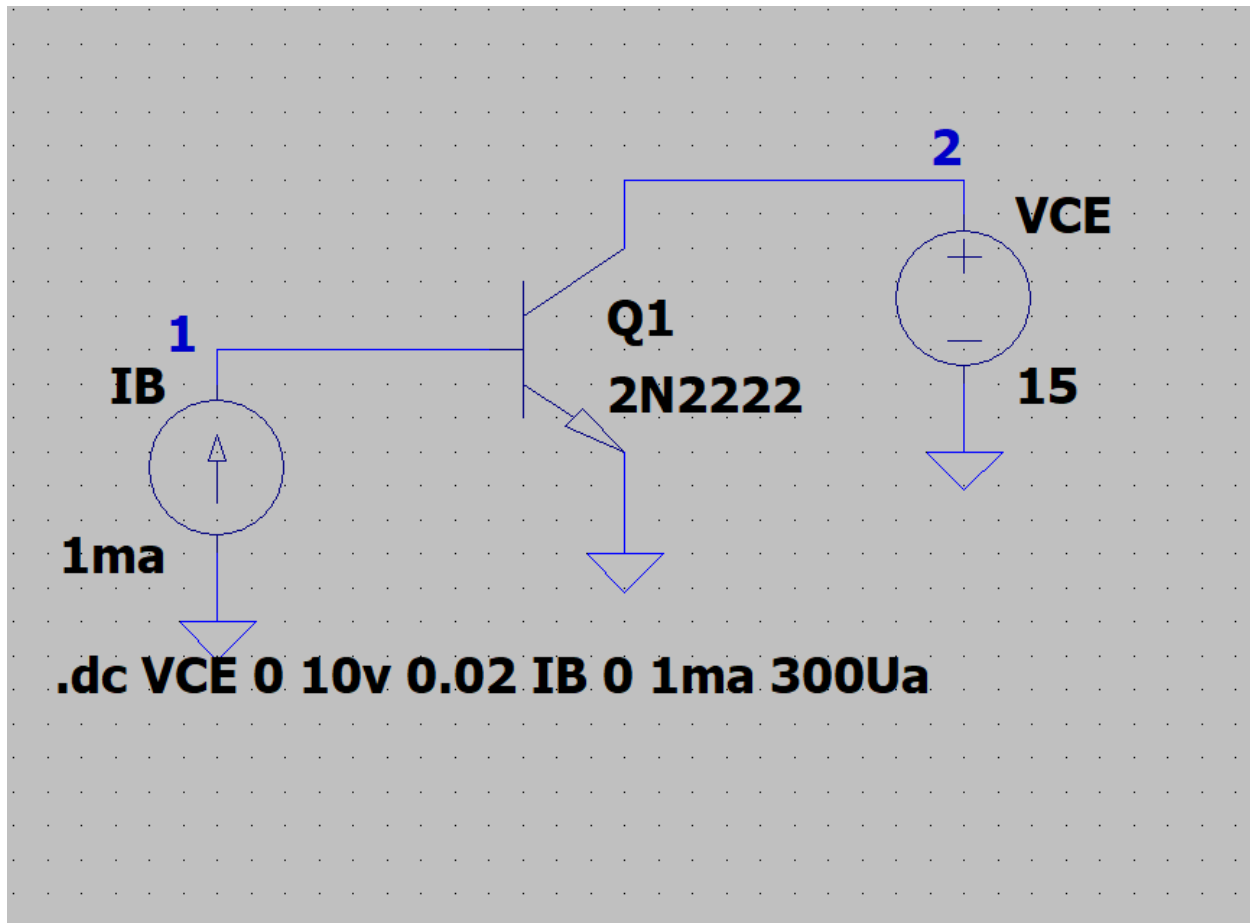


Output characteristics of NMOS Transistor is basically  $I_D$  versus  $V_D$

## EXPERIMENT 10

**Objective:** Write a program in LT spice for the output characteristics of BJT in common emitter mode.

CIRCUIT DIAGRAM:



**CODE:**

\*OUTPUT CHARACTERISTICS OF BJT.

IB 1 0 DC 1mA

VCE 2 0 15V

Q1 2 1 0 Q2N2222

```
.model Q2N2222 NPN (Is=14.34f Xti=3 Eg=1.11 Vaf=74.03 Bf=255.9 Ne=1.307  
+Ise=14.34f Ikf=.2847 Xtb=1.5 Br=6.092 Nc=2 Isc=0 Ikr=0 Rc=1
```

```

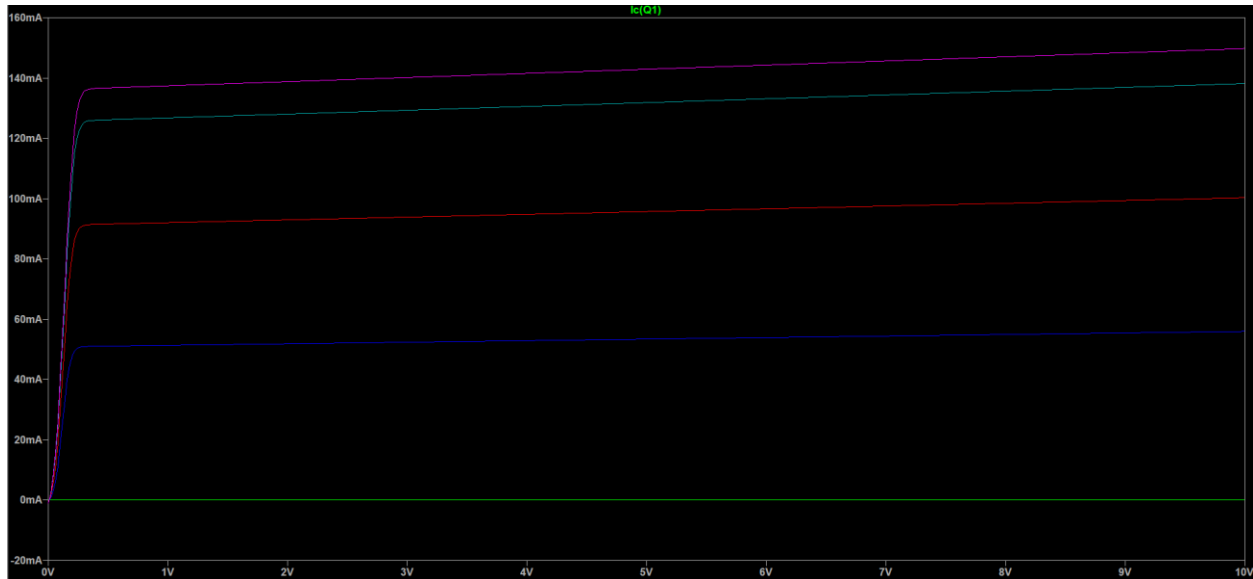
+Cjc=7.306p Mjc=.3416 Vjc=.75 Fc=.5 Cje=22.01p Mje=.377 Vje=.75
+Tr=46.91n Tf=411.1p Itf=.6 Vtf=1.7 Xtf=3 Rb=10)
.DC VCE 0 10V .02V IB 0 1MA 300UA

```

```
.PROBE
```

```
.END
```

OUTPUT:



Output characteristics of BJT in common emitter mode is the curve between collector current  $I_C$  and collector-emitter voltage  $V_{CE}$  at constant base current  $I_B$



## CHAPTER 4

### CONCLUSION

In this project report we have simulated so many circuit such as passive High pass filter, low pass filter, comparator, output characteristics of BJT, NMOS, DC Operating point of transistor, Half wave rectifier, Full wave rectifier, Astablemultivibrator, Inverting amplifier.

The first experiment is Inverting amplifier, Amplifier is a circuit that rises the strength of weak signal ( input signal ). In this experiment we have used LT1001 op-amp. Sine wave voltage source is applied at inverting terminal of an op-amp and the non-inverting terminal is grounded. A feedback resistor is connected between output and input of an op-amp. The sinusoidal output voltage is obtained. The amplitude of an output voltage is higher than input voltage.

This circuit is analysis by Transient analysis in which a non-linear, time domain analysis is performed.

The second experiment is high pass filter: A simple passive R-C circuit used as a high pass filter. A High pass filter ( HPF ) is an electronics filter that passes signal with a frequency higher than a certain cut-off frequency and attenuates signal with frequency lower than the cut-off frequency. The amount of attenuation for each frequency depends on the filter design.

In this experiment we have done AC analysis simulation. AC analysis computes the small-signal AC response of a circuit as a function of frequency.

The third experiment is R-C low pass filter: A passive low pass filter is design by connecting a resistor and capacitor with ac voltage source.

A low pass filter ( LPF ) is an electronics filter that passes signal with a frequency lower than a certain cut-off frequency and attenuates signal with frequency higher than the cut-off frequency.

This circuit is also simulated by AC analysis.

The fourth experiment is non-inverting op amp based comparator. Which compare two input voltages and produce single output voltage.

When the voltage non-inverting terminal is greater than inverting terminal ( reference terminal ) voltage. The comparator output is HIGH.

When the voltage non-inverting terminal is less than inverting terminal ( reference terminal ) voltage. The comparator output is LOW.

Here also simulation analysis is transient analysis.

The fifth experiment is Astablemultivibrator. A multivibrator is an electronic device that generates square wave of its own ( i.e. without any external triggering pulse ) is known as an astable or free running multivibrator. Astablemultivibrator has no stable states. It switches back and forth from one states to the other, remaining in each states for the time determined by circuit constants. It is analysis by transient analysis.

The sixth experiment is on operating point analysis. The .OP command replace all capacitor with an open circuit and all inductor with an short circuit and calculates the DC solution for the circuit. The results are displayed in a dialog box that pops up after the simulation is complete. Here we simulate the characteristics of BJT at DC operating point.

The seventh experiment is half wave rectifier.A Half wave rectifier is defined as a type of rectifier that allows one half cycle of an AC voltage waveform to pass, blocking the other half-cycle. Half wave rectifier used to convert AC to DC voltage, and require a single diode to construct. It is analysis by transient analysis.

The eight experiment is Full wave rectifier.A full wave rectifier convert both halves of each cycle of an alternating wave ( AC signal ) into pulsating DC signal. In full wave bridge rectifier four diodes are used.. It is analysis by transient analysis.

The ninth experiment is output characteristics of NMOS transistor. Output characteristics of NMOS transistor is basically a  $I_D$  versus  $V_D$  curve.

It is analysis by DC SWEEP analysis. In this analysis, a DC source is swept in a determined voltage range, so the response of a variable as function of such voltage sweep can be plotted. This means that we can find the transfer function from input source to some output variable.

The tenth experiment is on output characteristics of BJT in common emitter confuguration.Output characteristics of BJT in common emitter mode is the curve between collector current  $I_C$  and collector-emitter voltage  $V_{CE}$  at constant base current  $I_B$ . It is analysis by DC sweep analysis.

LTSPICE is easy to use circuit designing software and can be used for analysis for any type of circuit. It is less time consuming and all circuits can be executed accurately. Furthermore, it also provides graphical representation of the circuit as well.

It's cheaper than buying a lot of real parts and a real signal generator and oscilloscope.

In simulation, the values of components can be change. It can save time by designing a prototype of a circuit, by simulation we can check the behaviours of the circuits and then performed on the hardware.

## REFERENCES

[1] **From Wikipedia-** Ltspice- [Online] . <https://en.wikipedia.org>. [Accessed: 10- aug- 2020] .

---

[2] **Trevor Gamblin**. "Basic Circuit Simulation with LTspice". July 30, 2015. [Online] . <https://www.allaboutcircuits.com>. [Accessed: 10- aug- 2020] .

[3] "LTspice". <https://www.analog.com>. [Accessed: 10- aug- 2020] .