Introduction to SwitcherCAD

1 PREFACE

1.1   What is SwitcherCAD?

SwitcherCAD III is a new Spice based program that was developed for modelling board level switching regulator systems. The program consists of a schematic capture program that allows users to create and edit circuits, high performance Spice simulator called LTspice and a waveform viewer for displaying the simulated waveforms. SwitcherCAD is freely downloadable from the Linear Technology website

LTspice, simulation engine of SwitcherCAD, is a software package based on industrial standard software for circuit simulation called Spice (Simulated Program with Integrated Circuit Emphasis).

Spice was built in the seventies at the University of California, Berkley and subsequently improved and rewritten to commercial form by number of software companies. The most famous commercial form of Spice, called PSpice, was written in 1980 by Microsim Corporation and from 1997 is integrated in OrCad design software package.

Note 1: Linear Technology website: www.linear.com

Note 2: LTspice is a trade mark of Linear Technology Corporation

Note 3: Philosophy of work in SwitcherCAD is very similar to work in OrCad PSpice.

1.2   The structure of SwitcherCAD

SwitcherCAD incorporates three main parts:

- **Schematics Capture** is used for creating end editing schematics of analogue, digital or mixed circuits.

- **LTspice** can provide several types of analysis like operating point analysis (OP), direct current analysis (DC), alternating current analysis (AC) and transient analysis (TRAN).

- **Waveform viewer – S**witcherCAD includes an integrated waveform viewer.

2 LTSPICE FUNDAMENTALS

2.1 How does Spice work?

Programme Spice processes an input file which contains circuit description and simulation commands. Input file is a simple text file with .cir (or .net in LTspice) suffix. It is possible to create Spice input file in any text editor. Commercial Spice based software packages usually contain a schematics editor, which enables to draw schemes with all necessary entries for simulation and then distil a Spice input file. During its activity Spice generates output text file with .out (.log in LTspice) suffix which contains a copy of input file, information about simulation, errors and in reduced range simulation results (operating point); then the large file (with suffix .dat or .raw for LTspice) containing all simulation results is created. This file can be processed by any waveform viewer (for example Probe in OrCad PSpice package), which enables visualization of simulation data.

Even though in SwitcherCAD the three programmes mentioned above are integrated into one executable file, the philosophy of Spice was preserved. It is still possible to prepare the input file for LTspice in any text editor or export simulation results to the text file.
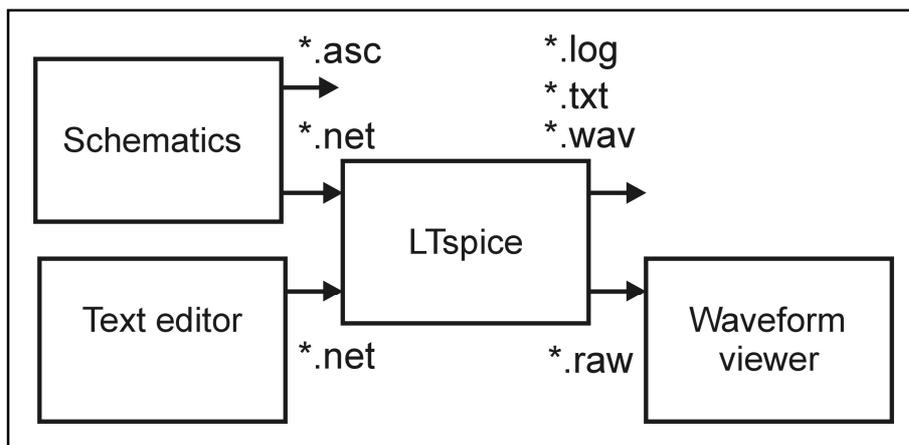

Figure 2.1 *SwitcherCAD structure*

2.1.1 LTspice input file structure

A general LTspice program consist of the following components:
- Title
- Element statements
- Control statements
- End statements

Title is the first line in the LTspice input file. It has only informational role, it is ignored by LTspice and can be blank.

Comments
An asterisk '*' in the first column of a line indicates a comment line. You can use semicolon ';' form commenting anywhere in your programme.

Element statement specifies the elements in the circuit. The element statement consist of the element name, circuit nodes to which it is connected and the values of the parameters that characterize the element.

Examples:

```
R3 N005 N003 480kOhm;  resistor declaration

V2 N003 0 12V ; independent voltage source declaration

Q1 N005 N004 0 0 NPN ; bipolar junction transistor
```

The element name must begin with a letter of the alphabet that is unique to a circuit element (see Note 4).
Circuit nodes can be any string. Node 0 is predefined for the ground of a circuit.
Elements values can be integer or floating point numbers. It is possible to use exponential form or numbers followed by scaling factor (see Note 5). Any character after the scaling factor abbreviation is ignored (see examples above).

Control statements
To run a simulation, not only must the circuit be defined, but also the type of analysis to be performed. Simulation progress is driven by control statements. All control statements start with a dot, therefore they are called "Dot commands".

It is possible to divide Dot commands into several categories:

1) References, commands for including or linking files

File insertion
```
    .INCLUDE  <"File name">
```
In the included file the first line must contain comment.

Library reference
```
    .LIB   <"File name">
```
This command can be used as a reference to models or subcircuits in any library file.

Note 4: LTspice elements

First letter of name	Circuit element
A	Special functions
B	Arbitrary behavioral voltage source
C	Capacitor
D	Diode
E	Voltage-controlled voltage source
F	Voltage-controlled current source
G	Current-controlled current source
H	Current-controlled voltage source
I	Current source
J	JFET transistor
K	Mutual inductors
L	Inductor
M	MOS FET transistor
O	Loosy transmission line
Q	Bipolar transistor
R	Resistor
S	Voltage controlled switch
T	Transmission line
U	Uniform RC line
V	Voltage source
W	Current controlled switch
X	Subcircuit
Z	MESFET transistor

Note 5: Abbreviations

Spice suffix	Multiplaying Factor
T	10^{12}
G	10^{9}
Meg	10^{6}
K	10^{3}
M	10^{-3}
U	10^{-6}
N	10^{-9}
P	10^{-12}
F	10^{-15}
Mil	25.4×10^{-6}

2) Commands for the part or circuit properties modification

Subcircuits

```
.SUBCKT <"Name"> ["Node list"]
[PARAMS:("Name"="Value")]
    device statements
.ENDS ["Name"]
```

Make possible to define a circuit. Repetitive circuitry can be enclosed in a subcircuit definition and used as multiple instances in the same circuit.

Models

```
.MODEL <"Model name"> <"Model type">
[("Parameter" = Value")]
```

This direction defines a set of parameters, which characterizes properties of the concrete part. `Model name` can chooses user, `model type` refer to standard models in LTspice.

Parameters

```
.PARAM  <"Parameter name" = "Value or
         expression">
```

The `.PARAM` direction allows creation of user defined variables. It is possible to simply assign the same value to several parts. Also this directive can by useful for parametric analysis (parameter can change during repeated simulation).

Initial values

```
.IC <V("Node")= "Voltage value">
    <I("Inductor")= "Voltage value">
```

For some types of analysis it is necessary to set initial conditions like voltages in nodes or currents through inductors.

Functions

```
.FUNC <Name> [List of arguments]
            {<expression>}
```

It is possible to define functions for simple usage of repetitive part of programme. The name of user-defined functions can not be equal to any predefined function. In expression you can use any function defined above (or predefined).

3. Commands for analysis control

Operating point analysis

```
.OP
```

Operating point is the steady state for direct currents and voltages. During calculation capacitances are open-circuited and inductances short-circuited.

Direct current (DC) analysis

```
.DC  <Source name> <Start value>
     <Stop value> <Increment value>
```

The .DC control statement specifies the values that will be used for direct current sweep analysis. It is useful for computing the DC transfer function of an amplifier or plotting the characteristic curves of a transistor for model verification. `Source name` is the name of an independent voltage or current source.

Alternating current (AC) analysis

```
.AC <lin,oct,dec> <Steps> <Start> <Stop>
```

The .AC control statement is used to computing the AC complex node voltages as a function of frequency which means it is computed both amplitude and phase.
First the DC operating point is found. Next, the nonlinear parts are linearized in the DC operating point. Frequency of all independent voltage sources changes simultaneously. If you choose `lin`, then `Steps` means total number of frequencies. If you choose `oct` or `dec,` then `Steps` means the number of steps in one octave or decade.

Transient analysis

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

This direction enables to perform a transient analysis. This is the most direct simulation of a circuit. It basically computes what happens when the circuit is powered up. Test signals are often applied as independent sources. During the computing the variable TIME is changed from zero to `Tstop`. Variables `Tstep`, `Tstart` and `dTmax` refer only to output to the waveform file. `Tstep` is a step for printout. The step for computation changes adaptively. If `Tstart` is specified, the waveform data between zero and `Tstart` is not saved.

Similarly data between `dTmax` and `Tstop` is not saved.

Fourier analysis

```
.FOUR <Frequency> [Nharmonics] [Nperiods]
      <Data trace1> [<Data trace2> ...]
```

Fourier analysis computes amplitude and phase of harmonic components of specified variables (`Data trace`). You can use Fourier analysis only after transient analysis. This direction is supported only for back compatibility. Fourier analysis implemented in waveform viewer is more useful.

Parametric analysis

```
.STEP [PARAM] [LIN,OCT,DEC] <Variable>
      [LIST Values] <Start> <Stop> <Step>
```

This command causes an analysis to be repeatedly performed while stepping the temperature, a model parameter, a global parameter, or an independent source. Steps may be linear, logarithmic, or specified as a list of values.

DC Transfer function

```
      .TF V(<Node>[, <ref>])  <Source>
      .TF I(<Voltage source>) <Source>
```

This is an analysis mode that finds the DC small signal transfer function of a node voltage or branch current due to small variations of an independent source.

3 WORKING WITH SWCAD

3.1   Operating point analysis

Examples in this chapter provide an introduction to the methods and tools for creating circuit design, running simulations, and analyzing simulation results.
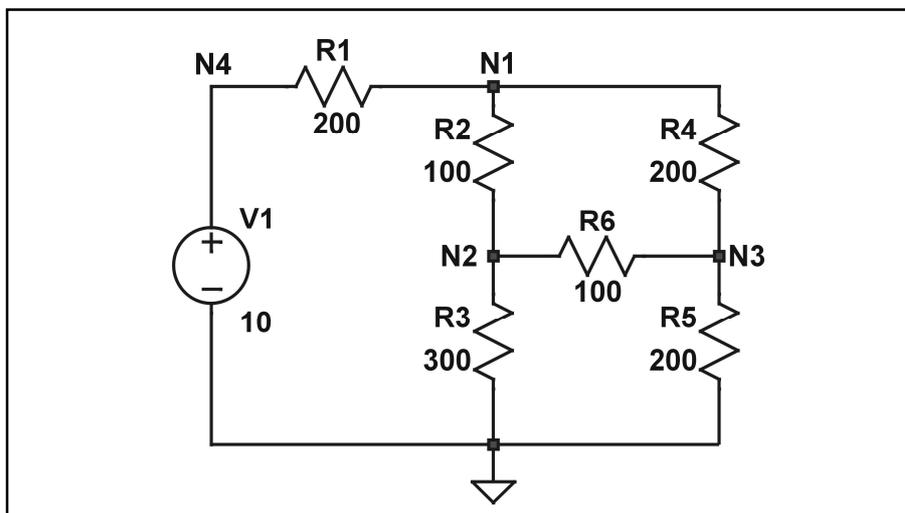

Figure 3.1 *Bridge circuit*

3.1.1   Creating new schemes

1   From the Windows Start menu, choose SWCAD III.
2   From the File menu choose New Schematics.

3.1.2   Placing parts

1   From the Edit menu choose Component – Select Component Symbol dialog window appears.
2   Select part from list and click OK.
3   If you want to rotate the part press Ctrl+R.
4   Move pointer to the correct position and click left to place a part.
5   Click right to stop placing the parts.

3.1.3   Connecting parts

1   From the Edit menu choose Draw Wire.
2   Click the connection point of the first part, move the pointer to the connection point of the second part and click again.
3   If you want to assign a name (label) to the wire, click right on the wire and choose Label Net item.

or press F2

or press F3

or press F4

Note: If you are preparing a scheme for simulation it is necessary to name any net 0 or connect with the special part called GND.

3.1.4   Placing GND

1   From the Edit menu choose Place GND.
2   Move pointer to the correct position and click left to place GND.
3   Connect GND to any net.

3.1.5   Editing Parts

1   Click right on the part.
2   In the dialog box which appears you can change part properties.

3.1.6   Deleting Parts

1   From the Edit menu choose Delete, scissors cursor appears.
2   Click on the part you want to delete.

Operating point is a DC solution with capacitances open circuited and inductances short circuited. Usually a DC solution is performed as a part of another analysis. During operation point analysis the following values are counted.

-    Voltages at each node of a circuit
-    Currents and power dissipation of all voltage sources in a circuit
-    Transistor diode parameters, if this devices are present in a circuit

The results of operating point analysis will appear in a dialog box. After operating point simulation, when you point at a node or current the solution will appear on the status bar.
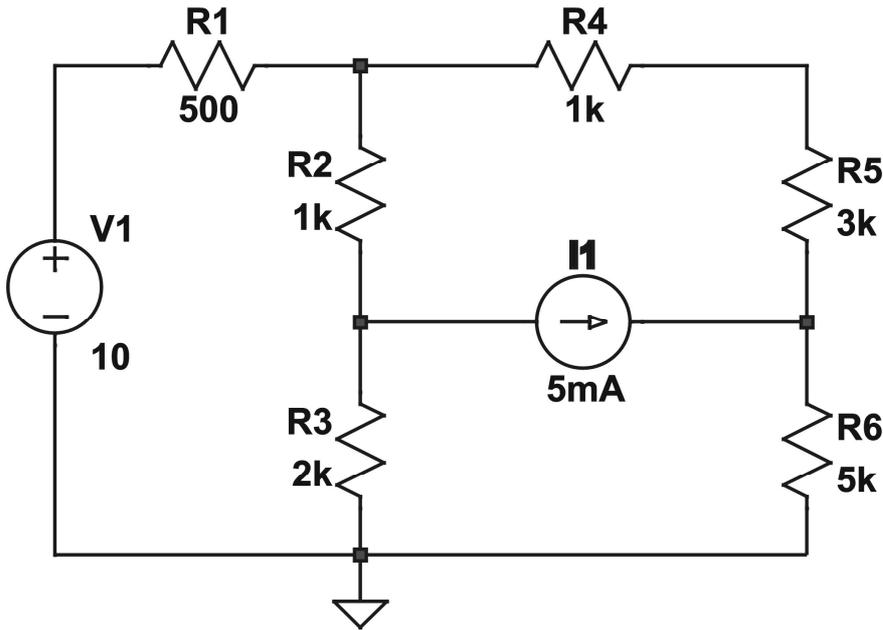
3.1.7   Providing Operating point analysis

1   From the Simulate menu choose Edit simulation cmd.
2   In the simulation dialog box click on the DC op pnt bookmark.
3   Click Ok.
4   From the Simulate menu choose Run, dialog box with results appears.
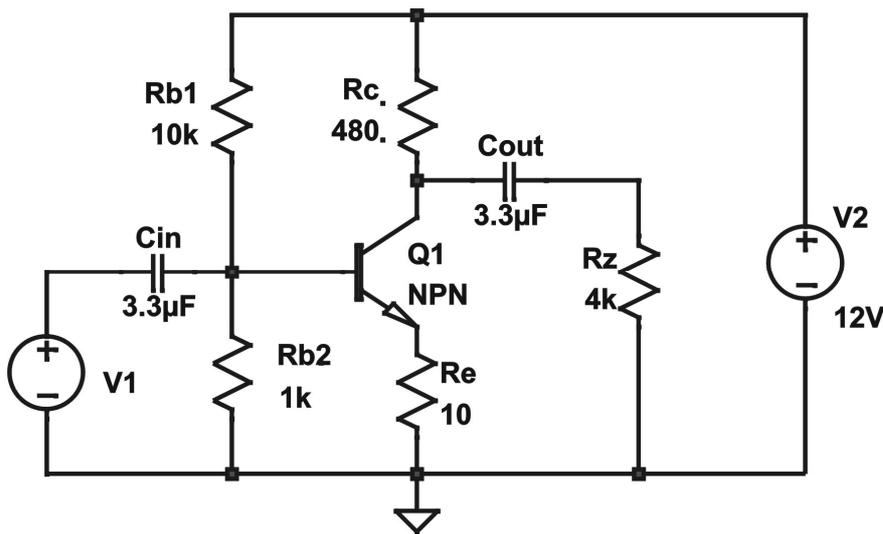5   If you click on any node or part the result will appear in the status line.

DC operating point: I(V1)=-25.5556mA   Dissipation=-255.556mW

or press G

or press F5

9

3.2   Exercise

1. Determine all node voltages for given resistive circuit with multiple sources.

2. Determine voltage on the collector of transistor for given common-emitter amplifier.

3.3   Parametric operating point analysis

Parametric analysis is a repeated calculation of operating point on condition that at least one of the circuit parameters is changed. A variance of circuit parameter can be linear or logarithmic.

Suppose a bridge circuit from the example 3.1. We can ask a question: "How will the current through the resistor R6 change, if the resistor R2 changes. To solve this problem we have to make several changes in the circuit from example 3.1.

If we would like the resistance of R2 to change during parametric analysis, we have to substitute concrete value by parameter (or variable).

3.3.1   Defining parameter

1   Click right on value of the R2 resistor, the dialog box appears.
2   Type {R} to the edit box.
3   Press OK

Note 5: Name of parameter must be between braces.

3.3.2   Preparing parametric analysis

It is necessary to use Spice directive .STEP (see 2.1 How does Spice work) for performing parametric analysis in LTspice.
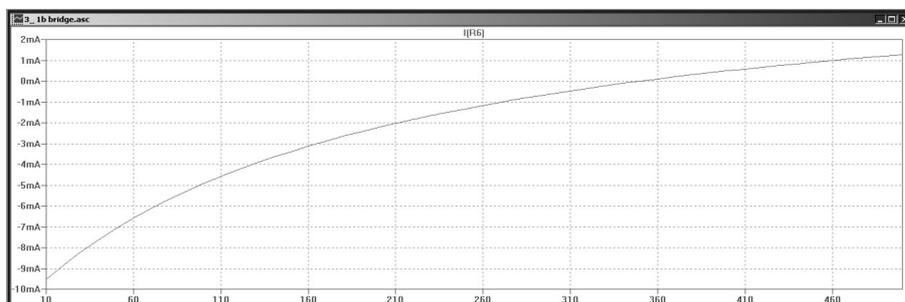
1   Choose Spice directive item from the Edit menu.
2   Type .STEP PARAM R 10 500 10 into the dialog box.
3   Click OK, choose suitable place on your schematics for placing the directive and then click left.

3.3.3   Running simulation

1   From the Simulate menu choose Run, dialog box with results appears.
2   Select I(R6) in the list and click OK.

3.3.4   Result

Problem: *What will happen when R6 resistance value changes simultaneously?*

3.4 DC sweep analysis

The DC sweep analysis causes a DC sweep to be performed on the circuit. DC sweep allows you to sweep a source (voltage or current), a global parameter or model parameter through a range of values.

Suppose bridge circuit from the first example (Figure 2). We can change source voltage and observe how current trough resistor R3 changes.

3.4.1 Setting up and running a DC sweep analysis

1   From the Simulate menu choose Edit simulation cmd.
2   In the Simulation dialog box click on the DC Sweep bookmark.
3   Type V1 into the Name text box.
4   Write -10 to the Start Value text box, 10 into the Stop Value text box and 0.1 to the Increment text box.
5   Press OK button.
6   Choose suitable place for analysis description and click left.
7   From the Simulate menu choose Run. Dialog box with results appears.
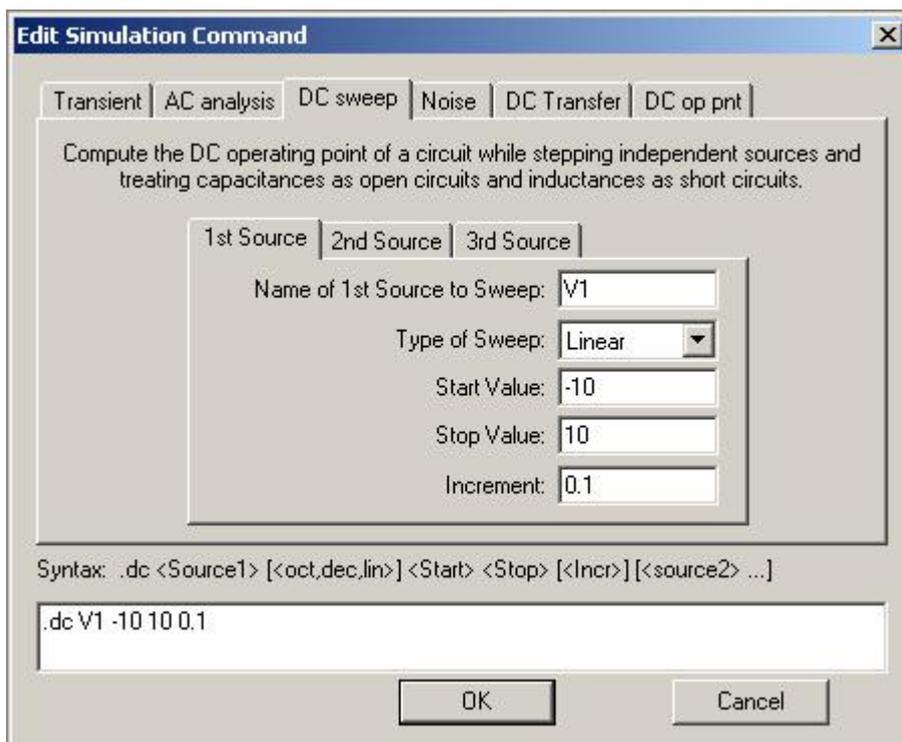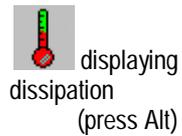8   From the dialog box choose values (voltages or currents) you would like to draw.

Edit Simulation Command

Transient | AC analysis | DC sweep | Noise | DC Transfer | DC op pnt

Compute the DC operating point of a circuit while stepping independent sources and treating capacitances as open circuits and inductances as short circuits.

1st Source | 2nd Source | 3rd Source

Name of 1st Source to Sweep: V1
Type of Sweep: Linear
Start Value: -10
Stop Value: 10
Increment: 0.1

Syntax: .dc <Source1> [<oct,dec,lin>] <Start> <Stop> [<Incr>] [<source2> ...]

.dc V1 -10 10 0.1

OK   Cancel

Figure 3.2 *DC sweep analysis settings*

3.4.2   Displaying DC sweep analysis results

1   A new window of waveform viewer appears after simulation.
2   If you want to add a new trace simply click on any wire (for adding voltage waveform) or any part (for adding current waveform).
3   For deleting trace choose Delete Traces from the Plot Settings menu and click on the trace name on the top of the waveform viewer.

displaying currents

displaying voltages

displaying dissipation
      (press Alt)

Note: You can click on one node and drag the mouse to another node for differential voltage plot.

Figure 3.3 SWCAD screen with waveform viewer

It is of course possible to sweep several sources simultaneously. Finding bipolar junction transistor output characteristics can be mentioned as a good example of application.

Note 6: In the Simulation dialog box on the DC sweep bookmark, you can set up three independent sources at the most, for DC sweep analysis.
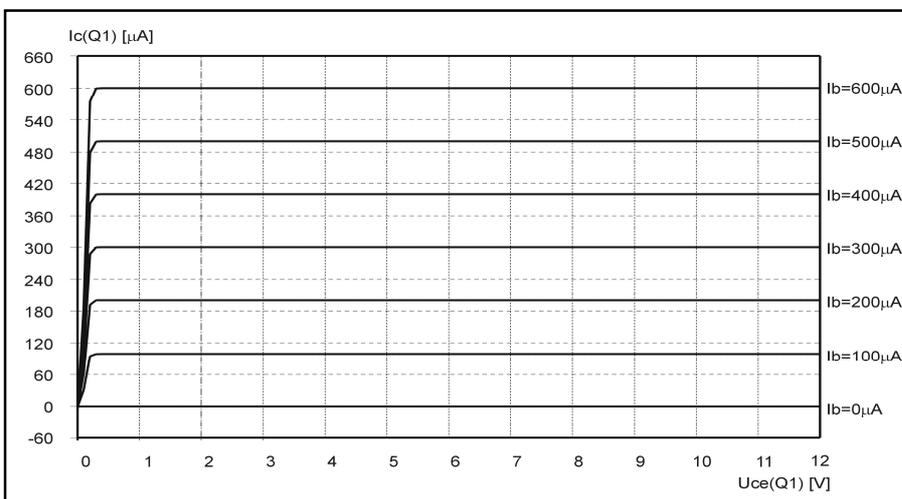
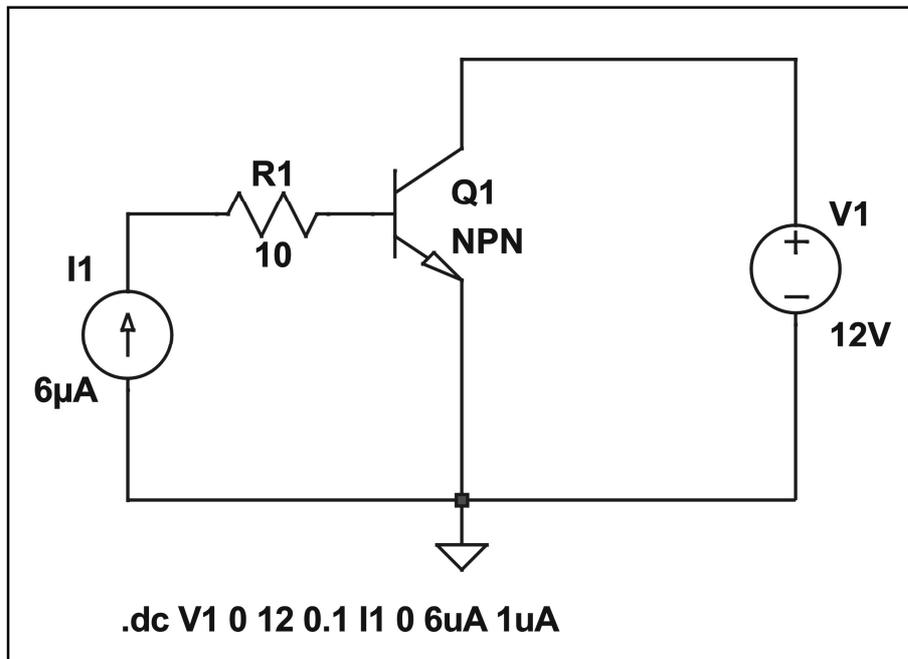**Figure 3.4** : *Bipolar junction transistor output characteristics*

Figure 3.5 : *Circuit for obtaining BJT output characteristics*

3.5 Small signal DC Transfer function

This analysis can be used to obtain small-signal gain, dc input resistance, and dc output resistance of a circuit by linearizing the circuit around a bias point.

Example: Determine input resistance, output resistance and gain of given inverting operational amplifier.
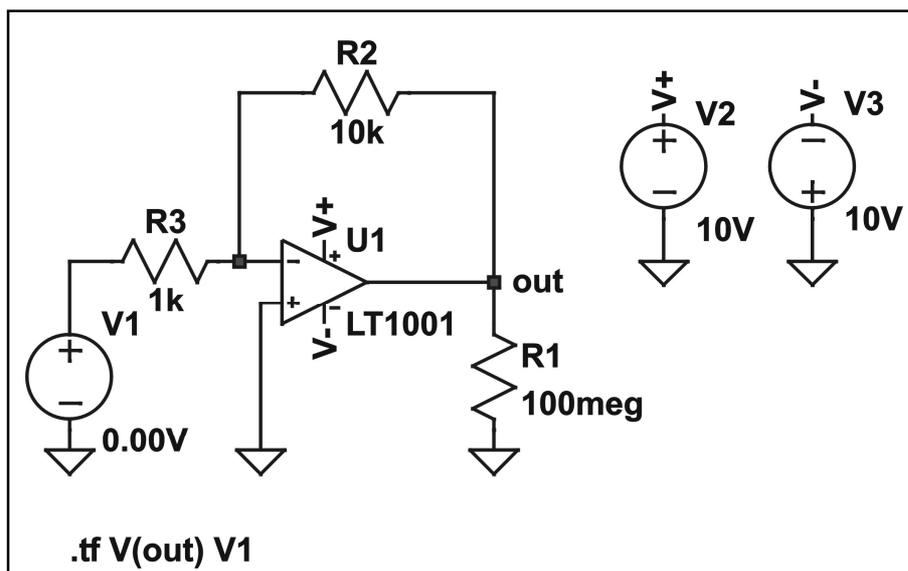

Figure 3.6 : Inverting operational amplifier

3.5.1   Simulation

1. Create a new file and draw the circuit. Operational amplifier LT 1001 you can find in Opamps folder.
2. It is necessary for the operational amplifier LT1001 to have power supply. In this case it is done using labels (see figure above).
3. It is suitable to assign labels to the output of operational amplifier. Click right on the LT1001 output node, a local menu appears. Select Label net from this list and then type out to the edit box.
4. From the Simulate menu choose Edit Simulation Cmd item.
5. Click on the DC Transfer bookmark.
6. Type V(out) to the Output edit box and V1 to the Source edit box.
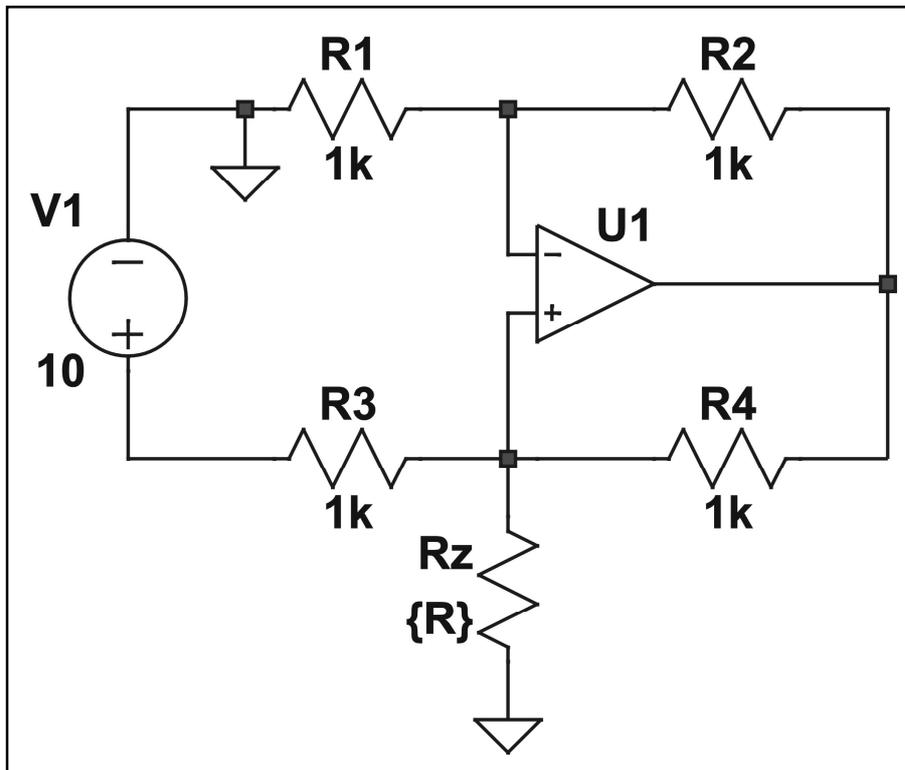7. Run the simulation.

or press F4

Note 7: It is possible to use DC transfer analysis together with parametric analysis.

3.6   Exercises

1. Draw the V/A characteristic of diode.
2. Suppose circuit on the figure below. Draw dependence $I_{R_z} = f(R_z)$. What is the use of this circuit?

3.7   AC sweep analysis

The AC sweep analysis in SwitcherCAD is a linear frequency domain analysis that can be used to observe the frequency response of any circuit at its bias point.
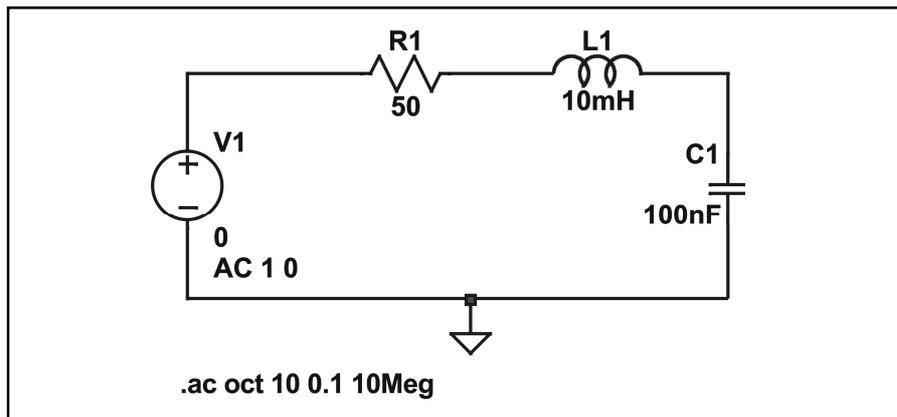

Figure 3.7 *Low pass filter*

Note 6: Abbreviations

Spice suffix	Multiplaying Factor
T	10^{12}
G	10^{9}
Meg	10^{6}
K	10^{3}
M	10^{-3}
U	10^{-6}
N	10^{-9}
P	10^{-12}
F	10^{-15}
Mil	25.4×10^{-6}

3.7.1   Example circuit creation

1   Start SwitcherCAD and create new schematics.
2   From the Edit menu choose Component, Select Component Symbol dialog box appears.
3   Select Voltage from the list.
4   Place Voltage source and click right, Independent Voltage Source dialog box appears.
5   Click on the Advanced button.
6   Type 0 into the DC Value edit box, 1 into the AC Value edit box, 0 intto the AC Phase edit box and click OK.
7   Place and connect the other parts (see Fig. 4).

3.7.2   Setting up and running an AC sweep analysis

1   From the Simulate menu choose Edit simulation cmd.
2   In the Simulation dialog box click on the AC Sweep bookmark.
3   Type 10 into the Number of points per octave edit box, 0.1 into the Start Frequency edit box and 10Meg into the Stop frequency edit box, then click OK.
4   Choose a suitable place for analysis description and click left.
5   From the Simulate menu choose Run, dialog box with results appears.
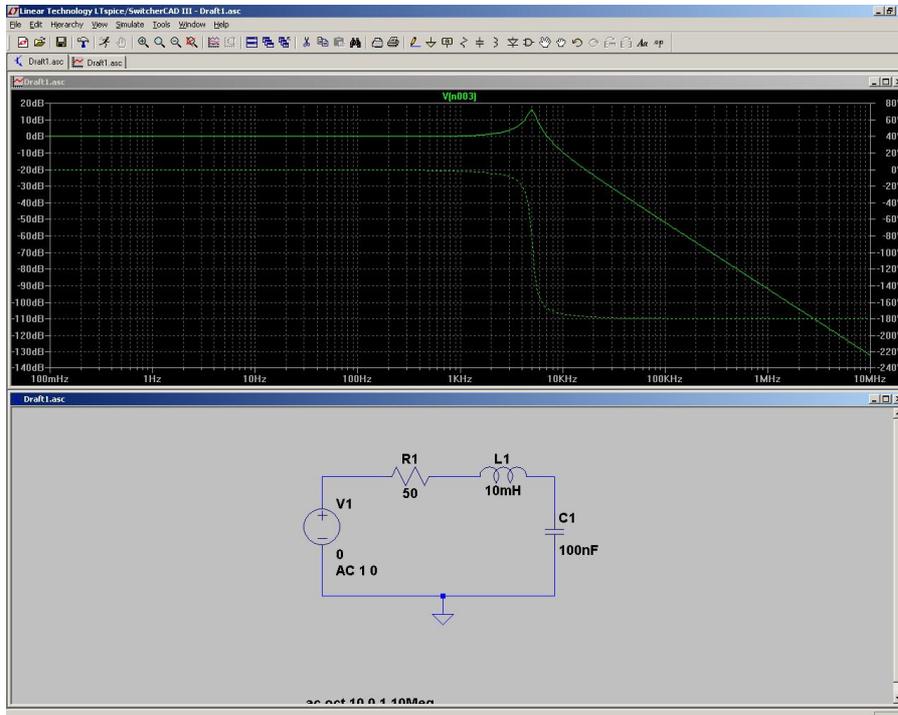6   From the dialog box choose values (voltages or currents) you can draw.

Figure 3.8 *AC analysis simulation results*

3.8 Transient analysis

3.8.1 Example circuit creation

1 Assume low pass filter (see Fig. 4). Create new schematics and draw the circuit. As a source use part Voltage again.
2 Right click on the voltage source, Independent Voltage Source dialog box appears.
3 Select Pulse item from the Functions list.
4 Set parameters as below and click OK.

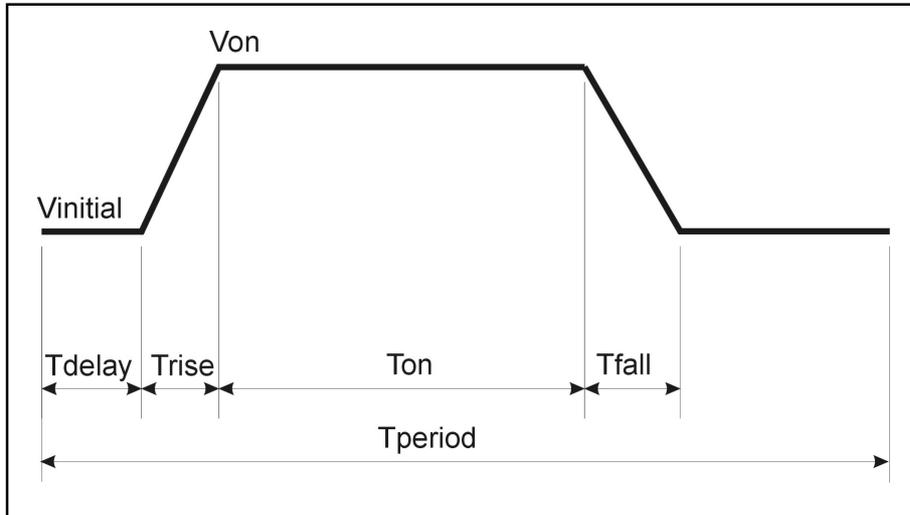| Parameter	Value
Vinitial	0
Von	10
Tdelay	0
Trise	0.1n
Tfall	0.1n
Ton	5u
Tperiod	0
Ncykles	0

17

Figure 3.9 *PULSE source voltage waveform*

3.8.2 Setting up and running a Transient analysis

1 From the Simulate menu choose Edit simulation cmd.

7 In the Simulation dialog box, click on the Transient bookmark.

8 Type 4ms into the Stop time edit box, 0 into the Time to start saving data edit box and 1u into the Maximum timestep edit box, then click OK.

9 Choose a suitable place for analysis description and click left.

10 From the Simulate menu choose Run, dialog box with results appears.

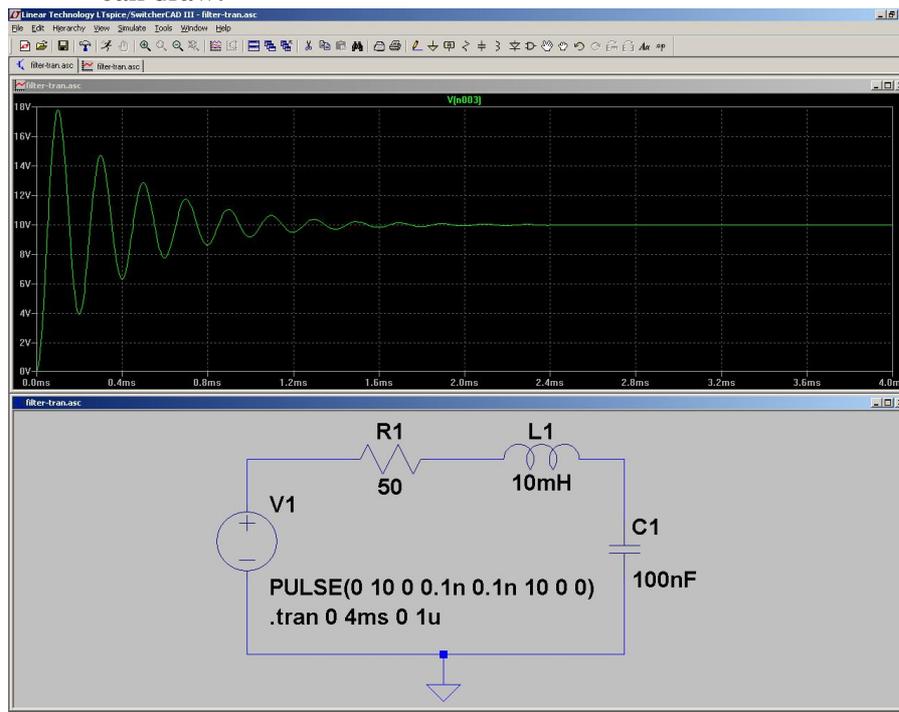11 From the dialog box choose values (voltages or currents) you can draw.

Figure 3.10 *Transient analysis simulation results*