304-334B Electronic Circuits II

Tutorial on PSPICE

Department of Electrical & Computer Engineering
McGill University, Montreal, Canada

Rola A. Baki, Winter 2002

Introduction

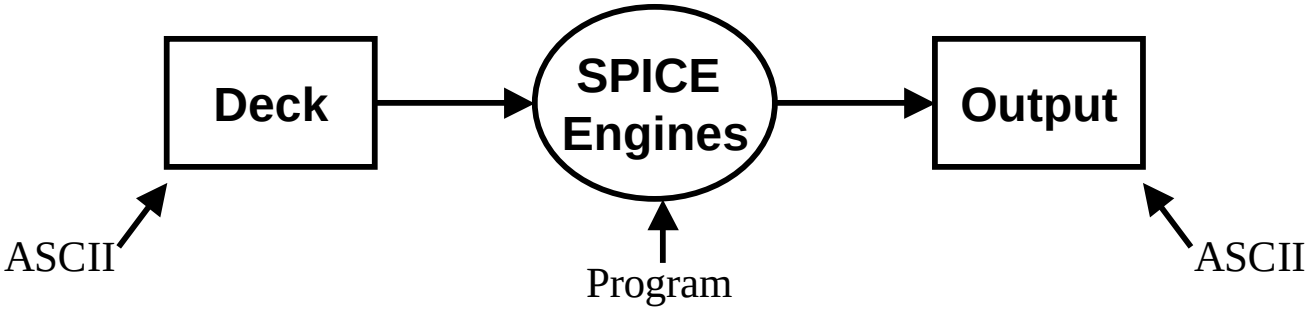
SPICE = **S**imulation **P**rogram with **I**ntegrated **C**ircuits **E**mphasis

⇨ Program designed to analyze integrated circuits.
Also simulates any network, which could be represented by discrete components (resistors, capacitors, inductors and dependant independent sources).

⇨ SPICE engines "digitally" solve continuous time differential equations describing circuits.

How to use Spice?

We first have to write a Spice input file, also called "deck".



Introduction

The input file must contain:

1. A title statement: ALWAYS on the first line of the input file.
2. A .END command: ALWAYS on the last line.
3. Circuit description: Lines that contains or a statement, which describe a circuit element, or a control line (i.e. analysis type, measurements nodes, or model parameters).

➔ Comment statements should contain an asterisk (*) as a first character of the comment line.

➔ The order of the SPICE lines is not important except for the title line and .END command.

➔ SPICE is not case sensitive and words can be separated by an arbitrary number of spaces.

➔ Components must be uniquely labeled.

➔ Every node of the circuit is designated by a number. The ground node is labeled “0”.

Example of an Input File

Title → `Low pass filter`

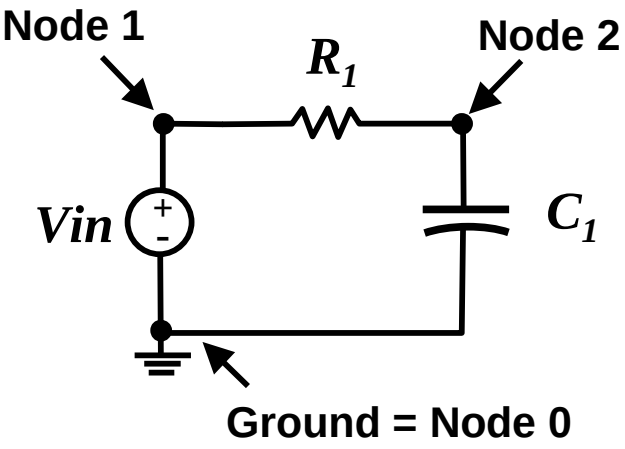
```
** Circuit Description **
* Power supply
Vin 1 0 PWL(0s, 0V,1ms,0V,
+ 1.0001ms, 1V)

* Elements description
R1   1   2 1Kohm
C1   2   0 1uF

*Analysis request
.OP
.Tran 0.1ms 5ms

.end
```

The "+" sign means that the previous command is continued on this line.

Positive Terminal (n+)

Negative Terminal (n-)

.END command



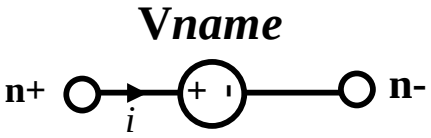
Passive elements

Basic elements types:

1st Letter Presentation	Element
C	Capacitor
R	Resistor
L	Inductor
Q	Bipolar Transistor
M	MOS Field Effect Transistor
I	Independent current source
V	Independent voltage source
E	Voltage-controlled voltage source
F	Current-controlled current source
G	Voltage-controlled current source
F	Current-controlled voltage source

Independent Source Representation in Spice

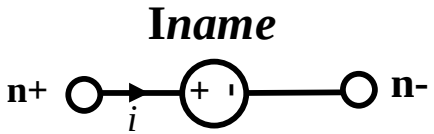
Voltage source:



Current flowing into (n+) is positive

Spice Description	Type of Analysis
V*name* n+ n- DC *value*	All types
V*name* n+ n- AC *Magnitude Phase*	AC Frequency response
V*name* n+ n- SIN (V_o V_a *freq* t_d *damp*)	Transient
V*name* n+ n- PULSE (V_1 V_2 t_d t_r *PWT*)	Transient
V*name* n+ n- PWL (t_1, v_1, t_2, v_2, …, t_n, v_n)	Transient

Current source:



The Spice description for an independent current source is the same as the voltage source, except replace “V” by “I”.

Analysis Request and Output Request

After specifying the circuit description, we still need to:

1. Specify the type of analysis for our simulations.
2. Choose the network variables that we want to display.

Analysis Requests:

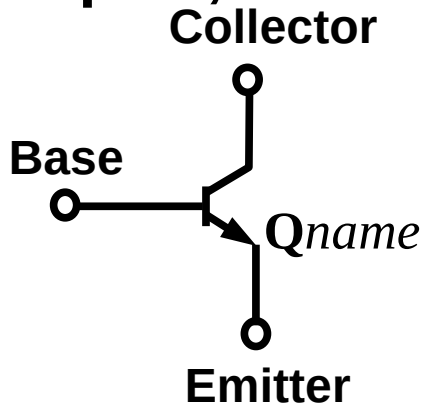
Operating point	**.**OP
DC sweep	**.**DC *source_name start_value_stop_value_step*
AC frequency response	**.**AC DEC *points_per_decade freq_start freq-stop*
	.AC OCT *points_per_octave freq_start freq-stop*
	.AC LIN *total_points freq_start freq-stop*
Transient response	**.**TRAN *time_step time_stop [(no_print_time Max_step_size)]*

Output Requests:

Print data points	**.**PRINT DC *output_variables*
	.PRINT AC *output_variables*
	.PRINT TRAN *output_variables*
Plot data points	**.**Plot DC *output_variables [(lower_plot_limit upper_plot_limit])*
	.Plot AC *output_variables [(lower_plot_limit upper_plot_limit])*
	.Plot TRAN *output_variables [(lower_plot_limit upper_plot_limit])*

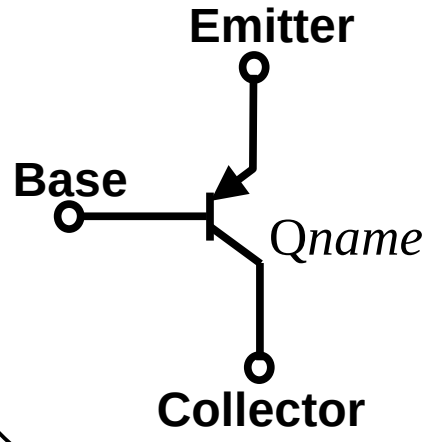
Description of a Bipolar Transistor

In Spice, the NPN transistor is defined as follows:



Spice description:
Q*name* collector base emitter substrate *BJT_model_name*
. MODEL *BJT_model_name* **NPN** *(parameter_name = value …)*

PNP transistor is defined as:



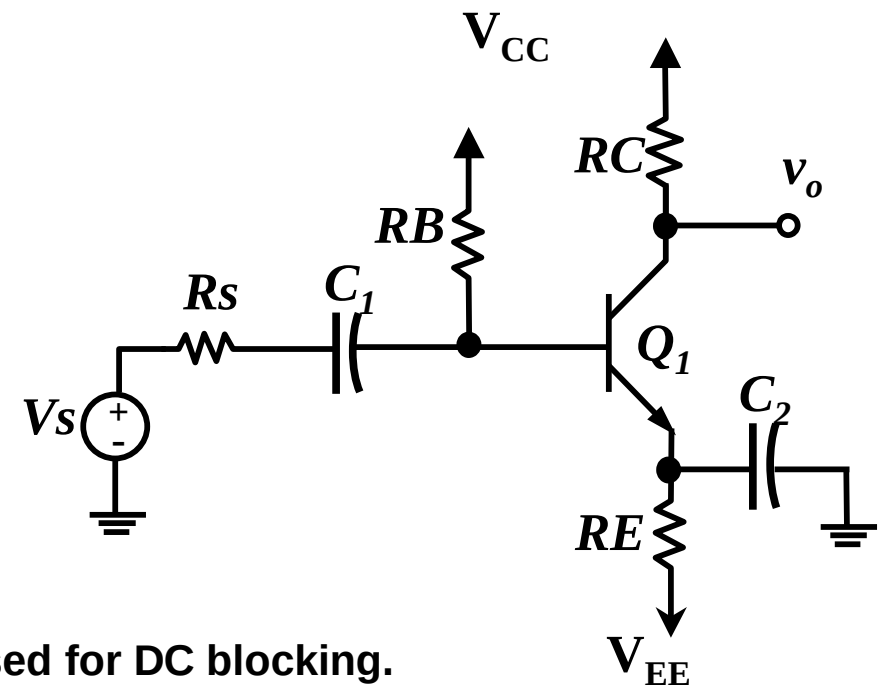
Spice description:
Q*name* collector base emitter substrate *BJT_model_name*
. MODEL *BJT_model_name* **PNP** *(parameter_name = value …)*

Example of a Common Emitter Amplifier

Common-Emitter Amplifier Stage

```
** Circuit Description **
power supplies
V_CC 1 0 DC +10V
V_EE 8 0 DC -10V
* input signal
Vs 6 0 AC 10mV
Rs 5 6 10k
* amplifier
Q1 2 4 3 Q2N2222A
RB 1 4 100k
RC 1 2 10k
RE 3 8 10k
C1 4 5 1GF
C2 3 0 1GF
```

Infinite capacitors, used for DC blocking.




```
* transistor model statement for the 2N2222A
.model Q2N2222A 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)
** Analysis Requests ** * calculate DC bias point information
.OP
.AC LIN 1 1kHz 1kHz
.end
```

Description of a Subcircuit

Some circuit elements are not always available in the SPICE library (for example, op-amps). To add an op-amp to the SPICE deck, a “subcircuit” that represents this op-amp could be defined and incorporated into the main circuit.

The definition in SPICE for a subcircuit is as follows:

.SUBCKT *subcircuit_name* list_of_nodes
Circuit Description
.ENDS

To incorporate the subcircuit into the main design, use the following statement, which starts with the letter “X”:

X*name* node_connections *subcircuit_name*

Example of An Amplifier with a gain of -1

```
Inverting Amplifier With Gain -1
*** op-amp subcircuit
.subckt small_signal_opamp 1 2 3
*          connections:    | | |
*                       output | |
*                     +ve input |
*                           -ve input
Ginput 0 4 2 3 0.19m
Iopen1 2 0 0A ; redundant connection made at +ve input terminal
Iopen2 3 0 0A ; redundant connection made at -ve input terminal
R1 4 0 1.323G
C1 4 0 30p
Eoutput 1 0 4 0 1
.ends small_signal_opamp

** Main Circuit
** signal source
Vi 3 0 AC 1V 0Degrees
Xopamp 1 0 2 small_signal_opamp
R1 3 2 1k
R2 2 1 1k ** Analysis Requests **
.AC DEC 5 0.1Hz 100MegHz *
* Output Requests **
.PRINT AC V(3) V(1)
.probe
.end
```