

SPICE Reference

Angelino Lefevers

March 14, 2018

Contents

Introduction

1	General Information about SPICE	1
1.1	What is SPICE?	1
2	Overview	2
2.1	Types of Circuit Analysis	2
2.2	Scaling Factors	2
2.3	Standard Components	2
2.4	SPICE Script Layout	4
3	General Script	5
3.1	Comments	5
3.2	Circuit Title	5
3.3	.END	5
3.4	Device Models	6
4	Standard Circuit Elements	6
	References	i

Introduction

This reference will compile a large amount of data and information of SPICE circuit software, a variety of distributions of SPICE simulation software, and a variety of distributions of SPICE graphical software. This reference will focus on the compilation and organization of this data and information. This reference will also act as an index of many resources which have been used to find this information, and will be notated in the bibliography of this reference. This reference is not purely written by the author notated on the title page and directly uses text and images from the resources noted in the bibliography section at the end.

1 General Information about SPICE

This section presents general information about the history and general use of SPICE.

1.1 What is SPICE?

SPICE is a powerful general purpose analog circuit simulator that is used to verify circuit designs and to predict the circuit behavior. This is of particular importance for integrated circuits. It was for this reason that SPICE was originally developed at the Electronics Research Laboratory of the University of California, Berkeley (1975), as its name implies:

Simulation **P**rogram for **I**ntegrated **C**ircuits **E**mphasis.

2 Overview

This section begins to present the things that SPICE are able to do and how it's able to do those things.

2.1 Types of Circuit Analysis

SPICE allows for a variety of types of simulation to be conducted on a circuit. Each types of analysis allows an engineer to verify and analyze the operation of a physical circuit. These analysis also help an engineer to identify inconsistencies between ideal circuit calculations and the operation of empirical circuitry.

- DC
- AC Small Signal
- Transient
- Pole Zero
- Noise
- Sensitivity
- Distortion
- Fourier
- Monte Carlo

2.2 Scaling Factors

SPICE uses a set of characters to scale the magnitudes which describe element and circuit characteristics. When a magnitude in spice is appended with one of these characters, it is scaled accordingly.

Suffix	Name	Factor
T	Tera	10^{12}
G	Giga	10^9
Meg	Mega	10^6
K	Kilo	10^3
mil	Mil	25.4×10^{-6}
m	milli	10^{-3}
u	micro	10^{-6}
n	nano	10^{-9}
p	pico	10^{-12}
f	femto	10^{-15}

Table 1: SPICE Scaling Factors

2.3 Standard Components

SPICE simulates circuits by using combinations of standard sets of components which are featured within the SPICE simulating software. SPICE circuitry can also use custom components created by 3rd parties to reflect the operation of virtually any piece of hardware, whether it is realizable physically or not. Each component in a SPICE circuit is represented by a name, the first letter of which describes the type of component. The types of components that can be implemented in a SPICE circuit are listed below with the preliminary letter which corresponds to each use.

Letter	Element Description
A	XSPICE Code Model
B	Behavioral Source
C	Capacitor
D	Diode
E	Voltage-Controlled Voltage Source
F	Current-Controlled Current Source
G	Voltage-Controlled Current Source
H	Current-Controlled Voltage Source
I	Current Source
J	Junction Field Effect Transistor (JFET)
K	Coupled (Mutual) Inductors
L	Inductor
M	Metal Oxide Field Effect Transistor (MOSFET)
N	Numerical Device for GSS
O	Lossy Transmission Line
P	Coupled Multiconductor Line (CPL)
Q	Bipolar Junction Transistor (BJT)
R	Resistor
S	Switch (Voltage Controlled)
T	Lossy Transmission Line
U	Uniformly Distributed RC Line
V	Voltage Source
W	Switch (Current Controlled)
X	Subcircuit
Y	Single Lossy Transmission Line (TXL)
Z	Metal Semiconductor Field Effect Transistor (MESFET)

Table 2: SPICE Elements

2.4 SPICE Script Layout

Below in Script 1 lies the pseudocode of a SPICE program. This illustrates how the general SPICE program is partitioned and where each functional element lies.

```
TITLE  
ELEMENTS  
.  
.  
CONTROL STATEMENTS  
OUTPUT STATEMENTS  
.END
```

Script 1: SPICE Program Layout (Pseudocode)

3 General Script

3.1 Comments

No script is complete without the ability to add comments into the original text. This allows further description of elements, functions, variables, and the ideas that go into implemented functionality. SPICE has two ways to insert comments into its code. The first way is illustrated in [2](#), this method comments the entire line of text after the `*` symbol. The other way to comment in SPICE is to comment the remainder of a line, this is done with the `$` symbol before the commented code.

```
* comment
```

Script 2: Line Commenting

```
R2 0 1 100 $ comment  
R3 1 2 200 $ comment
```

Script 3: End of Line Comment

3.2 Circuit Title

Each circuit description must begin with a title. The general form a title is described is illustrated below in [Script 4](#). The title can also be written as is without any sort of tag as displayed in [Script 5](#)

```
.TITLE title
```

Script 4: Declaring a Title with Tag TITLE

```
title
```

Script 5: Short Way to Declare a Title (Without Tag)

- *title* := Title of circuit

3.3 .END

Every circuit description must be finished by an **.END** tag, which signifies the end of the script. This end tag is illustrated below in [Script 7](#), and is as simple as it looks both there and in [Script 7](#). There are no arguments that go into this tag and is to be used as is.

```
.END
```

Script 6: End Script Tag

3.4 Device Models

`.MODEL modelName modelType (pname1 = pval1 pname2=pval2 ...)`

Script 7: End Script Tag

- *modelName* := Name of the model
- *modelType* := Type of model
- *pname* := Device Parameter
- *pval* := Value the parameter is to be set to

4 Standard Circuit Elements

References

- [1] Jan Van der Spiegel. Spice - a brief tutorial. <http://www.seas.upenn.edu/~jan/spice/spice.overview.html>.
- [2] Paolo Nenzi Holger Vogt, Marcel Hendrix. Ngspice users manual. <https://sourceforge.net/projects/ngspice/files/>.