

# SPICE Reference

Angelino Lefevers

March 13, 2018

# Contents

## Introduction

<b>1</b>	<b>General Information about SPICE</b>	<b>1</b>
1.1	What is SPICE? . . . . .	1
<b>2</b>	<b>Overview</b>	<b>2</b>
2.1	Types of Circuit Analysis . . . . .	2
2.2	Scaling Factors . . . . .	3
2.3	Standard Components . . . . .	3
2.4	SPICE Script Layout . . . . .	4

<b>References</b>	<b>i</b>
-------------------	----------

# 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	$25.4 \times 10^{-6}$
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.

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 STATEMENT  
ELEMENT STATEMENTS  
.  
.  
COMMAND (CONTROL) STATEMENTS  
OUTPUT STATEMENTS  
.END
```

Script 1: SPICE Program Layout (Pseudocode)

## References

- [1] Jan Van der Spiegel. Spice - a brief tutorial. <http://www.seas.upenn.edu/~jan/spice/spice.overview.html>.