Journal of
Semiconductor Devices and Circuits

ISSN: 2455-3379
Volume 11, Issue 3, 2024
September–December
DOI (Journal): 10.37591/JoSDC

https://journals.stmjournals.com/josdc

Research JoSDC

OrCAD PSpice Modified Switches for Electrical and Electronic Circuits

K. Bharath Kumar[1,]*

Abstract

Voltage-dependent switches and current-dependent switches are extensively used to obtain open/short circuits in various networks for various applications. PSpice/OrCAD software analog behavioral modeling options and Spice-dependent sources are used to obtain control-equivalent circuits for switches (PSpice built-in models) to be used for two types of applications. Different circuits (at DC, time domain), and Spice files are described to simulate circuits with analyses. The following features are available with this type of voltage-controlled/current-controlled switch. (1) Kirchhoff's voltage-current laws are used to obtain open/short circuits even after attaching a controlling voltage/current circuit to the main circuit. (2) The controlling circuit is attached at the desired NODE/across the branch of the main circuit. (3) Different networks can be connected or separated by this procedure without affecting the purpose for which they are separated or connected. The techniques described would find various uses, firstly, in analog behavioral modeling, and secondly in switching circuits. PSpice analog behavioral modeling and independent time domain, simulation with available models for pulse, piece-wise linear current/voltage generators. Normally, the Thevenin resistance is computed with DC or AC energy sources excitation shorting voltages and removing independent current sources. The Thevenin resistance for two types of circuits is determined by a conventional method using voltage/current pulse in the time domain, by making use of PSpice voltage and current switch. Two resistors with 1 Ohm, and -1 Ohm values are used in controlling the circuit.

Keywords: PSpice, OrCAD, voltage/current, Kirchhoff's current law, resistance

INTRODUCTION

A Spice model is used to describe an active or passive component of a network used in the simulation to mathematically determine the behavior of a circuit under consideration. Voltage-dependent and current-dependent switches are fundamental in modern circuit design, allowing networks to transition between open- and short-circuit states for various purposes. These switches are critical for systems that require precise control and consistent performance. The accurate modeling and simulation of these switches are essential for effective circuit design and analysis. This study focuses on leveraging the PSpice/OrCAD software, particularly its analog behavioral modeling capabilities and Spice-dependent sources, to create control-equivalent circuits for voltage- and current-controlled switches.

*__Author for Correspondence__
K. Bharath Kumar
E-mail: bbkrishnapuram3@yahoo.com

[1]Formerly Researcher in Oki Electric Company, R&D Division, Semiconductor and High-speed Devices, Tokyo, Japan

Received Date: November 15, 2024
Accepted Date: November 23, 2024
Published Date: December 03, 2024

Citation: K. Bharath Kumar. OrCAD PSpice Modified Switches for Electrical and Electronic Circuits. Journal of Semiconductor Devices and Circuits. 2024; 11(3): 28–46p.

OrCAD is an extensive set of electronic design automation (EDA) tools developed by Cadence Design Systems, designed to create and simulate electronic circuits. EDA involves the use of software tools to design and manufacture electronic systems including integrated circuits (ICs), printed circuit boards (PCBs), and other components. EDA tools assist engineers in automating processes, such as circuit design, simulation, verification, and layout, which enhance both precision and productivity. These tools play a key role in optimizing system performance, reducing errors, and shortening development cycles. EDA is crucial in sectors such as the consumer electronics, telecommunications, and automotive industries, where the complexity of electronic systems is rapidly increasing. Well-known EDA software packages include OrCAD, Cadence, Altium Designer, and Synopsys. It is widely used by engineers and designers for both analog and digital circuit development. Owing to its robust set of features, OrCAD plays a crucial role in the creation of PCBs, circuit schematics, and system-level designs.

A key component of OrCAD is OrCAD Capture, a schematic capture tool that enables users to create circuit diagrams by placing and connecting electronic components. This tool can handle both simple and complex designs, making it suitable for professionals across various levels of circuit design. Additionally, OrCAD Capture seamlessly integrates with other tools within the suite, facilitating simulation and layout tasks, which enhances the overall design workflow.

This research provides comprehensive details on circuit configurations and Spice file descriptions, illustrating the simulation of switching circuits under diverse conditions, including DC and time-domain scenarios. By applying Kirchhoff's voltage and current laws, the methodology ensures reliable open/short-circuit behavior even when control circuits are integrated at specific nodes or branches. This approach enables networks to be seamlessly connected or disconnected while maintaining their intended functionality.

This study explores two key applications of these techniques: analog behavioral modeling and the simulation of switching circuits. By utilizing PSpice's advanced features, such as time-domain simulations with pulse and piecewise linear signal generators, this research demonstrates the calculation of Thevenin resistance through traditional methods. Voltage and current switches are employed in various scenarios using controlling circuits with 1-Ohm and -1-Ohm resistors to ensure precise operation.

This work highlights the flexibility of PSpice/OrCAD software in addressing complex challenges in circuit design. The insights gained from this research are expected to contribute significantly to applications that require adaptable network configurations and dependable switching processes, thereby offering new possibilities for circuit analysis and design innovation.

In this paper, the design and modeling of a voltage/current-controlled switch for use in linear circuits requiring a controlling network are described.

An electronic switch is used to break or connect two nodes between which it is connected. It can connect or disconnect the conduction path [1–15]. An equivalent circuit model with controlled sources and resistors for a switch was described using Spice. The files describing the model applicable for both transient and steady-state analyses can be written using Spice.

```
S N++ N-- NK++ NK-- SNAME
. MODEL SNAME VSWITCH (RON= ROFF= VON= VOFF)
W N+ N- VN ++ WNAME
. MODEL WNAME ISWITCH (RON= ROFF= ION= IOFF)
```

In a circuit with resistors and independent voltage and current sources, a switch can be inserted using the switch model available in the PSpice simulation software. These switches are controlled by either voltage or current [16–20]. Two types of control circuits using Kirchhoff's current(voltage) laws were used in this study to obtain the control voltage (current).

Kirchhoff's laws, introduced by Gustav Kirchhoff in 1845, are foundational concepts in electrical circuit theory that aid the analysis of complex electrical circuits. These laws play a crucial role in understanding how the current and voltage are distributed across various components in a circuit.

Kirchhoff's current law (KCL) asserts that the total current flowing into a junction or node must be equal to the total current flowing out. This can be mathematically represented as follows:

$\sum I_{in}=\sum I_{out}$

The KCL is derived from the principle of charge conservation, which states that a charge cannot be created or destroyed. Simply put, this law ensures that the current entering a node is balanced by that leaving it. KCL is essential for analyzing circuits with multiple interconnected components, such as resistors or capacitors, and is used to set up equations to solve the unknown currents at each node.

Kirchhoff's voltage law (KVL) states that the sum of all voltage differences (or potential drops) around a closed loop in a circuit must equal zero. This is expressed as:

$\sum V=0$

KVL is based on the conservation of energy, meaning that the energy supplied by sources (such as batteries) within a closed loop is completely consumed by components (such as resistors or capacitors). The total energy supplied was equal to the total energy used. KVL is a valuable tool for determining unknown voltages or currents in a loop by setting up equations based on voltage changes across circuit elements.

Kirchhoff's current and voltage laws are vital for analyzing and solving both simple and complex electrical circuits, supporting the design and optimization of electronic devices.

Kirchhoff's KVL (there is no change in the node voltages if two voltage sources have the same value but opposite polarity) and KCL (there is no change in branch current if two current sources have the same value but different directions). The circuit voltages and currents were not altered, even if they were connected to the network.

CIRCUITS AND SIMULATION PROCEDURE

Two networks with pure resistance excited by DC sources (no sources) were considered. Figure 1(a) and (b) show the network with the switch incorporated (bias control) between the R and S terminals. Figure 1(a) and (b) shows the network with a switch incorporated (bias control) between R and S terminals.

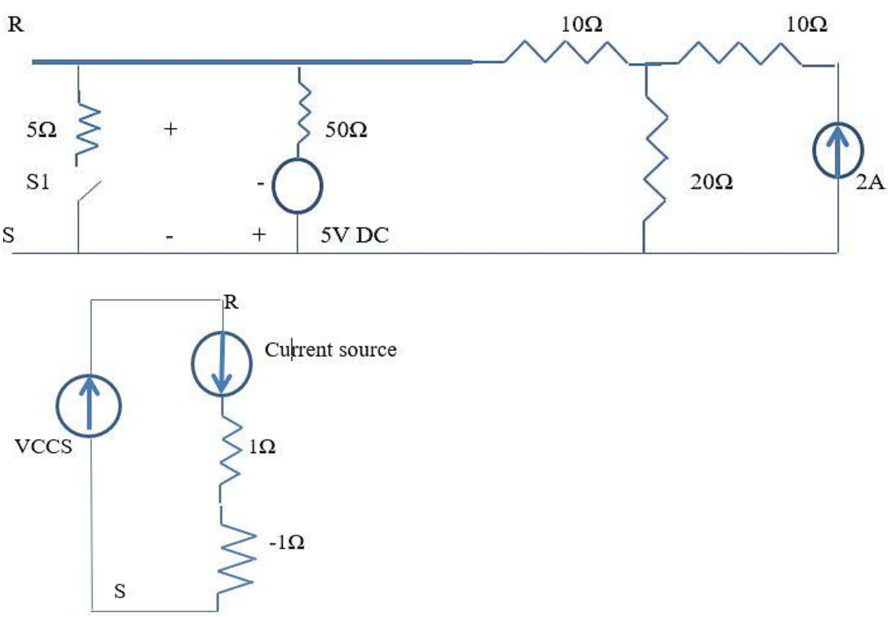

Figure 1. (a) A DC linear circuit incorporated with a switch with a controlling circuit across R-S terminals.

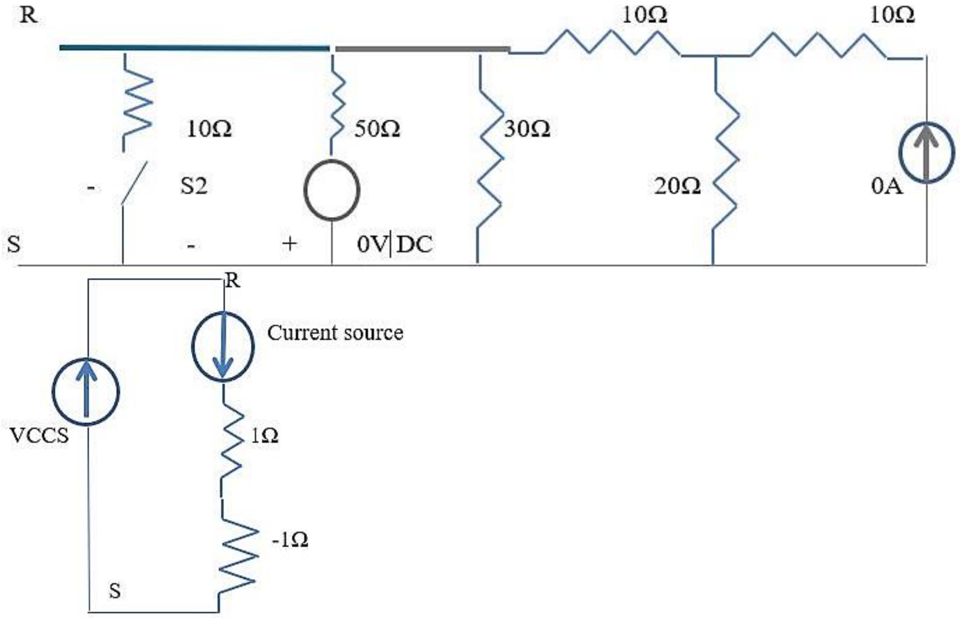

Figure 1. (b) A passive circuit incorporated with a switch across R-S terminals.

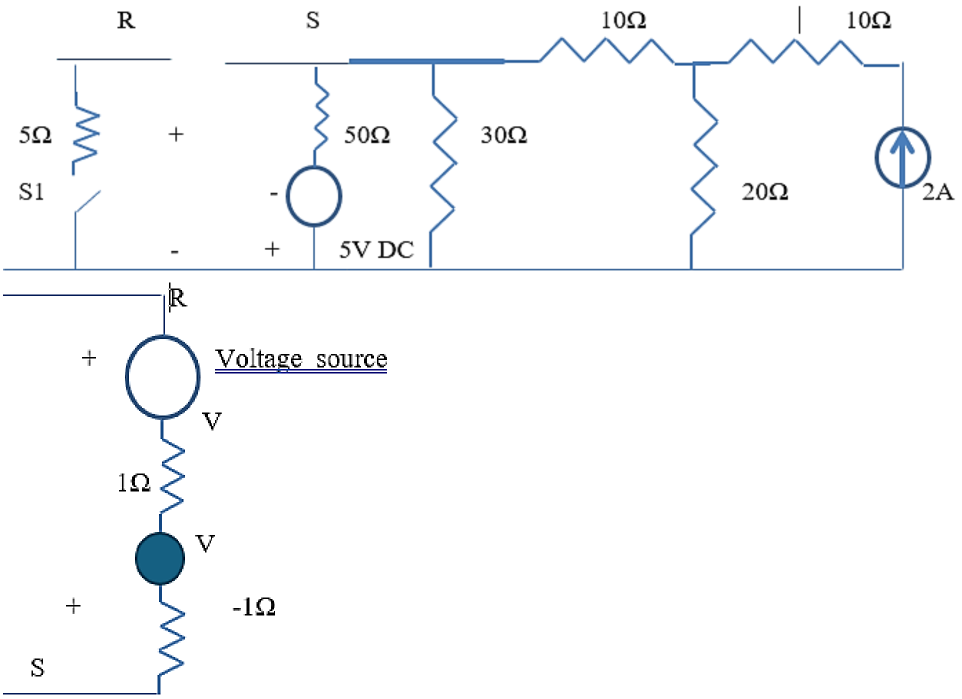

Figure 2. (a) A DC linear circuit incorporated with a switch with a controlling circuit across R-S terminals.

The controlling circuit that is used to turn ON/OFF the PSpice model consists of two current sources and two resistors of value 1 Ohm, -1 Ohm each. When connected to the main circuit, its contribution is zero, as is evident by Kirchhoff's current law.

A different type of controlling circuit for the switches between the R and S terminals is shown in Figures 2(a) and (b). The controlling circuit that is used to turn ON/OFF the PSpice model consists of two voltage sources and two resistors of 1 Ω and -1 Ω, respectively. When connected to the main circuit, its contribution is zero, as is evident by Kirchhoff's voltage law.

Journal of Semiconductor Devices and Circuits
Volume 11, Issue 3
ISSN: 2455-3379

Figure 2(b) describes a network used for the Thevenin resistance computation after removing independent current sources and shortening individual voltage sources. Two Thevenin resistances could be verified by connecting S between the R and S terminals using time-domain analysis, as it is a resistive circuit.

SIMULATION RESULTS

The following is a description of Spice/PSpice files used to simulate circuits with voltage/current-controlled switches (Figure 1(a) and (b) to Figure 2(a) and (b)). Table 1 describes the switch action with the control circuit (Figure 1(a)) using a DC voltage source. Table 2 describes the functioning of pulse current control, and Table 3 uses a piecewise linear source as the controlling voltage.

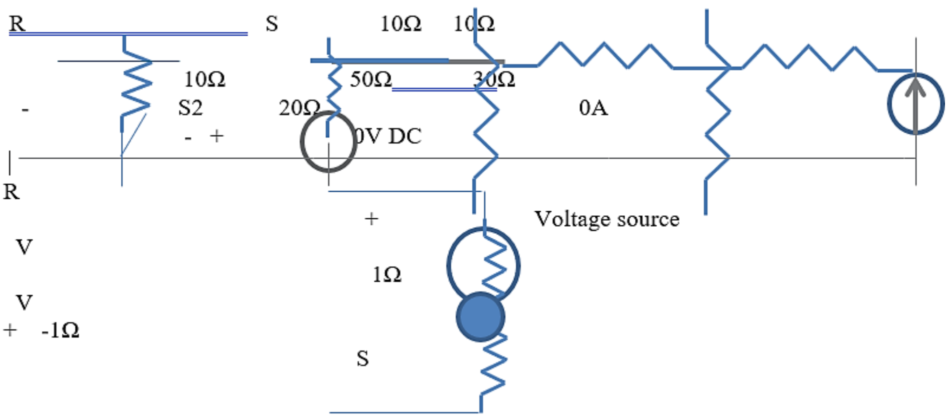

Figure 2. (b) A passive circuit incorporated with a switch across R-S terminals.

Table 1. PSpice circuit file for Figure 1(a) with DC excitation.

**** 11/12/24 17:46:19 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With circuit description
**** Circuit Description
****
*Controlling Circuit with Two Current Sources
*With Resistor
R12 1 2 5
R13 1 3 50
R14 1 4 10
R40 4 0 20
R45 4 5 10
IO5 0 5 DC 2
V03 3 0 DC -5V
S1 2 0 6 0 Sname
G01 0 1 6 0 0.5
I16 1 6 DC 100
R60 6 0 2
Model Sname Vswitch (RON=20 ROFF=1e09 VON=200 VOFF=0)
.op
.end
**** 11/12/24 17:46:19 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With model parameters
**** Voltage-Controlled Switch Model Parameters

****
SNAME
RON 20
ROFF 1.000000E+09
VON 200
VOFF 0
**** 11/12/24 17:46:19 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With model parameters
**** Small Signal Bias Solution Temperature = 27.000°C
****
Node Voltage Node Voltage Node Voltage Node Voltage
(1) 13.2140 (2) 10.5710 (3) -5.0000 (4) 22.1430
(5) 42.1430 (6) 200.0000
Voltage Source Currents
Name Current
V03 3.643E-01
Total power dissipation 1.82e+00 watts
**** 11/12/24 17:46:19 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With
**** Operating point information temperature = 27.000°C
****
**** Voltage-controlled Current Source
Name g01
I-source 1.000E+02
**** Voltage-Controlled Switches
NAME S1
Model Sname
I LOAD 5.29E-01
V LOAD 1.06E+01
R LOAD 2.00E+01
V CTRL 2.00E+02
Job Concluded
**** 11/12/24 17:46:19 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With model parameters
**** Job Statistics Summary

The same circuit described in Figure 1(a) is used in this example (Table 2), with the DC sources in the controlling network replaced by an independent pulse generator.

Table 2. PSpice circuit file for Figure 1(a) with pulse excitation.

**** 11/12/24 16:42:36 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With circuit description
**** Circuit description
****
*Controlling circuit with two current sources
*With resistor

R12 1 2 5
R13 1 3 50
R14 1 4 10
R40 4 0 20
R45 4 5 10
IO5 0 5 DC 2
V03 3 0 DC -5V
S1 2 0 6 0 Sname
G01 0 1 6 0 1
I16 1 6 Pulse (0 100V 0 0 0 20US 50US)
R60 6 0 1
.TRAN 10US 50US 5US
.Model Sname Vswitch (Ron=20 Roff=1e09 Von=100 Voff=0)
.PRINT TRAN V(1) V(2) V(3) V(4)
*.PROBE
.op
.end
**** 11/12/24 16:42:36 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With
**** Voltage-Controlled Switch Model Parameters
****
Sname
RON 20
ROFF 1.000000E+09
VON 100
VOFF 0
**** 11/12/24 16:42:36 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With
**** Small Signal Bias Solution Temperature = 27.000°C
****
Node voltage node voltage node voltage node voltage
(1) 23.1250 (2) 23.1250 (3) -5.0000 (4) 28.7500
(5) 48.7500 (6) 23.13E-12
Voltage source currents
Name Current
V03 5.625E-01
Total Power Dissipation 2.81e+00 Watts
**** 11/12/24 16:42:36 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With circuit description
**** Operating point information temperature = 27.000°C
****
**** Voltage-Controlled Current Sources
Name G01
I-source 2.313E-11
**** Voltage-Controlled Switches
Name S1

Model Sname
I LOAD 2.31E-08
V LOAD 2.31E+01
R LOAD 1.00E+09
V CTRL 2.31E-11
**** 11/12/24 16:42:36 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With circuit description
**** Initial transient solution temperature = 27.000°C
****
Node Voltage Node Voltage Node Voltage Node Voltage
(1) 23.1250 (2) 23.1250 (3) -5.0000 (4) 28.7500
(5) 48.7500 (6) 23.13E-12
Voltage source currents
Name Current
V03 5.625E-01
Total Power Dissipation 2.81e+00 Watts
**** 11/12/24 16:42:36 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With
**** Transient Analysis Temperature = 27.000°C
****
TIME V(1) V(2) V(3) V(4)
5.000E-06 2.312E+01 2.312E+01 -5.000E+00 2.875E+01
1.500E-05 1.321E+01 1.057E+01 -5.000E+00 2.214E+01
2.500E-05 1.321E+01 1.057E+01 -5.000E+00 2.214E+01
3.500E-05 2.312E+01 2.312E+01 -5.000E+00 2.875E+01
4.500E-05 2.313E+01 2.313E+01 -5.000E+00 2.875E+01
5.000E-05 2.313E+01 2.313E+01 -5.000E+00 2.875E+01
Job Concluded

It can be observed from Table 3 that the DC voltages are the same as those computed from Figure 1(a) without a switch connection between the R and S terminals, as when the switch is connected to the PWL source. Without the switch, the DC voltages are computed by assuming that the resistance between the R and S terminals is zero and infinity.

Table 3. PSpice circuit file for Figure 1(a) with piece-wise linear excitation.

****** 11/12/24 15:35:11 ****** PSpice Lite (October 2012) ****** ID# 10813 ******
Voltage-Controlled Switch With circuit description
**** Circuit description
****
*Controlling Circuit with Two Current Sources
*With Resistor, ramp Generator is Used
R12 1 2 5
R13 1 3 50
R14 1 4 10
R40 4 0 20
R45 4 5 10
IO5 0 5 DC 2

V03 3 0 DC -5V
S1 2 0 6 7 Sname
G01 0 1 6 7 1
I16 1 6 PWL(0 0 20US 0 22US 200V)
R60 6 7 1
R70 7 0 -1
.TRAN 10US 200US 5US
.Model Sname Vswitch (RON=20 ROFF=1e09 VON=200 VOFF=0)
.PRINT TRAN V(1) V(2) V(3) V(4) V(5)
.op
.end
**** 11/12/24 15:35:11 ****** PSpice Lite (October 2012) ****** ID# 10813 ***
Voltage-Controlled Switch With
**** Voltage-Controlled Switch Model Parameters
****
Sname
Ron 20
Roff 1.000000E+09
Von 200
Voff 0
**** 11/12/24 15:35:11 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With
**** Small Signal Bias Solution Temperature = 27.000°C
****
Node Voltage Node Voltage Node Voltage Node Voltage
(1) 23.1250 (2) 23.1250 (3) -5.0000 (4) 28.7500
(5) 48.7500 (6)-23.13E-24 (7)-23.13E-12
Voltage Source Currents
Name Current
V03 5.625E-01
Total power dissipation 2.81e+00 watts
**** 11/12/24 15:35:11 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With
**** Operating Point Information Temperature = 27.000°C
****
**** Voltage-Controlled Current Sources
NAME G01
I-SOURCE 2.313E-11
**** Voltage-Controlled Switches
Name S1
Model SNAME
I Load 2.31E-08
V LOAD 2.31E+01
R LOAD 1.00E+09
V CTRL 2.31E-11
**** 11/12/24 15:35:11 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With
**** Initial transient solution temperature = 27.000°C

****
Node Voltage Node Voltage Node Voltage Node Voltage
(1) 23.1250 (2) 23.1250 (3) -5.0000 (4) 28.7500
(5) 48.7500 (6)-23.13E-24 (7)-23.13E-12
Voltage source currents
Name Current
V03 5.625E-01
Total Power Dissipation 2.81E+00 WATTS
**** 11/12/24 15:35:11 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Voltage-Controlled Switch With
**** Transient analysis temperature = 27.000°C
****
TIME V(1) V(2) V(3) V(4) V(5)
5.000E-06 2.313E+01 2.313E+01 -5.000E+00 2.875E+01 4.875E+01
1.500E-05 2.313E+01 2.313E+01 -5.000E+00 2.875E+01 4.875E+01
2.500E-05 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
3.500E-05 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
4.500E-05 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
5.500E-05 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
6.500E-05 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
7.500E-05 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
8.500E-05 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
9.500E-05 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
1.050E-04 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
1.150E-04 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
1.250E-04 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
1.350E-04 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
1.450E-04 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
1.550E-04 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
1.650E-04 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
1.750E-04 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
1.850E-04 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
1.950E-04 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
2.000E-04 1.321E+01 1.057E+01 -5.000E+00 2.214E+01 4.214E+01
Job Concluded

The Thevenin resistance (Table 4) was observed to be the same as that obtained when the Switch is ON/OFF (two values) in the time domain, similar to that calculated with conventional techniques without the switch.

Table 4. Voltage-controlled Thevenin switch in Figure 1(b).

****** 11/14/24 12:52:50 ****** PSpice Lite (October 2012) ****** ID# 10813 ******
Switch for Figure 1(b) Voltage-Controlled Thevenin Switch
**** Circuit Description
****
*Two Different Values of Thevenin Resistances For
*Switch On/Off Positions
*V(1) is the Thevenin Resistance

R12 1 2 10
R13 1 3 50
R10 1 0 30
R14 1 4 10
R40 4 0 20
R45 4 5 10
IO5 0 5 DC 0
V03 3 0 DC 0
S1 2 0 6 7 SNAME
GI01 0 1 6 7 1
I16 1 6 Pulse (0 200 0 0 0 20U 50US)
R67 6 7 1
R70 7 0 -1
I01 0 1 DC 1
.DC LIN I16 50 200 50
.Model Sname Vswitch (RON=20 ROFF=1e09 VON=200 VOFF=0)
.TRAN 1US 30US 5US
.PRINT TRAN V(1)
.op
.end
**** 11/14/24 12:52:50 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 1(b) Voltage-Controlled Thevenin Switch
**** Voltage-Controlled Switch Model Parameters
****
Sname
RON 20
ROFF 1.000000E+09
VON 200
VOFF 0
**** 11/14/24 12:52:50 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 1(b) Voltage-controlled Thevenin Switch
**** Small Signal Bias Solution Temperature = 27.000°C
****
Node Voltage Node Voltage Node Voltage Node Voltage
(1) 11.5380 (2) 11.5380 (3) 0.0000 (4) 7.6923
(5) 7.6923 (6)-11.54E-24 (7)-11.54E-12
Voltage Source Currents
NAME CURRENT
V03 2.308E-01
Total Power Dissipation 0.00E+00 WATTS
**** 11/14/24 12:52:50 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 1(b) voltage-controlled Thevenin Switch
**** Operating point information temperature = 27.000°C
****
**** Voltage-Controlled Current Sources
Name GI01
I-source 1.154E-11
**** Voltage-Controlled Switches

Name S1
Model Sname
I load 1.15E-08
V load 1.15E+01
R load 1.00E+09
V CTRL 1.15E-11
**** 11/14/24 12:52:50 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 1(b) voltage-controlled Thevenin Switch
**** Initial Transient Solution Temperature = 27.000°C
****
Node Voltage Node Voltage Node Voltage Node Voltage
(1) 11.5380 (2) 11.5380 (3) 0.0000 (4) 7.6923
(5) 7.6923 (6)-11.54E-24 (7)-11.54E-12
Voltage Source Currents
Name Current
V03 2.308E-01
Total Power Dissipation 0.00e+00 Watts
**** 11/14/24 12:52:50 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 1(b) Voltage-Controlled Thevenin Switch
**** Transient Analysis Temperature = 27.000°C
****
TIME V(1)
5.000E-06 8.333E+00
6.000E-06 8.333E+00
7.000E-06 8.333E+00
8.000E-06 8.333E+00
9.000E-06 8.333E+00
1.000E-05 8.333E+00
1.100E-05 8.333E+00
1.200E-05 8.333E+00
1.300E-05 8.333E+00
1.400E-05 8.333E+00
1.500E-05 8.333E+00
1.600E-05 8.333E+00
1.700E-05 8.333E+00
1.800E-05 8.333E+00
1.900E-05 8.333E+00
2.000E-05 8.333E+00
2.100E-05 8.333E+00
2.200E-05 1.154E+01
2.300E-05 1.154E+01
2.400E-05 1.154E+01
2.500E-05 1.154E+01
2.600E-05 1.154E+01
2.700E-05 1.154E+01
2.800E-05 1.154E+01
2.900E-05 1.154E+01
3.000E-05 1.154E+01
Job Concluded

Table 5 shows an independent voltage source (DC) with a dependent voltage source for switching control.

Table 5. PSpice circuit for Figure 2(a) with DC switch.

**** 11/14/24 14:08:38 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 2(a) Voltage-controlled Switch
**** Circuit description
****
R12 1 2 5
R34 3 4 50
R30 3 0 30
R35 3 5 10
R50 5 0 20
R56 5 6 10
IO5 0 6 DC 2
V03 4 0 DC -5V
S1 2 0 1 7 Sname
V17 1 7 DC 200
E89 8 9 1 7 -1
R78 7 8 1
R93 9 3 -1
.Model Sname Vswitch (RON=20 ROFF=1e09 VON=200 VOFF=0)
.op
.end
**** 11/14/24 14:08:38 ****** PSpice Lite (October 2012) ****** ID# 10813 ***
Switch for Figure 2(a) Voltage-Controlled Switch
**** Voltage-Controlled Switch Model Parameter
*** SNAME
RON 20
ROFF 1.000000E+09
VON 200
VOFF 0
**** 11/14/24 14:08:38 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 2(a) Voltage-controlled Switch
**** Small signal bias solution temperature = 27.000°C
****
Node voltage node voltage node voltage node voltage
(1) 9.7368 (2) 7.7895 (3) 9.7368 (4) -5.0000
(5) 19.8250 (6) 39.8250 (7) -190.2600 (8) -189.8700
(9) 10.1260
Voltage source currents
Name Current
V03 2.947E-01
V17 -3.895E-01
Total Power Dissipation 7.94e+01 Watts
**** 11/14/24 14:08:38 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 2(a) Voltage-controlled Switch
**** operating point information temperature = 27.000°C

****
**** Voltage-controlled voltage sources
Name E89
V-source -2.000E+02
I-SOURCE -3.895E-01
**** Voltage-Controlled Switches
Name S1
Model Sname
I Load 3.89E-01
V Load 7.79E+00
R Load 2.00E+01
V Ctrl 2.00E+02
Job Concluded

Table 6 shows the same independent voltage source (DC), which makes the switch OFF, together with a voltage-dependent voltage source.

Table 6. PSpice circuit file for Figure 2(a) with DC (off) excitation.

**** 11/14/24 14:11:58 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 2(a) Voltage-controlled Switch
**** Circuit description
R12 1 2 5
R34 3 4 50
R30 3 0 30
R35 3 5 10
R50 5 0 20
R56 5 6 10
IO5 0 6 DC 2
V03 4 0 DC -5V
S1 2 0 1 7 Sname
V17 1 7 DC 100
E89 8 9 1 7 -1
R78 7 8 1
R93 9 3 -1
.Model Sname Vswitch (RON=20 ROFF=1e09 VON=200 VOFF=0)
.op
.end
**** 11/14/24 14:11:58 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 2(a) Voltage-Controlled Switch
**** Voltage-Controlled Switch Model Parameters
****
Sname
RON 20
ROFF 1.000000E+09
VON 200
VOFF 0

**** 11/14/24 14:11:58 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 2(a) voltage-controlled Switch
**** Small signal bias solution temperature = 27.000°C
****
Node Voltage Node Voltage Node Voltage Node Voltage
(1) 14.2300 (2) 14.2290 (3) 14.2300 (4) -5.0000
(5) 22.8200 (6) 42.8200 (7) -85.7700 (8) -85.7700
(9) 14.2300
Voltage Source Currents
Name Current
V03 3.846E-01
V17 -1.006E-04
Total power dissipation 1.93e+00 watts
**** 11/14/24 14:11:58 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 2(a) Voltage-controlled Switch
**** Operating point information temperature = 27.000°C
****
**** Voltage-Controlled Voltage Sources
NAME E89
V-source -1.000e+02
I-source -1.006e-04
**** Voltage-Controlled Switches
Name S1
Model SNAME
I Load 1.01E-04
V Load 1.42E+01
R Load 1.41E+05
V Ctrl 1.00E+02
Job Concluded

The Thevenin resistance was calculated for Figure 2(b) using an independent voltage pulse in the control circuit. The calculated values were verified using conventional methods. The PSpice file (Thevenin) for Figure 2(b), the pulse switch is shown in Table 7.

Table 7. PSpice file (Thevenin) for Figure 2(b) with pulse switch.

**** 11/14/24 14:58:35 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 2(b) Voltage-controlled Switch
**** Circuit description
****
*Thevenin resistance in on-off states
R12 1 2 10
R34 3 4 50
R30 3 0 30
R35 3 5 10
R50 5 0 20

R56 5 6 10
IO5 0 6 DC 0
V03 4 0 DC 0
I0 0 1 DC 1
S1 2 0 1 7 SNAME
V-Pulse 1 7 Pulse (0 200 0 0 0 20US 50US)
E89 8 9 1 7 -1
R78 7 8 1
R93 9 3 -1
.TRAN 5US 50US 5US
.Model Sname Vswitch (Ron=20 Roff=1e09 Von=200 Voff=0)
.Print Tran V(1)
.op
.end
**** 11/14/24 14:58:35 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 2(b) Voltage-controlled Switch
**** Voltage-controlled switch model parameters
****
Sname
RON 20
ROFF 1.000000E+09
VON 200
VOFF 0
**** 11/14/24 14:58:35 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 2(b) Voltage-controlled Switch
**** Small signal bias solution temperature = 27.000°C
****
Node Voltage Node Voltage Node Voltage Node Voltage
(1) 11.5380 (2) 11.5380 (3) 11.5380 (4) 0.0000
(5) 7.6923 (6) 7.6923 (7) 11.5380 (8) 10.5380
(9) 10.5380
Voltage Source Currents
NAME CURRENT
V03 2.308E-01
V-Pulse 1.000E+00
Total Power Dissipation 0.00E+00 WATTS
**** 11/14/24 14:58:35 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 2(b) Voltage-controlled Switch
**** Operating point information temperature = 27.000°C
****
**** Voltage-controlled Voltage Sources
NAME E89
V-source 0.000E+00
I-source 1.000E+00

**** Voltage-Controlled Switches
Name S1
Model Sname
I Load 1.15E-08
V Load 1.15E+01
R Load 1.00E+09
V CTRL 0.00E+00
**** 11/14/24 14:58:35 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 2(b) Voltage-controlled Switch
**** Initial transient solution temperature = 27.000 de
****
Node voltage node voltage node voltage node voltage
(1) 11.5380 (2) 11.5380 (3) 11.5380 (4) 0.0000
(5) 7.6923 (6) 7.6923 (7) 11.5380 (8) 10.5380
(9) 10.5380
Voltage source currents
Name Current
V03 2.308E-01
V-Pulse 1.000E+00
Total Power Dissipation 0.00e+00 Watts
**** 11/14/24 14:58:35 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Switch for Figure 2(b) Voltage-controlled Switch
**** Transient analysis temperature = 27.000°C
****
Time V(1)
5.000E-06 8.333E+00
1.000E-05 8.333E+00
1.500E-05 8.333E+00
2.000E-05 8.333E+00
2.500E-05 8.333E+00
3.000E-05 1.154E+01
3.500E-05 1.154E+01
4.000E-05 1.154E+01
4.500E-05 1.154E+01
5.000E-05 1.154E+01
Job Concluded

In Tables 1–3, the DC-independent sources can be changed using a DC sweep, and the output results for Figure 1(a) when the switch is excited with DC-, pulse-, and PWL-independent sources.

Tables 5 and 6 describe the results for Figure 2(a) when the switch is turned ON/OFF by the DC voltage source controlling network, and the independent sources in the controlling network can also be replaced by Pules and PWL sources. In Table 7, the Thevenin resistance calculation with the pulse voltage generator in the on/off states can be replaced by a DC-independent source with the Sweep generator option.

Challenges and Future Scope

The implementation and use of voltage-dependent and current-dependent switches face various challenges, even though they are highly versatile in enabling open or short circuits in networks. One major difficulty is achieving precise control of these switches when integrating control circuits at specific nodes or branches, without causing unwanted interference. The accurate simulation of such configurations in PSpice/OrCAD, especially for complex networks and time-domain analysis, requires careful modeling and rigorous validation to ensure system stability and reliability.

Another challenge involves maximizing the potential of PSpice analog behavioral modeling and Spice-dependent sources for different applications. Creating and optimizing circuits with control components, such as resistors with values of 1 Ohm and -1 Ohm, while adhering to Kirchhoff's voltage and current laws, requires specialized knowledge and iterative improvements. Additionally, calculating the Thevenin resistance using conventional methods in various scenarios adds complexity, particularly when dealing with dynamic or real-time network behavior.

The future of research and development in this area has immense potential. Enhancing the built-in models of PSpice/OrCAD to make voltage- and current-controlled switches more user-friendly and adaptive could simplify the design process. Incorporating machine learning to automate the tuning of switch parameters can further enhance performance in dynamic environments.

Expanding these techniques to emerging sectors, such as renewable energy, advanced communication systems, and electric vehicle technology, presents promising opportunities. Developing more effective approaches to simulate intricate circuits with multiple interdependent components could help address current challenges. Additionally, exploring hybrid modeling methods that combine PSpice/OrCAD with other simulation tools could improve accuracy and efficiency.

By addressing the existing limitations and adopting advanced computational and analytical techniques, this field has significant potential to drive progress in circuit design and analysis.

CONCLUSIONS

Two different linear circuits with pure resistors and independent energy sources were simulated with switch models using a PSpice. Two different types of control networks are described to make the switch on/off. Both control networks use Kirchhoff's current/voltage laws. Three different excitations for current (pulse, DC, Ramp) and two (DC and pulse) were used in the control networks. The Thevenin resistances in the ON/OFF states are determined in the time domain for two linear (resistor) networks with incorporated switches. The results obtained were verified using other simulation packages. Different networks with temperature-dependent resistors can also be analyzed.

REFERENCES

1. Xu J, Yu J. Equivalent circuit models of switches for SPICE simulation. Electronics Letters. 1988;24:437–438. DOI: 10.1049/el:19880296.
2. Nagel LW, Pederson DO. SPICE (Simulation Program with Integrated Circuit Emphasis) [Technical Report]. University of California Berkeley. UCB/ERL M382.
3. Nagel LW. SPICE2: A computer program to simulate semiconductor circuits. UCB/ERL, Vol. M520. [Technical Report]. University of California Berkeley. 1975.
4. Prigozy S. Novel applications of SPICE in engineering education. IEEE Transactions on Education. 1989;32:35–38. DOI: 10.1109/13.21160.
5. Berkeley UC. SPICE Version 2G User's Guide. USA. 1975.
6. Bharath Kumar K. Multi two port parameter simulation using Pspice. [Technical Report]. Semiconductor Research Laboratory. Oki Electric Industry. 1990.
7. Rashid MH, Rashid MH. SPICE for Circuits and Electronics Using PSpice. New Delhi: Prentice Hall; 1995.

8. Keown J. OrCAD PSpice and Circuit Analysis. New Delhi: Prentice Hall; 2000.
9. Du H, Gorcea D, So PPM, Hoefer WJR. A SPICE analog behavioral model of two-port devices with arbitrary port impedances based on the S-parameters extracted from time-domain field responses. International Journal of Numerical Modelling: Electronic Networks, Devices and Fields. 2008;21:77–90. DOI: 10.1002/jnm.670.
10. Van Valkenburg ME. Chapter IX. Network Analysis. 3rd ed. New Delhi: Prentice Hall; 1976. p. 259–261.
11. Nahin PJ. The Science of Radio: With MATLAB® and Electronics Workbench® Demonstrations. New York: Springer Science+Business Media; 2001.
12. Vladimirescu A, Zhang K, Newton A, Pederson A, Sangiovanni-Vincentelli (1981). SPICE Version 2G User’s Guide. [online] Available from: http://www.ece.umn.edu/~harjani/courses/common/spice2G6.pdf
13. Kumar KB. (2022). Techniques for using SPICE to derive Thevenin and Norton equivalent circuits [Online]. EDN. Available from: https://www.edn.com/techniques-for-using-spice-to-derive-thevenin-and-norton-equivalent-circuits/
14. Jin LM, Chan SP. A unified and efficient approach for determining Thevenin (Norton) equivalent circuits. IEEE Trans Educ. 1989;32(3):408–10. DOI: 10.1109/13.34184.
15. Holder ME. Thevenin’s theorem and a black box. IEEE Trans Educ. 2009;52(4):573–5. DOI: 10.1109/TE.2008.2010985.
16. Hongyang Z. Discussion on the Thevenin’s theorem and Norton’s theorem. Proceedings of 2011 International Conference on Electronic & Mechanical Engineering and Information Technology, Harbin, China. 2011. pp. 520-522. DOI: 10.1109/EMEIT.2011.6022970.
17. Attia JO. Electronics and Circuit Analysis Using MATLAB. Boca Raton, FL: CRC Press; 2018. DOI: 10.1201/9781315275727.
18. Yang P. Circuit simulation and modeling. IEEE Circuits Devices Mag. 1987;3(5):36–44. DOI: 10.1109/MCD.1987.6323156.
19. Kumar K. Inverse ABCD parameter determination using Spice. Int J Analog Integr Circuits. 2017;3:14–19.
20. Lindigkeit R, Boehme A, Eiserloh I, Luebbecke M, Wiggermann M, Ernst L, et al. Spice: A never-ending story? Forensic Sci Int. 2009;191:58–63. DOI: 10.1016/j.forsciint.2009.06.008. PubMed: 19589652.

