International Journal of
Microelectronics and Digital Integrated Circuits
https://ecc.journalspub.info/index.php?journal=JMDIC

ISSN: 2456-3986
Volume 10, Issue 2, 2024
DOI (Journal): 10.37628/IJMDIC

Review IJMDIC

Polynomial Thevenin Resistance Networks for Spice Simulation

K. Bharath Kumar*

Abstract

Two networks with resistors and independent sources are considered to compute Thevenin resistance. A different network is considered where the Thevenin resistance from the previous independent networks is to be incorporated (in a polynomial representation). Different methods using Spice/Pspice software to obtain the above representation are described. SPICE/OrCAD software is used to obtain equivalent circuits for Thevenin equivalent network parameters used in polynomial form are described and other different networks with these models at DC are used in simulation. Here, the individual parameters from a different Thevenin/Norton equivalent are implemented. New techniques to obtain Thevenin and Norton impedances are described with examples. Dependent voltage sources are used to describe independent energy sources, facilitating their application. Spice program to obtain impedances with a single sub-circuit description, eliminating to make zeros (current and voltages) in the circuit under consideration for Thevenin/Norton Impedance determination. Every independent voltage/current source need not be short/open circuited. Both cases (i) DC sources and pure resistors (ii) AC sources, with passive elements, are considered. The transfer function (TF command) is used for the above circuits, and the results are verified. A Thevenin equivalent circuit can be used in place of a circuit with resistors and separate sources of voltage and current. Both circuit analysis and simulation can benefit from this substitution.

Keywords: Analog passive network theory, non- linear circuits, complex variables, circuit analysis and modelling, solid state circuits, microwave circuits, VLSI technology

INTRODUCTION

Circuit design requires different reliable methods to predict and evaluate performance. Although experimental laboratory measurements are useful, because of the complexity and size of the circuits in modern integrated circuits, computer-aided analyses are essential. It permits (a) to simulate various effects due to variation of circuit elements like resistors, (b) the variation effect on performance enhancement and degradation, (c) evaluation of non-linear elements in circuit, and (d) adjustment of circuit elements for a desired behavior. SPICE (semiconductor modelling with integrated circuit emphasis) is used to simulate operating conditions of devices, time-domain response, and frequency response with small-signal models. Different types of analyses, like DC, AC, and noise, can be performed. Here, the DC analyses option is used with the analog behavioral modeling option available with Pspice. Accurately determining Thevenin and Norton equivalent resistances and impedances is essential for simplifying the analysis and design of complex electrical circuits. Conventional approaches to calculating these parameters often require time-consuming manual computations or rely on idealized assumptions that may not reflect real-world conditions. This study investigates the use of advanced circuit simulation

***Author for Correspondence**
K. Bharath Kumar
E-mail: bbkrishnapuram3@yahoo.com

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

Received Date: November 20, 2024
Accepted Date: November 29, 2024
Published Date: December 10, 2024

Citation: K. Bharath Kumar. Polynomial Thevenin Resistance Networks for Spice Simulation. International Journal of Microelectronics and Digital Integrated Circuits. 2024; 10(2): 22–36p.

© STM Journals 2024. All Rights Reserved

tools, specifically SPICE and OrCAD software, to calculate Thevenin resistance and integrate various network elements into polynomial models.

The research examines two distinct networks composed of resistors and independent sources, offering a comprehensive method for determining Thevenin resistance while accounting for the influence of dependent voltage sources. These sources are employed to represent independent energy sources, enabling the use of SPICE simulations to calculate impedances without resorting to traditional short-circuit or open-circuit techniques for each source. Additionally, the study considers both DC and AC scenarios, incorporating a diverse range of passive components and introducing innovative methods for calculating Thevenin and Norton impedances, with results verified through simulation. The research utilizes the transfer function (TF) command in SPICE simulations to demonstrate effective techniques for calculating equivalent circuits, providing valuable insights into the design and analysis of electrical networks. The paper also explores how these simulation tools simplify traditional processes, such as the need for manually creating zeros in the circuit, thereby improving the accuracy and ease of determining impedance in both linear and non-linear circuit models. The Thévenin equivalent circuit can be used in place of a circuit with resistors and separate sources of voltage and current. Both circuit modeling and circuit analysis can benefit from substitution [1–10]. An ideal independent voltage source coupled in series with a resistor is the description of the Thévenin equivalent circuit. In this work, few networks are employed. Thevenin equivalent parameters (open circuit voltage and resistance) represented as polynomial functions of different networks are simulated by the SPICE software program. The small-signal transfer function option of PSpice is used to verify the obtained results. After using the DC command to calculate all voltages and currents, the (.TF) command is used for verification. A new method is applied with the Spice program to obtain Thevenin/Norton impedance. It uses dependent sources, replacing all independent voltage/current sources. Two different configurations are given as examples. The results obtained were verified by conventional techniques. In this method, there is no need to short (open) independent voltage/current energy sources. Using the LAPLACE option, the behavioral modeling of the Pspice simulation program can be utilized to obtain equivalent circuits from the conversion methods of T-π (star-delta) for impedances circuits for Thevenin impedances. By employing Pspice computer simulations to acquire low AC values of voltages and currents at different nodes and branches, the conversions could be confirmed for two different circuit types. It is possible to determine and validate the DC response to the circuit with the necessary Thevenin resistance by cascading passive circuits with common emitter, common base, and common collector circuits from fake data to simulate non-linear models. The approach and process outlined here might be beneficial for currently accessible circuit simulation software, such as OrCAD/Spice, which isn't always at low frequencies.

Voltages or currents might be the output variables or controlling variables in the SPICE description of polynomial sources. The rules for describing the four types of polynomials in Spice are:

i. The number of dimensions is half the number of controlling nodes for voltage described as polynomial.
ii. The number of dimensions is equal to the number of controlling sources for current described polynomial.

The VALUE option in analog behavioral modelling can be specified by the following statements in Spice netlist.

Exxx n+ n- VALUE = {expression} or

Gxxx n+ n- VALUE = {expression}

The expression in the above specification is effectively used to obtain polynomial description of Thevenin dc resistance with available voltages and currents.

© STM Journals 2024. All Rights Reserved

CIRCUITS AND SIMULATION

Figure 1(a) and (b) shows a network and its equivalent to compute Thevenin resistance between R hand S terminals. It consists of passive resistive elements, independent DC voltage and current sources. This circuit is described in Table 1(a). The polynomial with this Thevenin resistance is also shown in this Table1.

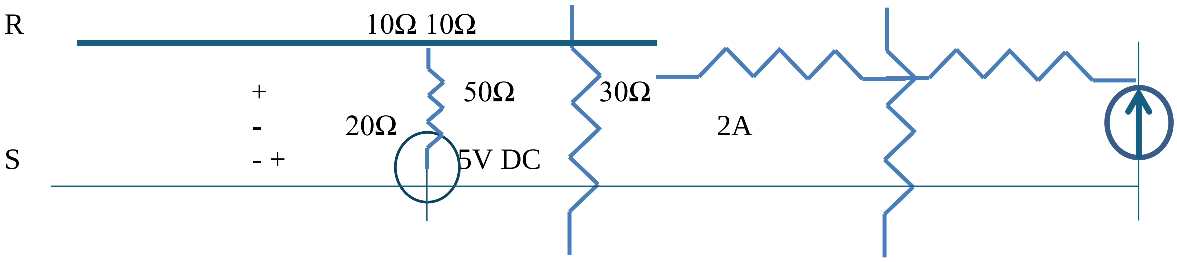

Figure 1(a). A DC linear circuit whose Thevenin equivalent is across R-S terminals.

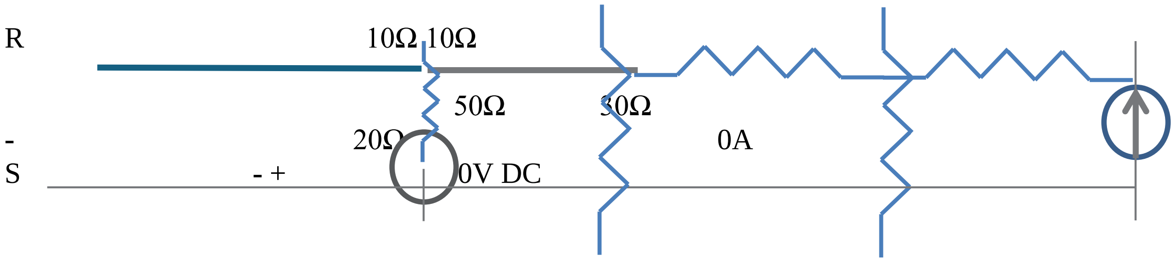

Figure 1(b). Equivalent circuit for Thevenin resistance across R-S terminals.

Figure 2(a) and (b) shows another network with resistances and independent energy sources and its equivalent circuit to obtain Thevenin/Norton resistance, between terminals R and S.

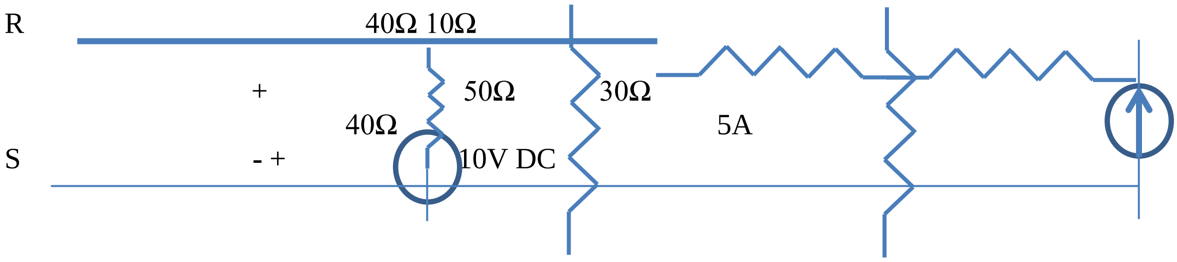

Figure 2(a). Linear DC network whose Thevenin parameters are across R_S terminals.

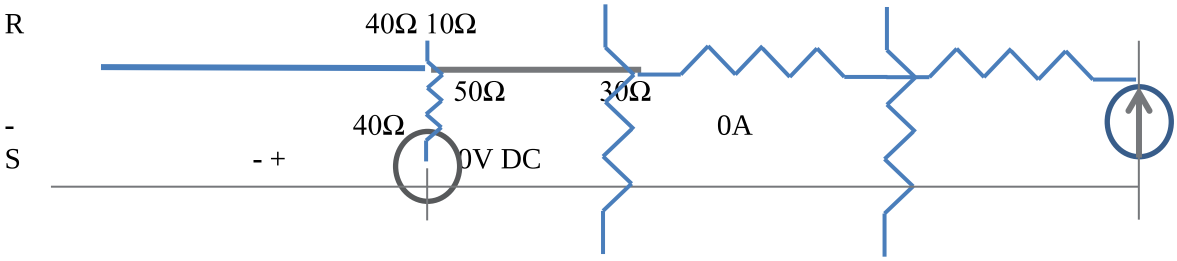

Figure 2(b). Equivalent circuit (second example) for Thevenin resistance across R-S terminals.

A circuit (Figure 3) shows with polynomial Thevenin/Norton resistance (Figure 1(b) and 2(b)) implemented.

© STM Journals 2024. All Rights Reserved

The SPICE/PSpice file to simulate Figure 3 is given in Tables 1–2 and a different Thevenin polynomial (RT+RT*RT) is incorporated into Table 3.

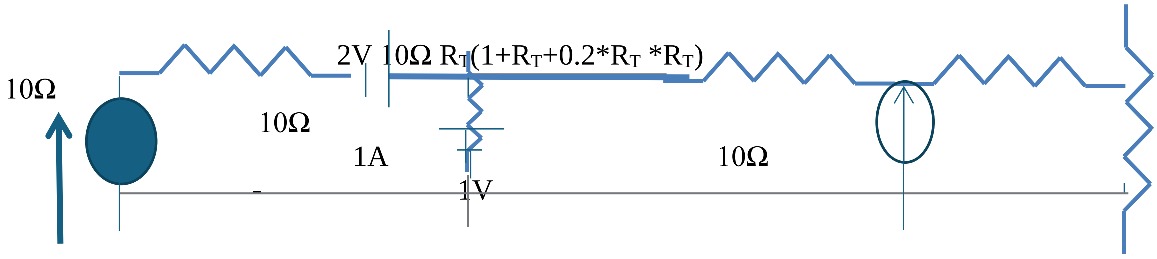

Figure 3. A Network implementing polynomial Thevenin equivalent parameters.

In the next section the SPICE/Pspice simulation files to obtain results for two different circuits with polynomial functions of Thevenin resistances (Figures 1 and 2).

Three different methods to obtain polynomial functions with Thevenin resistances are described, (a) Using SPICE polynomial dependent sources, (b) using VALUE option of analog behavioral modeling of Pspice, and (c) in Table 3, Thevenin resistance together with square of it are obtained by a different method. All the results are verified by conventional methods.

SIMULATION RESULTS

The following are description of SPICE/Pspice files. In Table 1(a) the circuit Figure 1(a) and (b) are described (first part) to obtain Thevenin equivalent circuit parameters. The results in Table 1(a) describe realization of polynomial equivalent ($R_T + R_T$**2) and its verification at the end. Thevenin resistance calculation with zero sources is described in second part. The remaining part describes polynomial Thevenin resistance and implementation. Small signal bias solution, current through various sources are also given. In Table 1(b) the circuit Figure 2(a) and (b) is described and the same polynomial is realized and implemented and the network where it is implemented is described.

Table 1(a). Circuit file for Figure 1 with SPICE simulation.

****** 10/09/24 12:59:10 ****** PSpice Lite (October 2012) ****** ID# 10813 ******
A Network for Polynomial Thevenin Parameters
****** Circuit Description**
****
R12 1 2 50
V20 2 0 DC 5
R10 1 0 30
R13 1 3 10
R30 3 0 20
R34 3 4 10
I04 0 4 DC 2
***Above Circuit Representation for Thevenin Resistance**
R50 5 0 50
R550 5 0 30
R56 5 6 10
R60 6 0 20
R67 6 7 10
R70 7 0 1E20
I05 0 5 DC 1

***Implementation of Polynomial Equivalent**
I08 0 8 DC 3
R89 8 9 10
V109 10 9 2
R1011 10 11 10
V110 11 0 DC 1
R1012 10 15 10
I012 0 15 DC 1
R130 13 0 10
***Realization of Polynomial Resistance Between Nodes (12) and (13)**
V1512 15 12
E1213 12 13 POLY (2) 14 0 16 0 0 0 0 0 0 1
GO16 0 16 POLY (1) 5 0 0 1 1 0.2
R160 16 0 1
FO14 0 14 V1512 1
R140 14 0 1
***Verification of Resistance Between Nodes (12) and (13)**
G018 0 18 POLY (2) 12 0 13 0 0 1 -1
R180 18 0 1
G017 0 17 POLY (2) 17 0 19 0 0 0 0 0 1
G170 17 0 18 0 1
FIV1512 0 19 V1512 1
R190 19 0 1
R170 17 0 1E20
.OP
.end
****** 10/09/24 12:59:10 ****** PSpice Lite (October 2012) ****** ID# 10813 ******
A Network for Polynomial Thevenin Parameters
****** Small Signal Bias Solution Temperature = 27.000 DEG C**
****
NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE
(1) 16.5380 (2) 5.0000 (3) 24.3590 (4) 44.3590
(5) 11.5380 (6) 7.6923 (7) 7.6923 (8) 67.9420
(9) 37.9420 (10) 39.9420 (11) 1.0000 (12) 48.8830
(13) 1.0583 (14) .1058 (15) 48.8830 (16) 451.9100
(17) 451.9100 (18) 47.8250 (19) .1058
VOLTAGE SOURCE CURRENTS
NAME CURRENT
V20 2.308E-01
V109 -3.000E+00
V110 3.894E+00
V1512 1.058E-01
Total Power Dissipation 9.52e-01 Watts
**** 10/09/24 12:59:10 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
A Network for Polynomial Thevenin Parameters
**** OPERATING POINT INFORMATION TEMPERATURE = 27.000 DEG C
****
**** VOLTAGE-CONTROLLED CURRENT SOURCES

© STM Journals 2024. All Rights Reserved

NAME GO16 G018 G017 G170
I-SOURCE 4.519E + 02 4.783E + 01 4.783E + 01 4.783E + 01
**** VOLTAGE-CONTROLLED VOLTAGE SOURCES
NAME E1213
V-SOURCE 4.783E + 01
I-SOURCE 1.058E-01
**** CURRENT-CONTROLLED CURRENT SOURCES
Name FO14 FIV1512
I-SOURCE 1.058E-01 1.058E-01
Job Concluded
**** 10/09/24 12:59:10 ****** PSpice Lite (October 2012) ****** ID# 10813

Table 2 describes a different circuit with polynomial implemented network with verifying procedure.

Table 1(b). Circuit file for Figure 2 with SPICE Simulation.

****** 10/09/24 16:42:36 ****** PSpice Lite (October 2012) ****** ID# 10813 ******
A Network for Polynomial Thevenin Parameters
****** Circuit Description**
****
*Figure 2(a) & (b) network
R12 1 2 50
V20 2 0 DC 10
R10 1 0 30
R13 1 3 40
R30 3 0 40
R34 3 4 10
I04 0 4 DC 5
*ABOVE CIRCUIT REPRESENTATION FOR THEVENIN RESISTANCE
R50 5 0 50
R550 5 0 30
R56 5 6 40
R60 6 0 40
R67 6 7 10
R70 7 0 1E20
I05 0 5 DC 1
*IMPLEMENTATION OF POLYNOMIAL EQUIVALENT
I08 0 8 DC 3
R89 8 9 10
V109 10 9 2
R1011 10 11 10
V110 11 0 DC 1
R1012 10 15 10
I012 0 15 DC 1
R130 13 0 10
*REALIZATION OF POLYNOMIAL RESISTANCE BETWEEN NODES (12) AND (13)
V1512 15 12
E1213 12 13 POLY (2) 14 0 16 0 0 0 0 0 1
GO16 0 16 POLY (1) 5 0 0 1 1 0.2

R160 16 0 1
FO14 0 14 V1512 1
R140 14 0 1
*VERIFICATION OF RESISTANCE BETWEEN NODES (12) AND (`13)
G018 0 18 POLY (2) 12 0 13 0 0 1 -1
R180 18 0 1
G017 0 17 POLY (2) 17 0 19 0 0 0 0 0 1
G170 17 0 18 0 1
FIV1512 0 19 V1512 1
R190 19 0 1
R170 17 0 1E20
.OP
.end
**** 10/09/24 16:42:36 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
A NETWORK FOR POLYNOMIAL THEVENIN PARAMETERS
**** SMALL SIGNAL BIAS SOLUTION TEMPERATURE = 27.000 DEG C
****
NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE
(1) 41.0130 (2) 10.0000 (3) 120.5100 (4) 170.5100
(5) 15.1900 (6) 7.5949 (7) 7.5949 (8) 68.4780
(9) 38.4780 (10) 40.4780 (11) 1.0000 (12) 49.9560
(13) .5221 (14) .0522 (15) 49.9560 (16) 946.8800
(17) 946.8800 (18) 49.4340 (19) .0522
VOLTAGE SOURCE CURRENTS
NAME CURRENT
V20 6.203E-01
V109 -3.000E+00
V110 3.948E+00
V1512 5.221E-02
TOTAL POWER DISSIPATION -4.15E+00 WATTS
**** 10/09/24 16:42:36 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
A NETWORK FOR POLYNOMIAL THEVENIN PARAMETERS
**** OPERATING POINT INFORMATION TEMPERATURE = 27.000 DEG C
****
**** VOLTAGE-CONTROLLED CURRENT SOURCES
NAME GO16 G018 G017 G170
I-SOURCE 9.469E + 02 4.943E + 01 4.943E + 01 4.943E+01
**** VOLTAGE-CONTROLLED VOLTAGE SOURCES
NAME E1213
V-SOURCE 4.943E+01
I-SOURCE 5.221E-02
**** CURRENT-CONTROLLED CURRENT SOURCES
NAME FO14 FIV1512
I-SOURCE 5.221E-02 5.221E-02
JOB CONCLUDED
**** 10/09/24 16:42:36 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
A NETWORK FOR POLYNOMIAL THEVENIN PARAMETERS
**** JOB STATISTICS SUMMARY

© STM Journals 2024. All Rights Reserved

****

Analog behavioral modeling of P-spice (VALUE) option is used in Table 2 to describe polynomial implementation at DC.

Table 2. Circuit file for Figure 1 with second technique using SPICE.

**** 10/09/24 13:07:20 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
A Network for Polynomial Thevenin Parameters
**** CIRCUIT DESCRIPTION
****
***Analog Behavioral Modeling Is Used**
R12 1 2 50
V20 2 0 DC 5
R10 1 0 30
R13 1 3 10
R30 3 0 20
R34 3 4 10
I04 0 4 DC 2
***Above Circuit Representation for Thevenin Resistance**
R50 5 0 50
R550 5 0 30
R56 5 6 10
R60 6 0 20
R67 6 7 10
R70 7 0 1E20
I05 0 5 DC 1
***Implementation of Polynomial Equivalent**
I08 0 8 DC 3
R89 8 9 10
V109 10 9 2
R1011 10 11 10
V110 11 0 DC 1
R1012 10 15 10
I012 0 15 DC 1
R130 13 0 10
***Realization of Polynomial Resistance Between Nodes (12) and (13)**
V1512 15 12
E1213 12 13 VALUE = {(V (5) + V(5)*V(5)+0.2*V(5)*V(5)*V(5))*I(V1512)}
.OP
.end
**** 10/09/24 13:07:20 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
A Network for Polynomial Thevenin Parameters
**** SMALL SIGNAL BIAS SOLUTION TEMPERATURE = 27.000 DEG C
****
Node Voltage Node Voltage Node Voltage Node Voltage
(1) 16.5380 (2) 5.0000 (3) 24.3590 (4) 44.3590
(5) 11.5380 (6) 7.6923 (7) 7.6923 (8) 67.9420
(9) 37.9420 (10) 39.9420 (11) 1.0000 (12) 48.8830

© STM Journals 2024. All Rights Reserved

(13) 1.0583 (15) 48.8830
VOLTAGE SOURCE CURRENTS
NAME CURRENT
V20 2.308E-01
V109 -3.000E+00
V110 3.894E+00
V1512 1.058E-01
Total Power Dissipation 9.52e-01 Watts
**** 10/09/24 13:07:20 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
A NETWORK FOR POLYNOMIAL THEVENIN PARAMETERS
**** OPERATING POINT INFORMATION TEMPERATURE = 27.000 DEG C
****
****** Voltage-Controlled Voltage Sources**
NAME E1213
V-SOURCE 4.783E+01
I-SOURCE 1.058E-01
JOB CONCLUDED
**** 10/09/24 13:07:20 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
A Network for Polynomial Thevenin Parameters
**** JOB STATISTICS SUMMARY
****

In Table 2, the first part describes the circuit under consideration and Thevenin resistance description of same circuit. In the later portion implementation of polynomial form is described by analog behavioral modeling VALUE option.

A third technique is described in Table 3 to describe polynomial connected in Figure 3. The results are verified [11–15].

Table 3. Circuit file for Figure 1 with third technique using SPICE.

**** 10/08/24 19:40:21 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
A Network for Polynomial Thevenin Parameters
**** CIRCUIT DESCRIPTION
****
*Thevenin Resistance = RT + RT*RT
*USING MODIFIED CONVENTIONAL METHOD
R12 1 2 50
V20 2 0 DC 5
R10 1 0 30
R13 1 3 10
R30 3 0 20
R34 3 4 10
I04 0 4 DC 2
***Above Circuit Representation for Thevenin Resistance**
R50 5 0 50
R550 5 0 30
R56 5 6 10

© STM Journals 2024. All Rights Reserved

R60 6 0 20
R67 6 7 10
R70 7 0 1E20
FI05 0 5 V1512 1
***Implementation of Polynomial Equivalent**
I08 0 8 DC 3
R89 8 9 10
V109 10 9 2
R1011 10 11 10
V110 11 0 DC 1
R1012 10 15 10
I012 0 15 DC 1
R130 13 0 10
***Realization of Polynomial Resistance Between Nodes (15) and (13)**
V1512 15 12
E1220 12 20 5 0 1
E1213 20 13 POLY (2) 19 0 18 0 0 0 0 0 1
***Verification of Resistance Between Nodes (15) And (\`13)**
G018 0 18 POLY (2) 19 0 18 0 0 0 0 0 0 0 0 1
GFR180 18 0 POLY (1) 5 0 0 0 1
R180 18 0 1E22
FIX 0 19 V1512 1
R190 19 0 1
.OP
.end
**** 10/08/24 19:40:21 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
A Network for Polynomial Thevenin Parameters
****** Small Signal Bias Solution Temperature = 27.000 Deg C**
****
NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE
(1) 16.5380 (2) 5.0000 (3) 24.3590 (4) 44.3590
(5) 3.3689 (6) 2.2459 (7) 2.2459 (8) 66.0800
(9) 36.0800 (10) 38.0800 (11) 1.0000 (12) 45.1610
(13) 2.9197 (15) 45.1610 (18) 133.1400 (19) .2920
(20) 41.7920
VOLTAGE SOURCE CURRENTS
NAME CURRENT
V20 2.308E-01
V109 -3.000E+00
V110 3.708E+00
V1512 2.920E-01
Total Power Dissipation 1.14e+00 Watts
**** 10/08/24 19:40:21 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
A NETWORK FOR POLYNOMIAL THEVENIN PARAMETERS
**** OPERATING POINT INFORMATION TEMPERATURE = 27.000 DEG C
****
**** VOLTAGE-CONTROLLED CURRENT SOURCES
NAME G018 GFR180

© STM Journals 2024. All Rights Reserved

I-SOURCE 1.135E+01 1.135E+01
**** VOLTAGE-CONTROLLED VOLTAGE SOURCES
NAME E1220 E1213
V-SOURCE 3.369E+00 3.887E+01
I-SOURCE 2.920E-01 2.920E-01
**** CURRENT-CONTROLLED CURRENT SOURCES
NAME FI05 FIX
I-SOURCE 2.920E-01 2.920E-01
JOB CONCLUDED
**** 10/08/24 19:40:21 ****** PSpice Lite (October 2012) ****** ID# 10813 ****

In Table 4 a different method using four independent voltage and four independent current sources is used and is connected at the input of the circuit for which AC/DC Thevenin impedance/resistance has to be determined. Equivalent dependent sources with regulating voltage and current at the input, where independent voltages are connected, take the place of the dependent voltage and current sources in the circuit whose Thevenin impedance/resistance must be ascertained.

Table 4. A new method to obtain Thevenin impedance with example.

A New Method for Computation of Thevenin/Norton Impedance
***Using Dependent Voltage/Current Sources**
L12 1 2 1NH
C20 2 0 1PF
R23 2 3 10
EV30 3 0 6 7 5
R220 2 0 20
R24 2 4 10
GI04 0 4 8 9 1
R40 4 0 15
R45 4 5 5
GI05 0 5 6 7 1
L1011 10 11 1NH
HV1112 11 12 V201 1
R120 12 0 5
R1113 11 13 2
FI013 0 13 V211 2
V67 6 7 DC 1
V78 7 8 DC -1
V89 8 9 AC 1
V91 9 1 AC -1
I67 6 20 DC 1
V201 20 1
I76 1 6 AC 1
I667 6 21 AC 1
V211 21 1
I776 1 6 DC 1
I01 0 6 DC 1
I001 0 6 AC 1
*THE THEVENIN RESISTANCE IS THE NODE VOLTAGE AT NODE (6)

© STM Journals 2024. All Rights Reserved

*AFTER RUNNING THE POGRAM AGAIN WITH V67, V78, V89, V91 ZEROS
*ALSO I67, I76, I667, I776 CURRENT SOURCES ZEROS
*THE VALUES ARE 5.263 OHMS AT DC
*V (6) MAGNITUDE VM (6) 13.3, PHASE VP (6) 66.79 DEGREE AT 2GHZ
.AC LIN 10 1GHZ 10GHZ
.PRINT AC VM (6) VP (6) VR (6) VI (6)
.op

The Thevenin resistance for Figure 3 is calculated by TF command available with SPICE. It is also verified by conventional methods to calculate Thevenin resistance and both found to be the same. The content in Table 5 gives the results obtained for TF command using Figure 3.

Table 5. Verification of Thevenin resistance using TF command of spice.

Verification with Transfer Function tf Command
**** **Circuit Description**
****
***Above Circuit Representation for Thevenin Resistance**
R50 5 0 50
R550 5 0 30
R56 5 6 10
R60 6 0 20
R67 6 7 10
R70 7 0 1E20
I05 0 5 DC 1
***Implementation of Polynomial Equivalent**
I08 0 8 DC 3
R89 8 9 10
V109 10 9 2
R1011 10 11 10
V110 11 0 DC 1
R1012 10 15 10
I012 0 15 DC 1
R130 13 0 10\
V1512 15 12
E1213 12 13 POLY (2) 14 0 16 0 0 0 0 0 1
GO16 0 16 POLY (1) 5 0 0 1 1 0.2
R160 16 0 1
FO14 0 14 V1512 1
R140 14 0 1
.TF V (8) V110
.OP
.end
**** 11/19/24 12:39:27 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
Verification with Transfer Function Tf Command
**** SMALL SIGNAL BIAS SOLUTION TEMPERATURE = 27.000 DEG C
****
Node Voltage Node Voltage Node Voltage Node Voltage
(5) 11.5380 (6) 7.6923 (7) 7.6923 (8) 67.9420

© STM Journals 2024. All Rights Reserved

(9) 37.9420 (10) 39.9420 (11) 1.0000 (12) 48.8830
(13) 1.0583 (14) .1058 (15) 48.8830 (16) 451.9100
VOLTAGE SOURCE CURRENTS
NAME CURRENT
V109 -3.000E+00
V110 3.894E+00
V1512 1.058E-01
Total Power Dissipation 2.11e+00 Watts
**** 11/19/24 12:39:27 ****** PSpice Lite (October 2012) ****** ID# 10813 ****
VERIFICATION WITH TRANSFER FUNCTION TF COMMAND
**** OPERATING POINT INFORMATION TEMPERATURE = 27.000 DEG C
****
**** VOLTAGE-CONTROLLED CURRENT SOURCES
NAME GO16
I-SOURCE 4.519E+02
**** VOLTAGE-CONTROLLED VOLTAGE SOURCES
NAME E1213
V-SOURCE 4.783E+01
I-SOURCE 1.058E-01
****** Current-Controlled Current Sources**
NAME FO14
I-SOURCE 1.058E-01
**** SMALL-SIGNAL CHARACTERISTICS
V (8)/V110 = 9.792E-01
INPUT RESISTANCE AT V110 = 4.819E+02
OUTPUT RESISTANCE AT V (8) = 1.979E+01
Job Concluded

FUTURE AND SCOPE

The research presented establishes a foundational method for calculating Thevenin and Norton equivalent resistances and impedances using advanced simulation tools like SPICE and OrCAD. Future developments in this field could explore several directions to refine and extend these approaches. Exploration of More Complex Networks: Future research could address more intricate networks that involve non-linear components, active elements, or time-varying sources. Investigating the use of polynomial representations for these advanced circuit configurations could improve the accuracy and adaptability of impedance calculations in dynamic systems.

Enhancement of Simulation Efficiency: Although this study utilizes SPICE and OrCAD for impedance determination, there is potential for optimizing these simulation tools to reduce computation time, particularly in large-scale networks. Additionally, integrating machine learning techniques to predict Thevenin and Norton equivalents based on circuit parameters could accelerate the process and minimize simulation duration. Application to Real-World Systems: The practical implementation of these methods in real-world applications, such as power distribution networks, telecommunications, and embedded systems, offers an important area for future exploration. Validating the proposed techniques using real-world data would help confirm their effectiveness in more complex and practical scenarios. Incorporation of Advanced Circuit Components: Future studies may also investigate the inclusion of advanced circuit elements, such as superconductors, semiconductors, or MEMS (Micro-Electromechanical Systems), broadening the scope of this research to cover emerging technologies. Development of Automated Tools and User Interfaces: To increase accessibility, future work could involve the creation of user-friendly interfaces and automated software tools that integrate these

© STM Journals 2024. All Rights Reserved

simulation techniques for impedance determination. This could lead to the development of standalone or cloud-based applications that simplify circuit design and analysis for engineers and designers.

CONCLUSIONS

DC linear electronic circuits with independent sources circuit elements (resistors) are broken into two separate portions for the Thevenin equivalent circuit parameters and are evaluated. The Thevenin equivalent circuit parameters thus obtained are used as polynomial representations (in three ways) in other networks by Spice-compatible techniques. Examples are given as Spice software-compatible circuits, and the polynomial equivalent resistance between nodes (12) and (13), (15), and (13) are verified with conventional techniques. Using dependent sources to represent independent sources, a different method to obtain Thevenin impedance is presented and verified. The DC analysis with the following descriptions:

a. DC sweep of a source (current/voltage), model parameter/temperature over different ranges.
b. Making linearized model parameters of nonlinear devices.
c. Transfer functions, like input resistance, gain, and output resistance.
d. DC sensitivity calculation.

All these can be made use of in the extended works (ideas) with slight changes in required description of Thevenin resistance function.

The Thevenin and Norton equivalent circuits, along with their parameter values in both linear and nonlinear circuits, are of great interest to designers and researchers working in circuit simulation. To determine the Thevenin resistance and Norton conductance for a network, various circuit theorems can be applied, with the substitution network theorem being suitable for simulation tools, such as OrCAD, SPICE, PSpice AD, and Microcap. The SPICE/PSpice student version software can also be used to verify the DC Thevenin results through the substitution theorem, offering a helpful resource for students. In place of DC analysis, the transient analysis option in SPICE can be employed by students to calculate Thevenin/Norton resistance/conductance for pure resistive and active circuits, utilizing SPICE's built-in polynomial voltage-to-current and current-to-voltage conversion sources. This method is also applicable to active circuits using HEMTs and BJTs, represented by their small signal/linear models, and the outcomes can be confirmed through the superposition theorem. Additionally, resistances can be replaced using analog behavioral modeling (VALUE) feature in the OrCAD simulation tool. Temperature-dependent resistor models, available through lookup tables, can also be applied in the calculation of Thevenin and Norton equivalents for electronic circuits.

REFERENCES

1. Prigozy S. Novel applications of SPICE in engineering education. IEEE Trans Educ. 1989;32(1):35–38.
2. Nagel LW. SPICE 2, a computer program to simulate semiconductor circuits. Electronic Research Laboratory Report ERL-M520. Berkeley, USA: University of California; 1975.
3. SPICE version 2G user's guide. Berkeley (CA): University of California; 1975.
4. Wilson B. Tutorial review: trends in current conveyor and current-mode amplifier design. Int J Electron. 1992;73(3):573–583.
5. Bharath KK. Multi two-port parameter simulation using Pspice. Technical Report, Semiconductor Research Laboratory. Japan: Oki Electric Industry; 1990.
6. Van Valkenburg ME. Network Analysis. 3rd ed. Englewood Cliffs (NJ): Prentice-Hall; 1976. Chapter IX. 259–261.
7. Scott HH. A new type of selective circuit and some applications. Proc Inst Radio Eng. 1938;26(2):226–235.
8. Vladimirescu A, Zhang K, Newton AR, Pederson DO, Sangiovanni-Vincentelli A. SPICE Version 2G User's Guide. Berkeley (CA): Dept. of Electrical Engineering and Computer Science, University of California; 1975.

© STM Journals 2024. All Rights Reserved

9. Madec M, Lallement C, Haiech J. Modeling and simulation of biological systems using SPICE language. PLoS One. 2017;12(8):e0182385.
10. Jin LM, Chan SP. A unified and efficient approach for determining Thevenin (Norton) equivalent circuits. IEEE Trans Educ. 1989;32(3):408–410.
11. Bhattacharyya SP, Oliveira VA, Magossi RF. Thevenin's Theorem, Cramer's Rule, and Parameterized systems: Some connections. IEEE Control Syst Mag. 2019;39(2):84.
12. Hongyang Z. Discussion on Thevenin's theorem and Norton's theorem. International Conference on Electronic & Mechanical Engineering and Information Technology. Harbin, China. 2011, Aug 12-14. 520–522. IEEE.
13. MicroSim Corporation. Micro Sim PSpice & Basics, Circuit Analysis Software, User's Guide, Version 8.0. Irvine (CA): MicroSim Corporation; 1997.
14. Epler B. SPICE2 application notes for dependent sources. IEEE Circuits Devices Mag. 1984;3(5):36–44.
15. Bharath KK. Inverse ABCD parameter determination using SPICE. Int J Analog Integr Circuits. 2017;3(1):1–6.

© STM Journals 2024. All Rights Reserved