

Solving Non-Linear Equations for Thevenin and Norton Circuits Using Analog Behavioral Modeling with PSpice Software

K. Bharath Kumar*

Abstract

A new approach useful for circuit designers for implementing circuit parameters described by non-linear equations and Thevenin and Norton network equivalent parameters represented by equations is given. The content is divided into two parts, (a) Solving Non-linear equations using analog behavioral modelling using Pspice software (b) Thevenin and Norton equivalent circuit parameter determination for pure resistance DC circuits using SPICE simulator. The non-linear equations are solved using network elements/models described by SPICE. A different method which is not previously described is used for computing Thevenin/Norton DC equivalents. The results are useful in methods to obtain DC Thevenin/Norton circuits for pure DC networks and a new circuit described by non-linear, linear functions with Thevenin/Norton circuit parameters. Techniques to obtain DC Thevenin network parameters, other than usual transfer function (TF) are described. Different examples are described, and results are obtained by running Pspice simulation. Also, any circuit variables described by non-linear equations of different networks can be solved and implemented in the extension of circuits described by Thevenin/Norton equivalents. The resistances represented in this paper can be modeled with VALUE option of analog behavioral modelling option. Mostly all passive networks can be replaced by their equivalent two-ports and analogue behavioral modelling can be used to represent two-ports and the same procedure described in this paper. Temperature dependence can be introduced for variables in a future work. We can also simulate conductance's with reverse representation of VALUE options. The procedure could be used for T type representation of passive circuits and two-ports represented by Z and Y parameters.

Keywords: Analog circuits, circuit theory, simulation and modeling, spice simulation, RF circuit and devices.

INTRODUCTION

Spice2G, Pspice [1–3] are simulation programs used by circuit designers to analyze electronic active and passive circuits. It has additional features [4] compared to Spice2G, where I–V devices can be modeled [5] using analog behavior modeling. First, in this design, (a) non-linear simultaneous equations of three variables and (b) non-linear trigonometric two-variable simultaneous equations are solved using the Pspice analog behavior option by suitable manipulation. Second, a new procedure to obtain DC Thevenin/Norton equivalents is described with circuit examples. Finally, we describe how to combine the two for useful and complicated circuit analyses.

*Author for Correspondence

K. Bharath Kumar
E-mail: bbkrishnapuram3@gmail.com

Researcher, Oki Electric Company, R&D Division, Tokyo, Japan.

Received Date: August 21, 2026
Accepted Date: August 26, 2026
Published Date: September 02, 2026

Citation: K. Bharath Kumar. Solving Non-Linear Equations for Thevenin and Norton Circuits Using Analog Behavioral Modeling with PSpice Software. Journal of Semiconductor Devices and Circuits. 2026; 13(2): 28–36p.

METHODOLOGY AND LITERATURE SURVEY

Solving Equations

Solving non-linear equations refers to finding the roots of the function, which are the points where the function values are exactly zero: $f(x) = 0$. When applied to a system of equations, the problem is generalized as obtaining a vector of unknown variables x that brings all equations/expressions to simultaneously equal zero as follows:

$$f_1(x_1, x_2, \dots, x_n) = 0$$

Numerical Solvers & Algorithms

Because exact zero solutions (algebraic solutions) are not possible, numerical algorithms make an initial guess and iteratively modify it until the solution converges to the true value. The most common methods include

- *Newton-Raphson Method*: Using partial derivatives (the Jacobian matrix) to project the next guess, the method converges very quickly.
- *Secant / Modified Secant Method*: The Jacobian is estimated using both finite differences and analytic derivatives, which is often difficult to obtain.
- *Bisection / Bracketing Methods*: These are safer for single-equation problems because the boundary containing the root is known, although they are much slower.
- *Quasi-Newton Methods*: Here optimization algorithms that approximate the Jacobian without computing it from scratch are used at every iteration.

Simulating Dynamic Systems

- Using time-stepping solvers like the Runge-Kutta or Euler methods.
- Non-linear equations represented as differential equations (solving these equations means simulating their response over time).

Software Tools

Several CAD software and computer environments provide built-in, highly optimized solver tools and packages. They are,

- *MATLAB*: The `fsolve` function was used to solve systems of non-linear algebraic equations, and differential solvers such as `ode45` were used to simulate dynamic, non-linear systems.
- *Python*: Here, the SciPy Optimize package contains `fsolve` and `root` for algebraic systems and similar other non-linear differential simulations.
- *Wolfram Mathematica / Maple*: Useful package for symbolic simplification and high-accuracy numeric solver with `NSolve` and `NDSolve`.

The following sections describe the procedure described in this article.

FIRST PROCEDURE

Non-Linear Simultaneous Equations of Three Variables

Consider the following three equations in three variables,

$$(x_1 + x_2)^2 - x_1 \cdot x_2 + x_3 = 10$$

$$x_1 \cdot x_3 + (x_1)^2 = 4$$

$$(x_2)^2 + 2 \cdot (x_1 + x_3) = 12$$

Figure 1 shows the schematic Pspice circuit used to solve the non-linear equations in three variables. The following describes the Pspice circuit file Table 1 to solve the above nonlinear equations in three variables.

Non-Linear two Variable Trigonometry Simultaneous Equations

Consider the following non-linear trigonometric two variable functions

$$X_1^2 - 2 \cdot \sin(X_2) + X_1 \cdot X_2 = 5.14158$$

$$X_2 - X_1 \cdot X_2 = -1.570796$$

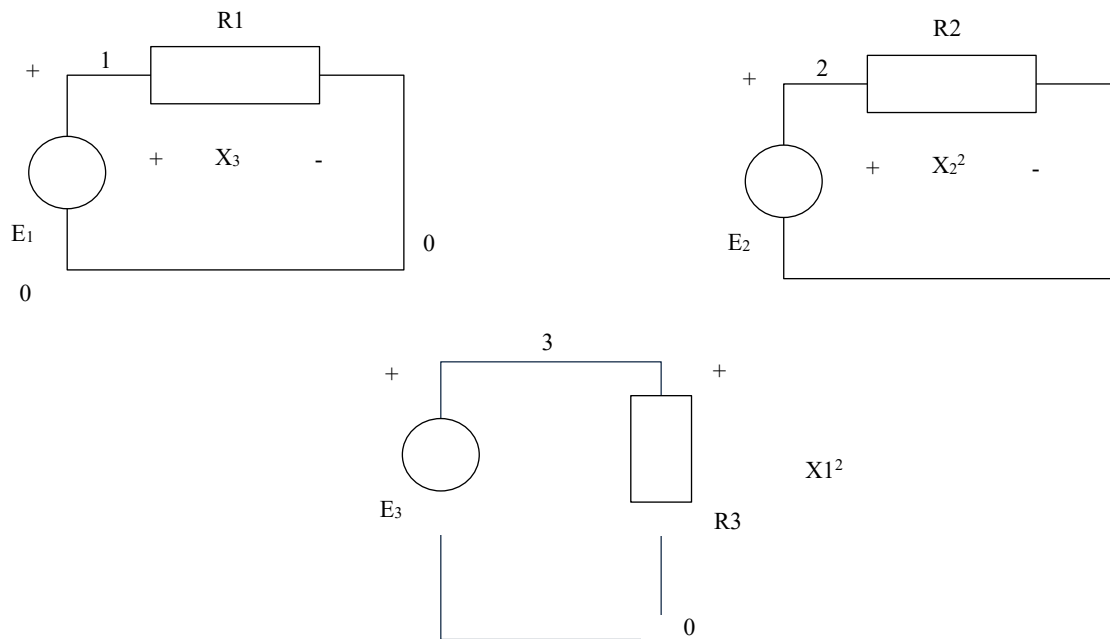


Figure 1. Schematic Pspice circuit to solve non-linear equations in three variables.

Table 1. Solving nonlinear equations with Pspice.

**** 08/23/22 13:13:50 ***** PSpice Lite (October 2012) ***** ID# 10813 ****
TO VERIFY USING ANALOG BEHAVIOUR MODELLING PSPICE
**** CIRCUIT DESCRIPTION

R1 1 0 1
R2 2 0 1
R3 3 0 1
E1 1 0 VALUE={-((SQRT(V(3))+SQRT(V(2))))**2-SQRT(V(3))*SQRT(V(2))-10}
E2 0 3 VALUE={SQRT(V(3))*V(1)-4.}
E3 2 0 VALUE={12-2.*(SQRT(V(3))+V(1))}
E4 4 0 VALUE={SQRT(V(3))}
R4 4 0 1
E5 5 0 VALUE={SQRT(V(2))}
R5 5 0 1
* VALUE OF X1 IS VOLTAGE ACROSS R4
*VALUE OF X2 IS VOLTAGE ACROSS R5
*VALUE OF X3 IS VOLTAGE ACROSS R1
.END
**** 08/23/22 13:13:50 ***** PSpice Lite (October 2012) ***** ID# 10813 ****
TO VERIFY USING ANALOG BEHAVIOUR MODELLING PSPICE
**** SMALL SIGNAL BIAS SOLUTION TEMPERATURE = 27.000 DEG C

NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE
(1) 3.0000 (2) 4.0000 (3) 1.0000 (4) 1.0000
(5) 2.0000

Table 2. PSPICE circuit file for solving equations with SINE functions.

**** 08/24/22 18:58:09 ***** PSpice Lite (October 2012) ***** ID# 10813 ****
TO VERIFY USING ANALOG BEHAVIOUR MODELLING PSPICE
**** CIRCUIT DESCRIPTION

R1 1 0 1
R2 2 0 1
E1 1 0 VALUE={2.*sin(V(2))-SQRT(V(1))*V(2)+5.14159}
E2 2 0 VALUE={-1.570796+SQRT(V(1))*V(2)}
E3 3 0 VALUE={SQRT(V(1))}
R3 3 0 1
E5 5 0 VALUE={SIN(1.570796)}
R5 5 0 1
EVERIFY 7 0 VALUE={1.6999**2-2*sin(2.2444)+1.6999*2.2444}
R70 7 0 1
EVERIFY2 8 0 VALUE={2.2444-1.6999*2.2444}
R8 8 0 1
EVERY 9 0 VALUE={2**2-2.*SIN(1.570796)+2.*1.570796}
R9 9 0 1
EVERQ 10 0 VALUE={1.570796-2.*1.570796}
R10 10 0 1
*VALUE OF X1 IS VOLTAGE ACROSS R3
*VALUE OF X2 IS VOLTAGE ACROSS R2
.END
**** 08/24/22 18:58:09 ***** PSpice Lite (October 2012) ***** ID# 10813 ****
TO VERIFY USING ANALOG BEHAVIOUR MODELLING PSPICE
**** SMALL SIGNAL BIAS SOLUTION TEMPERATURE = 27.000 DEG C

NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE
(1) 2.8896 (2) 2.2444 (3) 1.6999 (5) 1.0000
(7) 5.1418 (8) -1.5709 (9) 5.1416 (10) -1.5708

Table 2 describes the PSpice circuit file and results for the set of the above non-linear equations in two variables.

Now, consider a different set of two-variable nonlinear trigonometric equations in X, Y,
 $X = 2 \cdot \sin(Y) - \sqrt{\text{abs}(X)} \cdot Y + 5.14159$
 $Y = -1.570796 + \sqrt{\text{abs}(X)} \cdot Y$

Table 3 describes the above equations, and the results are also given in the form of voltages.

Consider a different set of nonlinear equations in two variables as described below,
 $-X_1^2 = 2 \cdot \sin(X_2) - \sqrt{\text{abs}(X_1^2)} \cdot X_2 + 5.14159$
 $-X_2 = -1.570796 + \sqrt{\text{abs}(X_1^2)} \cdot X_2$

Table 4 describes the SPICE file used to solve the above equations.

Table 3. A PSPICE File to solve non-linear equations described by two variables.

**** 08/26/22 09:13:32 **** PSpice Lite (October 2012) **** ID# 10813 ****
TO VERIFY USING ANALOG BEHAVIOUR MODELLING PSPICE
**** CIRCUIT DESCRIPTION

R1 1 0 1
R2 2 0 1
E1 1 0 VALUE={2.*sin(V(2))-SQRT(V(1))*V(2)+5.14159}
E2 2 0 VALUE={-1.570796+SQRT(V(1))*V(2)}
E3 3 0 VALUE={SQRT(V(1))}
R3 3 0 1
*VALUE OF X1 IS VOLTAGE ACROSS R1
*VALUE OF X2 IS VOLTAGE ACROSS R2
.END
**** 08/26/22 09:13:32 **** PSpice Lite (October 2012) **** ID# 10813 ****
TO VERIFY USING ANALOG BEHAVIOUR MODELLING PSPICE
**** SMALL SIGNAL BIAS SOLUTION TEMPERATURE = 27.000 DEG C

NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE
(1) 2.8896 (2) 2.2444 (3) 1.6999

Table 4. PSpice circuit schematic to solve the equations with results.

**** 08/26/22 09:37:32 **** PSpice Lite (October 2012) **** ID# 10813 ****
TO VERIFY USING ANALOG BEHAVIOUR MODELLING PSPICE
**** CIRCUIT DESCRIPTION

R1 1 0 1
R2 2 0 1
E1 4 0 VALUE={ABS(V(1))}
R40 4 0 1
G1 1 0 VALUE={2.*sin(V(2))-SQRT(V(4))*V(2)+5.14159}
G2 2 0 VALUE={-1.570796+SQRT(V(4))*V(2)}
G3 3 0 VALUE={SQRT(V(1))}
R3 3 0 1
*Result verification
G4 5 0 VALUE={2*sin(.4857)-SQRT(4.9902)*.4857+5.14159}
R50 5 0 1
G5 6 0 VALUE={-1.570796+SQRT(4.9902)*.4857}
R60 6 0 1
*VALUE OF X1 IS VOLTAGE ACROSS R3
*VALUE OF X2 IS VOLTAGE ACROSS R2
.END
**** 08/26/22 09:37:32 **** PSpice Lite (October 2012) **** ID# 10813 ****
TO VERIFY USING ANALOG BEHAVIOUR MODELLING PSPICE
**** SMALL SIGNAL BIAS SOLUTION TEMPERATURE = 27.000 DEG C

NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE
(1) -4.9902 (2) .4857 (3) -2.2339 (4) 4.9902
(5) -4.9903 (6) .4858

The effective use of the analog behavior modeling option of PSPICE is useful in solving sets of nonlinear multivariable equations using circuit theory principles.

SECOND PROCEDURE

This description is for obtaining the Thevenin and Norton equivalent circuit parameter determination for pure resistive DC circuits using SPICE simulation. The Thevenin and Norton equivalent parameters (Thevenin resistance, Thevenin voltage, and Norton current) are of great interest to circuit designers. In this design idea, computers aided the determination of these parameters using circuit simulators such as Pspice, Spice2G is described which are useful for analyzing both practical and theoretical DC circuits [6–15].

The effective use of the DC option and VALUE option of the analog behavioral modeling option of PSpice is useful in obtaining the Thevenin and Norton equivalent circuit parameters (Table 5) knowing one at a time for the DC circuit.

Table 5. PSPICE circuit file to determine DC Thevenin/Norton equivalents.

**** 08/28/22 12:15:11 ***** PSpice Lite (October 2012) ***** ID# 10813 ****
**** CIRCUIT DESCRIPTION

*THE FOLLOWING FILE DESCRIBES AND SOLVES FOR NORTON SHORT CIRCUIT
*CURRENT, THEVENIN RESISTANCE AND THEVENIN VOLTAGE BY KNOWING ANY
*ONE PARAMETER AT A TIME FOR THE CIRCUIT OF FIGURE. 2
*TO OBTAIN NORTON CURRENT USING THEVENIN VOLTAGE AND RESISTANCE
*A PART OF CIRCUIT IS DESCRIBED BY THE FOLLOWING SUB CIRCUIT
SUBCKT RS 3
R1 3 5 2.2
R2 5 0 1.8
R3 5 4 4.2
R4 4 0 2
ENDS RS
X 3 RS
VO3 3 0 DC 1
ET1 6 0 VALUE={-3V*(-I(VO3))}
RT1 6 0 1
*V(6) IS NORTON SHORT CIRCUIT CURRENT ISC
*TO OBTAIN THEVENIN RESISTANCE USING THEVENIN VOLTAGE AND NORTON CURRENT
ET2 7 0 VALUE={-3./V(6)}
R70 7 0 1
*THE VALUE OF OF THEVENIN RESISTANCE IS THE VOLTAGE V(7)
* TO OBTAIN THEVENIN VOLTAGE BY KNOWING NORTON CURRENT AND THEVENIN RESISTANCE
ET3 8 0 VALUE = {V(7)*V(6)}
R90 8 0 1
*V(8) IS THE THEVENIN VOLTAGE
.END
**** 08/28/22 12:15:11 ***** PSpice Lite (October 2012) ***** ID# 10813 ****
**** SMALL SIGNAL BIAS SOLUTION TEMPERATURE = 27.000 DEG C

NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE
(3) 1.0000 (6) -.8345 (7) 3.5950 (8) -3.0000
(X.4) .1252 (X.5) .3880
VOLTAGE SOURCE CURRENTS

NAME	CURRENT
VO3	-2.782E-01
TOTAL POWER DISSIPATION 2.78E-01 WATTS	

Table 6. Circuit file to determine Thevenin Norton equivalents.

**** 08/28/22 17:36:50 ***** SPICE2G (October 2022) ***** ID# 10813 ****
**** CIRCUIT DESCRIPTION

*THE FOLLOWING FILE DESCRIBES spice2g COMPATIBLE FILE FOR DETERMINING
*NORTON SHORT CIRCUIT CURRENT, THEVENIN(NORTON) RESISTANCE, AND THEVENIN
*VOLTAGE BY KNOWING ANY PARAMETER AT A TIME FOR THE CIRCUIT OF FIGURE. 2
*TO OBTAIN NORTON CURRENT USING THEVENIN VOLTAGE AND RESISTANCE
*A PART OF CIRCUIT IS DESCRIBED BY THE FOLLOWING SUB CIRCUIT
.SUBCKT RS 1
R1 1 3 2.2
R2 3 0 1.8
R3 3 4 4.2
R4 4 0 2
.ENDS RS
IDC 0 1 DC 1
X 1 RS
E1 5 0 POLY(2) 1 0 5 6 0 0 0 0 1000
VT 6 0 DC -3
R56 5 6 0.001
G1 0 7 5 6 1000
R70 7 0 1
*V(7) IS NORTON SHORT CIRCUIT CURRENT ISC
GT 0 8 POLY(2) 7 0 8 0 0 0 0 1
GSC 8 0 6 0 1
R80 8 0 1E12
*V(8) IS THE THEVENIN(NORTON) RESISTANCE
GC 0 9 POLY(2) 7 0 8 0 0 0 0 1
R90 9 0 1
*V(9) IS THE THEVENIN VOLTAGE
.END
**** 08/28/22 17:36:50 ***** SPICE2G (October 2022) ***** ID# 10813 ****
**** SMALL SIGNAL BIAS SOLUTION TEMPERATURE = 27.000 DEG C

NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE NODE VOLTAGE
(1) 3.5950 (5) -3.0008 (6) -3.0000 (7) -.8347
(8) 3.5940 (9) -3.0000 (X.3) 1.3950 (X.4) .4500
VOLTAGE SOURCE CURRENTS
NAME CURRENT
VT -8.347E-01
TOTAL POWER DISSIPATION -2.50E+00 WATTS
JOB CONCLUDED

The same problem of determining the Thevenin voltage, Thevenin resistance, and Norton current can be solved using SPICE2G compatible files (Table 6), as shown below.

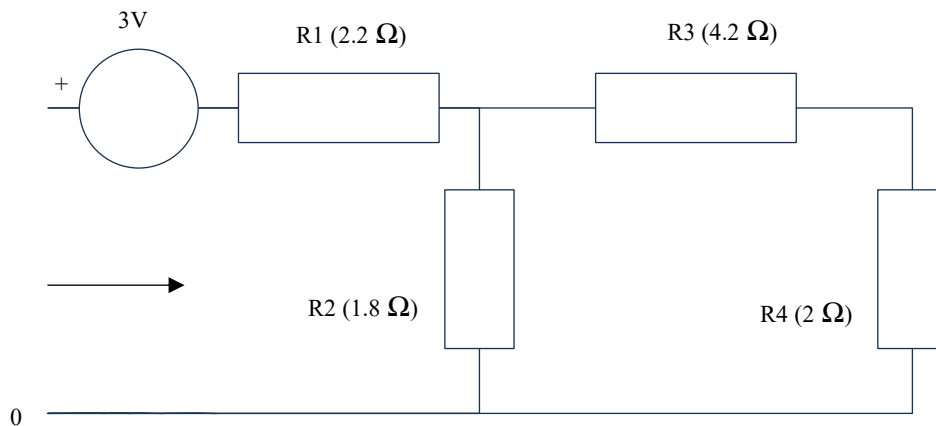


Figure 2. Circuit whose Thevenin and Norton equivalent parameters must be determined at the input.

It is evident from the above analyses and results that both Pspice and Spice2G provide the same answers for the circuit (Figure 2) for Thevenin voltage, Thevenin (Norton) Resistance, Norton current. This method can be applied to various DC resistive linear circuits.

CONCLUSIONS

Usually, nonlinear equations in circuit/network theory are solved (when analyzing networks containing components such as diodes or BJT transistors) by numerical methods such as the Newton-Raphson, Load-Line graphical methods combined with Thevenin's or Norton and other circuit theorems. Using PSpice, one cannot directly calculate the Thevenin or Norton equivalent of a circuit. However, PSpice simulation capabilities can be used with rapid calculations by determining the open-circuit voltage (VOC) and short-circuit current (ISC) values. After constructing the circuit in PSpice or SPICE, we can simulate a short circuit with a small resistance value and then replace the short-circuit value with a larger value to obtain the open circuit VOC. Non-linear equations and systems of non-linear equations cannot be solved using traditional matrix operations or direct algebraic elimination because they contain polynomial expressions with powers, variables multiplied together, or trigonometric functions. Because exact (analytical) solutions rarely exist for these problems, their simulation and solution rely on numerical methods and iterative algorithms. Once the Thevenin/Norton equivalents or solutions of non-linear equations are obtained using the same Pspice software, the Thevenin parameters for various circuits can be represented with nonlinear equations, and the variables solved by non-linear equations can be used as Thevenin and Norton circuit representations for a new network.

REFERENCES

1. Witham M. Analog Design Verification with PSpice and MICRO-CAP III. 1990 Aug 1.
2. Kumar KB. Polynomial Thevenin Resistance Networks for Spice Simulation. *International Journal of Microelectronics and Digital Integrated Circuits*. 2024;10(2):22-36p.
3. Kumar KB. Polynomial Thevenin Resistance Networks for Spice Simulation. *International Journal of Microelectronics and Digital Integrated Circuits*. 2024;10(2):22-36p.
4. Tuinenga PW. SPICE: a guide to circuit simulation and analysis using PSpice. Prentice Hall PTR; 1988 Sep 1.
5. Attia JO. PSPICE and MATLAB for electronics: an integrated approach. CRC press; 2010 Jun 23.
6. Rak A, Cserey G. Macromodeling of the memristor in SPICE. *IEEE Transactions on Computer-Aided Design of Integrated Circuits and Systems*. 2010 Mar 18;29(4):632-6.
7. Kumar KB. T- π , π -T Network Conversion for Three Identical Impedances and Equivalent Thevenin Networks Using Pspice Software. *International Journal of Microelectronics and Digital Integrated Circuits*. 2023 Dec 27;9(1):8-25.
8. Rashid MH. SPICE and LTspice for power electronics and electric power. CRC Press; 2024 Nov 13.
9. Fitzpatrick D. Analog design and simulation using OrCAD Capture and PSpice. Newnes; 2017 Dec 11.

10. Al-Hashimi B. The Art of Simulation Using PSPICE Analog and Digital. CRC press; 2019 Dec 6.
11. Ogrodzki J. Circuit simulation methods and algorithms. crc Press; 2018 May 4.
12. Chamas I, Nokali ME. Automated PSpice simulation as an effective design tool in teaching power electronics. IEEE Transactions on Education. 2004 Aug 31;47(3):415-21.
13. Laghari JR, Suthar JL, Cygan S. PSPICE applications in high voltage engineering education. Computers & Education. 1990 Jan 1;14(6):455-62.
14. Thorpe TW. Computerized circuit analysis with SPICE is a complete guide to SPICE with applications. Wiley-Interscience; 1992 Apr 1.
15. Tobin P. Introduction to PSpice and Ohm's Law. In PSpice for Circuit Theory and Electronic Devices 2022 Mar 17 (pp. 1-12). Cham: Springer International Publishing.