

Electronic Devices and Circuits

Jimmie J. Cathey

November 16, 2025

1 Circuit Analysis

1.1 Spice Elements

The passive and active circuit elements introduced in the previous section are all available in SPICE modeling; however, the manner of node specification and the voltage and current sense or direction are clarified for each element by Figure 1.1. The universal ground node is assigned the number 0. Otherwise, the node numbers n_1 (positive node) and n_2 (negative node) are positive integers selected to uniquely define each node in the network. The assumed direction of positive current flow is from the node n_1 to node n_2 . The four controlled sources—voltage-controlled voltage source (VCVS), current-controlled voltage source (CCVS), voltage-controlled current source (VCCS), and current-controlled current source (CCCS)—have the associated controlling element also shown with its nodes indicated by cn_1 (positive) and cn_2 (negative). Each element is described by an *element specification statement* in the SPICE netlist code. Table 1.1 presents the basic format for the element specification statement for each of the elements in Figure 1.1. The first letter of the element name specifies the device and the remaining characters must assure a unique name.

1.2 Network Theorems

Example 1.1. Use SPICE methods to determine the Thévenin equivalent circuit looking to the left through terminals 3,0 for the circuit of Figure 1.1. In SPICE independent source models, an ideal voltage source of 0V acts as a short circuit and an ideal current source of 0 A acts as an infinite impedance

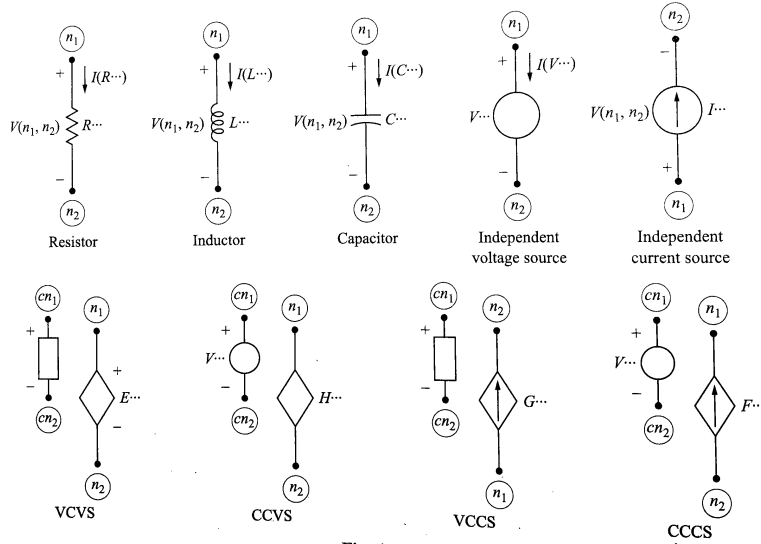


Fig. 1-2

Element	Name	Signal Type	Control Source	Value
Resistor	R...			Ω
Inductor	L...			H
Capacitor	C...			F
Voltage source	V...	AC or DC ^a		V ^b
Current source	I...	AC or DC ^a		A ^b
VCVS	E...		(cn1, cn2)	V/V
CCVS	H...		V...	V/A
VCCS	G...		(cn1, cn2)	A/V
CCCS	F...		V...	A/A

a. Time-varying signal types (SIN, PULSE, EXP, PWL, SFFM) also available.

b. AC signal types may specify phase angle as well as magnitude.

or open circuit. Advantage will be taken of these two features to solve the problem.

Load resistor R_L of Figure 1.1(a) is replaced by the driving point current source I_{dp} of Figure 1.1(b). The netlist code that follows forms a SPICE description of the resulting circuit. The code is set up with parameter-assigned values for V_1 , I_2 , and I_{dp} .

```
Ex1_5.CIR - Thevenin equivalent circuit
.PARAM V1value=0V I2value=0A Idpvalue=1A
```

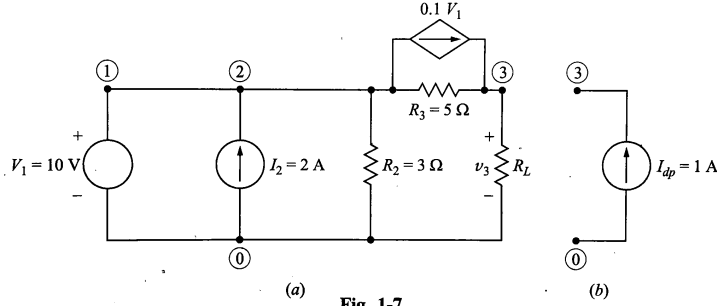


Fig. 1-7

```

V1 1 0 DC {V1value}
R1 1 2 1ohm
I2 0 2 DC {I2value}
R2 2 0 3ohm
R3 2 3 5ohm
G3 2 3 (1,0) 0.1 ; Voltage-controlled current-source
Idp 0 3 DC {Idpvalue}
.END

```

If both V_1 and I_2 are deactivated by setting $V1value=I2value=0$, current $I_{dp} = 1$ A must flow through the Thévening equivalent impedance $Z_{Th} = R_{Th}$ so that $v_3 = I_{dp} R_{Th} = R_{Th}$. Execution of `<Ex1_5.CIR>` by a SPICE program writes the values of the node voltages for nodes 1, 2, and 3 with respect to the universal ground node 0 in a file `<Ex1_5.OUT>`. Poll the output file to find $v_3 = V(3) = R_{Th} = 5.75 \Omega$.

In order to determine V_{Th} (open-circuit voltage between terminals 3,0), edit `<Ex1_5.CIR>` to set $V1value=10V$, $I2value=2A$, and $Idpvalue=0A$. Execute `<Ex1_5.CIR>` and poll the output file to find $V_{Th} = v_3 = V(3) = 14$ V.

1.3 Two-Port Networks

The z parameters and the H parameters can be numerically evaluated by SPICE methods. In electronics applications, the z and h parameters find application in analysis when small ac signals are impressed on circuits that exhibit limited-range linearity. Thus, in general, the test sources in the SPICE analysis should be of magnitudes comparable to the impressed signals of the anticipated application. Typically, the devices used in an electronic circuit will have one or more dc sources connected to bias or that place the device at a favorable point of operation. The input and output ports may be coupled by large capacitors that act to block the appearance of any dc

voltages at the input and output ports while presenting negligible impedance to ac signals. Further, electronic circuits are usually frequency-sensitive so that any set of z or h parameters is valid for a particular frequency. Any SPICE-based evaluation of the z and h parameters should be capable of addressing the above outlined characteristics of electronics circuits.