Electronic Devices and Circuits

Jimmie J. Cathey

November 22, 2025

1 Circuit Analysis

1.1 Spice Elements

The passive and active circuit elements introduced in the previous section are all avaialable in SPIC modeling; however, the manner of node specification and the voltage and current sense or direction are clarified for each element by Figure 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 t 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)— hae 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 presents the basic format for the element specification statement for each of the elements in Figure 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 therminals 3,0 for the circuit of Figure 2. 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

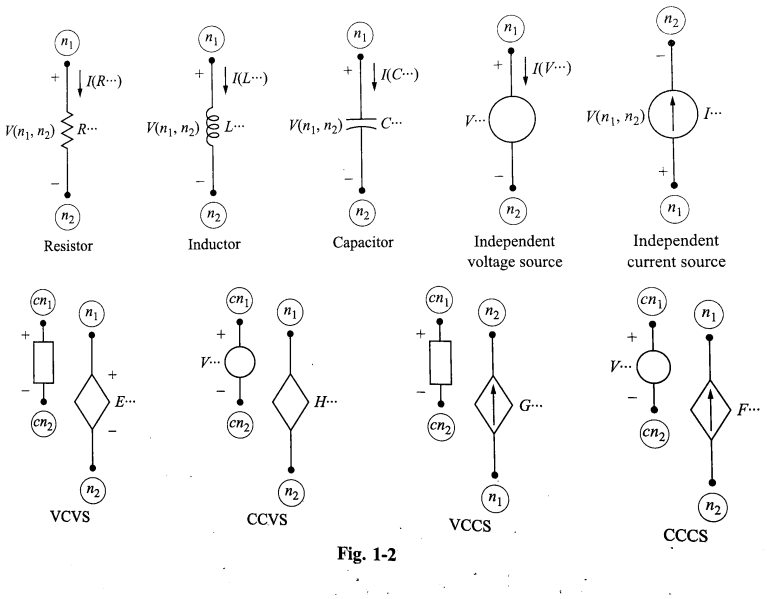


Figure 1:

Table 1:

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···		(cn_1, cn_2)	V/V
CCVS	H···		V···	V/A
VCCS	G···		(cn_1, cn_2)	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 2(a) is replaced by the driving point current source I_dp of Figure 2(b). The netlist code that follows forms a SPICE description

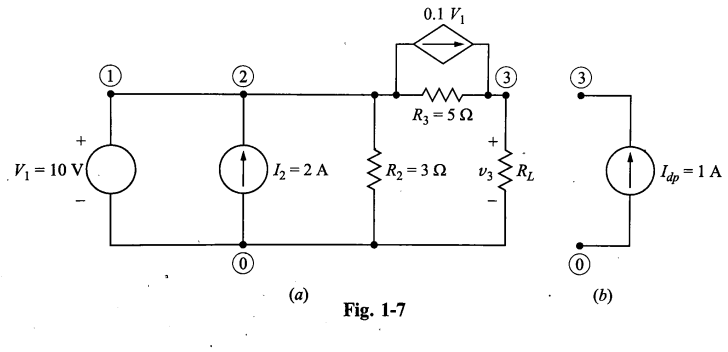


Fig. 1-7

Figure 2:

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
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 Ω.
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.

2 Semiconductor Diodes

2.1 The Ideal Diode

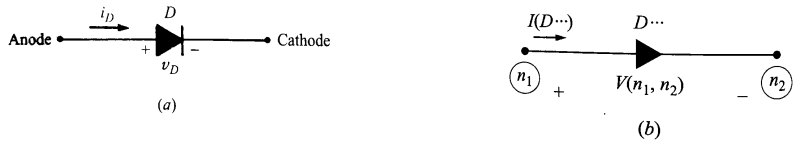


Fig. 2-1

Figure 3:

2.2 Diode Terminal Characteristics

Use of the Fermi-Dirac probability function to predict charge neutralization give the *static* (non-time-varying) equation for diode junction current:

$$i_D = I_0(e^{v_D/\eta V_T} - 1\,A \tag{1}$$

where

$V_T \equiv kT/q$, V

$v_D \equiv$ diode terminal voltage, V

$I_0 \equiv$ temperature-dependent saturation current, A

$T \equiv$ absolute temperature of p-n junction, K

$k \equiv$ Boltzmann's constant (1.38×10^{-23}J/K)

$q \equiv$ electron charge (1.6×10^{-19}C)

$\eta \equiv$ empirical constant, 1 for Ge and 2 for Si

2.3 The Diode SPICE Model

The element specification statement for a diode must explicitly name a model even if the default model parameters are intended for use. The general form of the diode specification statement is a as follows, where the *model name* is arbitrarily chosen:

D$\cdots n_1 n_2$ *model name*

Node n_1 is the anode and node n_2 is the cathode of the diode. Positive current and voltage directions are clarified by Figure 3(b).
In addition, the .MODEL control statement must be added to the netlist code even if the default parameters are acceptable. This control statement is

MODEL *model name* D *(parameters)*

If the parameters field is left blank, default values are assigned. Otherwise, the parameters field contains the number of desired specifications in the format *parameter name = value*. Specific parameters of concern in this book are documented by Table 2.

Table 2:

Parameter	Description	Reference	Default	Units
Is	saturation current	I_0 of (1)	1×10^{-14}	A
n	emission coefficient	η of (1)	1	
BV	reverse breakdown voltage	V_R of Figure **??**	∞	V
IBV	reverse breakdown current	I_R of Figure **??**	1×10^{-10}	A
Rs	ohmic resistance	Subsection2.2)	0	Ω

3 Characteristics of Bipolar Junction Transistors

3.1 BJT SPICE Model

The element specification statement for a BJT must explicitly name a model even if the default model parameters are intended for use. The general form of the transistor specification statement is as follows:

$$\text{Q}\cdots n_1 n_2 n_3 \ \textit{model name}$$

Nodes n_1, n_2, and n_3 belong to the collector, base, and emitter, respectively. The *model name* is an arbitrary selection of alpha and numeric characters to uniquely identify the model. Positive current and voltage directions for the *pnp* and *npn* transistor are clarified by Figure **??**.
In addition, a .MODEL control statement must be added to the netlist code. This control statement specifies whether the transistor is *pnp* or *npn* and thus has one of the following two forms:

MODEL *model name* PNP *(parameters)*
MODEL *model name* NPN *(parameters)*

If the parameter field is left blank, default values are assigned. Non-default desired parameter specifications are entered in the parameter field using the format *parameter name* = *value*. Specific parameters that are of concern in this book are documented in Table 3.
All parameter values are entered with positive values regardless of whether the transistor is *pnp* or *npn*. Two transistor models will be used in this chapter–*generic model* and *default model*–as introduced in Example 3.1.

Example 3.1. Use SPICE methods to generate the CE collector characteristics for an *npn* transistor characterized by (a) the default parameter values and (b) a reasonable set of values for the parameters appearing in Table 3.

```
Ex3_2.CIR
Ib 0 1 0uA
Q 2 1 0 QNPN
*Q 2 1 0 QNPNG
VC 2 0 0V
.MODEL QNPN NPN() ; Default BJT
*.MODEL QNPNG NPN(Is=10fA Ikf=150mA Isc=10fA Bf=150
*+ Br=3 Rb=1ohm Rc=1ohm Va=30V Cjc=10pF Cje=15pf)
.DC VC 0V 15V 1V Ib 0uA 150uA 25uA
.PROBE
.END
```

Table 3:

Parameter	Description	Major Impact	Default	Units
Is	saturation current	$\uparrow$ Is, $\downarrow V_{\mathrm{BEQ}}$	1×10^{-16}	A
Ikf	high current roll-off	$\downarrow$ Ikf, $\downarrow$ $\mathrm{I_C}$	∞	A
Isc	base-collector leakage	$\uparrow$ Isc, $\uparrow$ $\mathrm{I_C}$	0	A
Bf	forward current gain	$\uparrow$ Bf, $\uparrow$ $\mathrm{I_C}$	100	
Br	reverse current gain	$\uparrow$ Br, $\uparrow$ rev. $\mathrm{I_C}$	1	
Rb	base resistance	$\uparrow$ Rb, $\downarrow$ $\mathrm{di_B/dv_{BE}}$	0	Ω
Rc	collector resistance	$\uparrow$ Rc, $\uparrow$ $\mathrm{V_{CEsat}}$	0	Ω
Va	forward Early voltage	$\uparrow$ Va, $\uparrow$ $\mathrm{di_C/dt}$	∞	V
Cjc	base-collector capacitance	high freq. response	0	F
Cje	base-emitter capacitance	high freq. response	0	F

4 Characteristics of Field-Effect Transistors and Triodes

4.1 JFET SPICE Model

The element specification statement for a JFET must explicitly assign a *model name* that is an arbitrary selection of alpha and numeric characters. The general form is

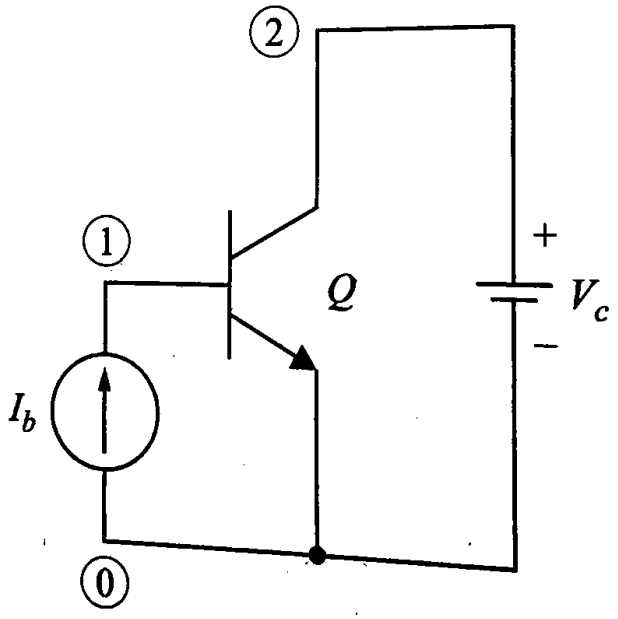
$$\mathrm{J}\cdots n_1 n_2 n_3 \textit{ model name}$$

Nodes n_1, n_2, and n_3 belong to the drain, gate, and source, respectively. Only the n-channel JFET is addressed in this book. Positive voltage and current directions for the device are clarified by Figure 5. A .MODEL control statement must appear in the netlist code for a JFET circuit. The control statement has the following format:

MODEL *model name* NJF *(parameters)*

If the parameter field is left blank, default values are assigned. Nondefault parameters are entered in the parameter field using the format *parameter name = value*. The specific parameters of concern in the book are documented by Table 4. The SPICE model describes the JFET in the pinchoff region by

$$i_d = \frac{I_{DSS}}{(V_{to})^2}(V_{to} + v_{GS})^2 = Beta(V_{to} + v_{GS})^2 \tag{2}$$



(a)

Fig. 3-5

Figure 4:

Table 4:

Parameter	Description	Major Impact	Default	Units
Vto	pinchoff voltage	shorted-gate current	-2	V
Beta	transcond. coeff.	shorted-gate current	0.0001	A/V^2
Rd	drain resistance	current limit	0	Ω
Rs	source resistance	current limit	0	Ω
CGS	gate-source cap.	high frequency	0	F
CGD	gate-drain cap.	high frequency	0	F

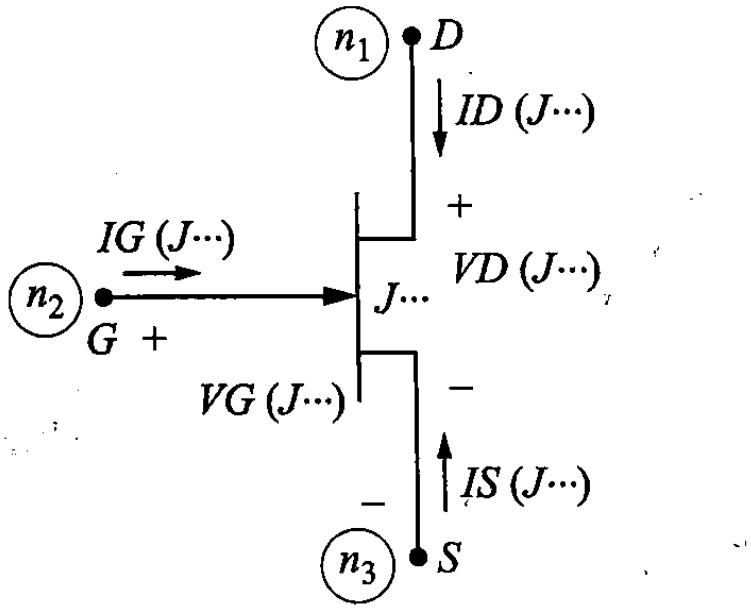


Fig. 4-3

Figure 5:

4.2 MOSFET SPICE Model

The element specification statement for a MOSFET must explicitly assign a *model name* (an arbitrary selection of alpha and numeric characters) having the general form

$$\text{M}\cdots n_1 n_2 n_3 n_4 \textit{ model name}$$

Nodes n_1, n_2, n_3, and n_4 belong to the drain, gate, source, and substrate, respectively. Only the n-channel MOSFET is addressed where the device positive voltage and current directions are clarified by Figure 6. Format of the .MODEL control statement that must appear in the netlist code for a MOSFET circuit is as follows:

MODEL *model name* NMOS *(parameters)*

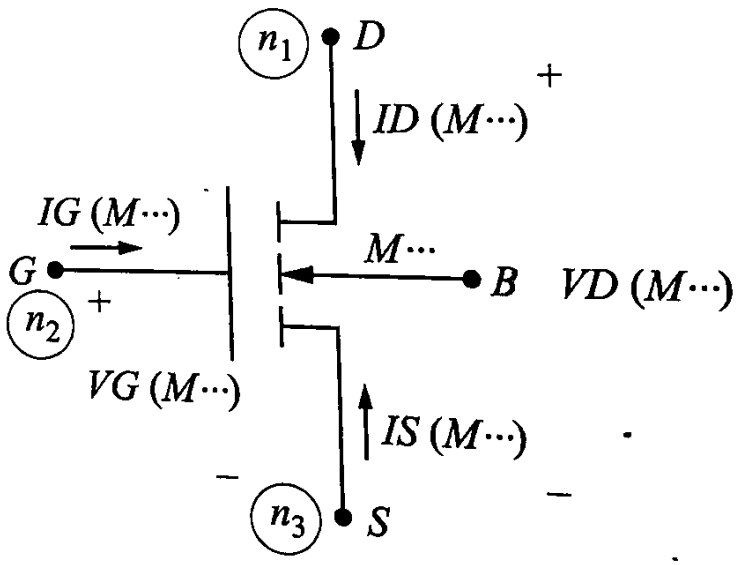


Fig. 4-10

Figure 6:

A blank parameter field results in assignment of default paramter values. Nondefault parameters are entered in the parameter field as *parameter name = value*. The specific parameters of concern in the book are documented by Table 5. The SPICE model characterizes the enhancement mode MOSFET in the pinchoff region by

$$i_d = \frac{I_{D\text{on}}}{(V_T^2}(v_{GS} - V_T)^2 = \frac{Kp}{2}(v_{GS} - V_T)^2 \tag{3}$$

Table 5:

Parameter	Description	Default	Units
Vto	Threshold voltage	0	V
Kp	Transcond. coeff.	2×10^{-5}	A/V^2
Rd	Drain resistance	0	Ω
Rg	Gate resistance	0	Ω

5 Transistor Bias Considerations

5.1 Stability-Factor Analysis

$$S_I = \left.\frac{\Delta I_{CQ}}{\Delta I_{CBO}}\right|_Q \approx \left.\frac{\partial I_{CQ}}{\partial I_{CBO}}\right|_Q \tag{4}$$

5.2 Parameter Variation Analysis with SPICE

PSpice offers two features that allow direct study of circuit performance change due to parameters variation. The first of these features is simply called *sensitivity analysis*. It is invoked by a control statement of the following format:

.SENS *sensitive variable*

The *sensitive variable* can be any node voltage or the current through any independent voltage source. A table is generated in the output file that gives the sensitivity of the sensitive variable to each parameter (specified or default) in the model of all BJTs and diodes that are directly comparable with (4) and

6 Small-Signal Midfrequency BJT Amplifiers

6.1 BJT Amplifier Analysis with SPICE

Since SPICE models of the BJT (see Chapter 3) provide the device terminal characteristics, a transistor amplifier can be properly biased and a time-varying input signal can be directly applied to the completely modeled amplifier circuit. Any desired signal that results can be measured directly in the time domain to form signal ratios that yield the current and voltage gains. With

such modeling, any signal distortion that results from nonlinear operation of the BJT is readily apparent from inspection of the signal-time plots. Such an analysis approach is the analytical equivalent of laboratory operation of the amplifier where the time plot of signals is analogous to oscilloscope observation of the amplifier circuit signals.

7 Small-Signal Midfrequency FET and Triode Amplifiers

7.1 FET Amplifier Gain Calculation with SPICE

SPICE models of the JFET and MOSFET (introduced in Chapter 4) provide the terminal characteristics of the devices; thus, an amplifier can be properly biased and time-varying input signal directly applied to the completely modeled amplifier circuit. Such a simulation is the analytical equivalent of laboratory amplifier circuit operation. Any desired signal can be measured directly in the time domain to form signal ratios that yield current and voltage gains. Any signal distoration that may result from device nonlinearity is readily apparent from inspection of the signal time plots.

8 Frequency Effects in Amplifiers

8.1 Frequency Response Using SPICE

9 Operational Amplifiers

9.1 SPICE Op Amp Model

10 Switched Mode Power Supplies

10.1 Buck Converter

The SMPS circuit of Figure 7, known as a *buck converter*, produces an average value output voltage $V_2 = \langle v_2 \rangle \leq V_1$.

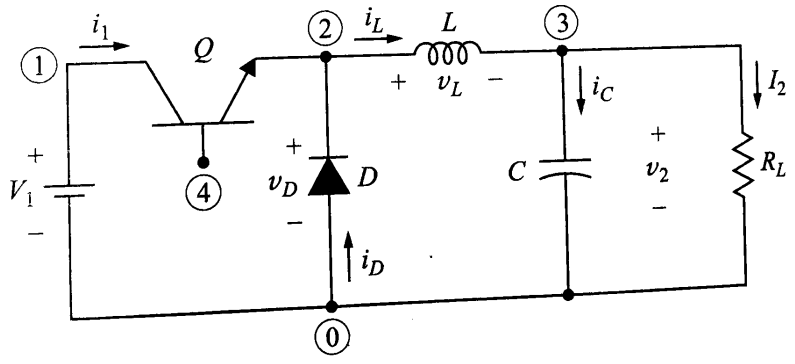


Fig. 10-2 Buck converter

Figure 7: Buck converter

10.2 Boost Converter

The *boost converter* SMPS circuit of Figure 8 produces an average value output voltage $V_2 = \langle v_2 \rangle > V_1$.

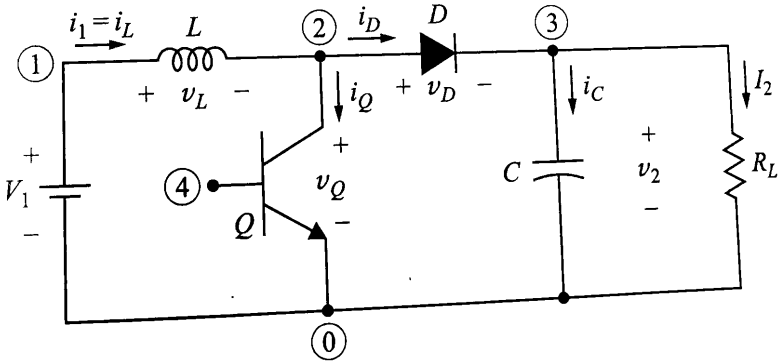


Fig. 10-4 Boost converter

Figure 8: Boost converter

10.3 SPICE Analysis of SMPS

For simulation of near ideal (lossless) SMPS, the switch element Q can readily be modeled using the PSpice voltage-controlled switch. The element specification statement for the voltage-controlled switch has the form

$$\text{S}\cdots n_1\, n_2\, c_1\, c_2 \text{ VCS}$$

Any alpha-numeric combination suffix can follow S to uniquely specify the voltage-controlled switch. The nodes are clarified by Figure **??**. A fast rise and fall time (5 ns), 1-V pulse should be used for the control voltage v_{SW}. Accepting the default ON state and OFF state control voltages of 1 V and 0 V, respectively, results in duty cycle ON time approximately equal to the pulse duration. For minimum conduction losses, the ON state resistance of the voltage control switch should be specified in the .MODEL statement by

$$\text{.MODEL VCS VSWITCH (RON = 1e-6)}$$

Example 10.1. Use SPICE methods to model the buck converter of Figure 7; let $D = 0.5$, $f_s = 25$ kHz, $L = 100\,\mu$H, $C = 50\,\mu$F, and $R_L = 5\,\Omega$. Generate the set of waveforms analogous to Figure **??**.
The netlist code follows, where the initial conditions on inductor current and capacitor voltage were determined after running a large integer number of cycles to find the repetitive values.

```
Ex10_5.CIR
* BUCK CONVERTER
* D=DUTY CYCLE, fs=SWITCHING FREQUENCY
.PARAM D=-.5 fs=25e3Hz
V1 1 0 DC 12V
SW 1 2 4 2 VCS
VSW 4 2 PULSE(0V 1V 0s 5ns 5ns {D/fs} {1/fs})
L 2 3 100uH IC=0.6A
D 0 2 DMOD
C 3 0 50uF IC=6V
RL 3 0 5ohm
.MODEL DMOD D(N=0.01)
.MODEL VCS VSWITCH (RON=1e-6ohm)
.TRAN 5us 0.2ms 0s 100ns UIC
.PROBE
.END
```

Execute Listing 10.1 <Ex10_5.CIR> and use the Probe feature of PSpice to plot the waveforms of Figure 10.1.

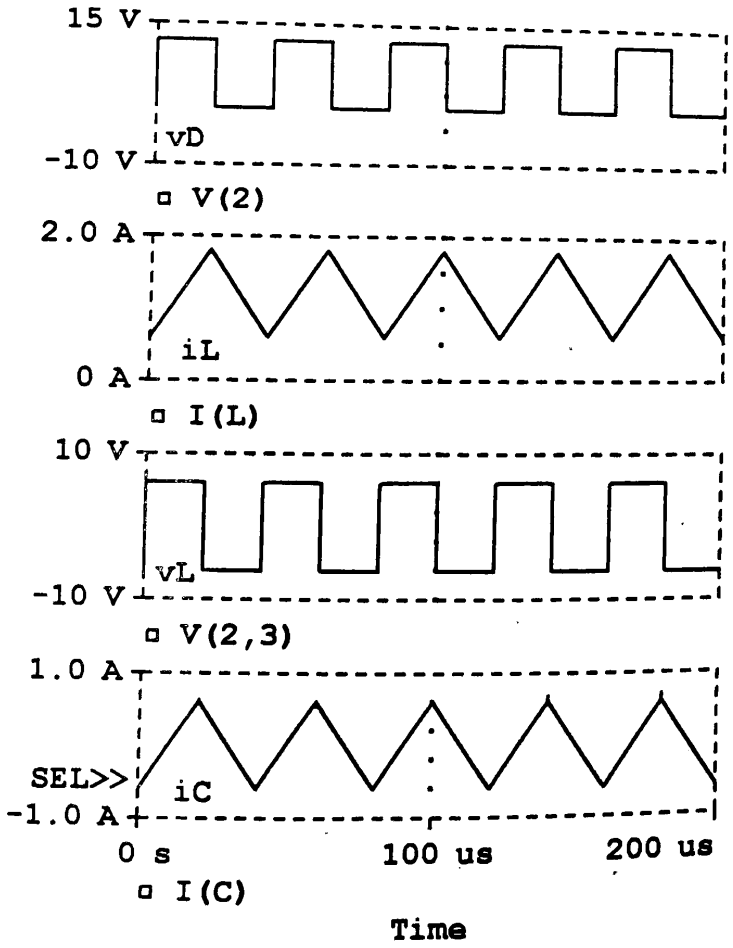

15 V
vD
-10 V
V(2)
2.0 A
iL
0 A
I(L)
10 V
vL
-10 V
V(2,3)
1.0 A
SEL>>
iC
-1.0 A
0 s
100 us
200 us
I(C)
Time

Fig. 10-9