APPENDIX B

SPICE DEVICE MODELS AND SIMULATION EXAMPLES

Introduction

This appendix is concerned with the very important topic of using SPICE to simulate the operation of electronic circuits. We described the need for and the role of computer simulation in circuit design in the preface. This Appendix presents a brief description of the models that SPICE uses to describe the operation of op amps, diodes, MOSFETs, and BJTs. Furthermore, this Appendix is accompanied by design and simulation examples using SPICE simulators, including instructions on how to install and use the simulators, SPICE netlists for the examples, and a summary of the simulation results that can be expected. All of these resources are available via the book website.

Contents

B.1 SPICE Device Models

To the designer, the value of simulation results depends entirely on the quality of the models used for the devices. The more faithfully the models represent the devices' characteristics, the more accurately the simulation results will describe the operation of an actual circuit. Device nonidealities must be included in the device model, otherwise their impact will not appear in the simulation results.

B.1.1 The Op-Amp Model

In simulating circuits that use one or more op amps, a **macromodel** can be used to represent each op amp. A macromodel is based on the observed terminal characteristics of the op amp rather than on the modeling of every transistor in the op-amp internal circuit. Macromodels can be developed from data-sheet specifications without knowing the details of the internal circuitry of the op amp.

Linear Macromodel A linear macromodel for an internally compensated op amp with finite gain and bandwidth is shown in Fig. B.1. In this equivalent-circuit model, the gain constant A_{0d} of the voltage-controlled voltage source E_d corresponds to the differential gain of the op amp at dc. Resistor R_b and capacitor C_b form a single-time-constant (STC) filter with a corner frequency

$$f_b = \frac{1}{2\pi R_b C_b} \tag{B.1}$$

The low-pass response of this filter is used to model the frequency response of the internally compensated op amp. The values of R_b and C_b used in the macromodel are chosen such that f_b corresponds to the 3-dB frequency of the op amp being modeled. This is done by arbitrarily selecting a value for either R_b or C_b (the selected value does not need to be a practical one) and then using Eq. (B.1) to compute the other value. In Fig. B.1, the voltage-controlled voltage source E_b with a gain constant of unity is used as a buffer to isolate the low-pass filter from any load at the op-amp output. Thus any op-amp loading will not affect the frequency response of the filter and hence that of the op amp.

The linear macromodel in Fig. B.1 can be further expanded to account for other op-amp nonidealities. For example, the equivalent-circuit model in Fig. B.2 can be used to model an internally compensated op amp while accounting for the following op-amp nonidealities:

1. **Input Offset Voltage (V_{OS}).** The dc voltage source V_{OS} models the op-amp input offset voltage.

2. **Input Bias Current (I_B) and Input Offset Current (I_{OS}).** The dc current sources I_{B1} and I_{B2} model the input bias current at each input terminal of the op amp, with

$$I_{B1} = I_B + \frac{I_{OS}}{2} \qquad \text{and} \qquad I_{B2} = I_B - \frac{I_{OS}}{2}$$

where I_B and I_{OS} are, respectively, the input bias current and the input offset current specified by the op-amp manufacturer.

3. **Common-Mode Input Resistance (R_{icm}).** If the two input terminals of an op amp are tied together and the input resistance (to ground) is measured, the result is the common-mode input resistance R_{icm}. In the macromodel of Fig. B.2, we have split R_{icm} into two equal parts ($2R_{icm}$), each connected between one of the input terminals and ground.

4. **Differential-Input Resistance (R_{id}).** The resistance seen between the two input terminals of an op amp is the differential input resistance R_{id}.

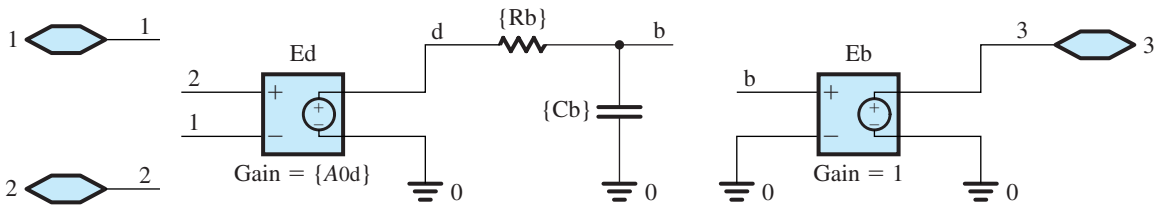


Figure B.1 A linear macromodel used to model the finite gain and bandwidth of an internally compensated op amp.

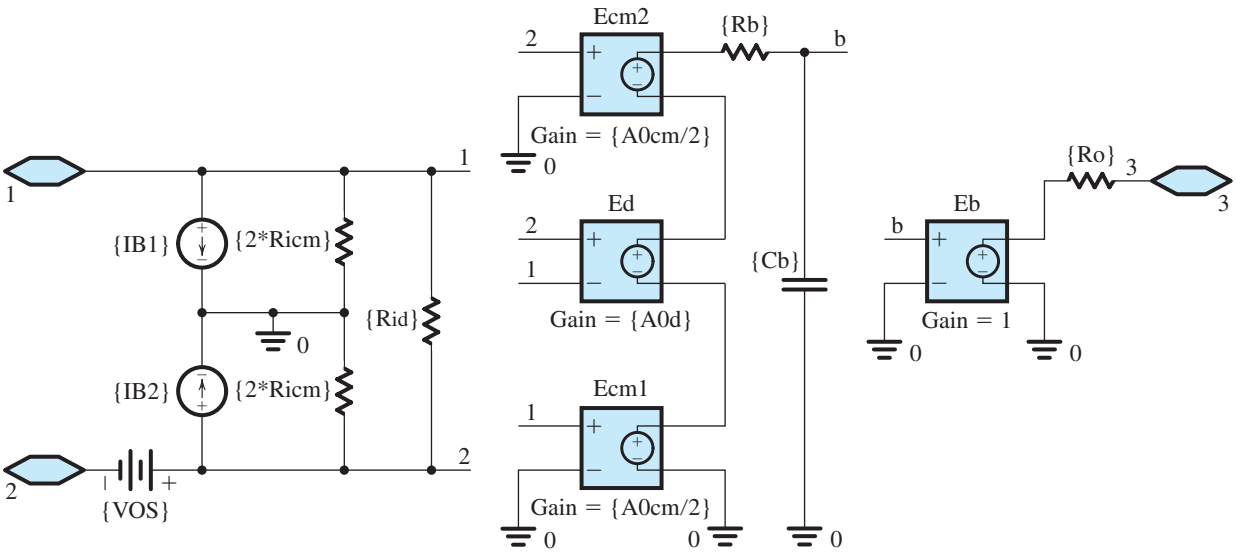


Figure B.2 A comprehensive linear macromodel of an internally compensated op amp.

5. **Differential Gain at DC (A_{0d}) and Common-Mode Rejection Ratio (CMRR).** The output voltage of an op amp at dc can be expressed as

$$V_3 = A_{0d}(V_2 - V_1) + \frac{A_{0cm}}{2}(V_1 + V_2) \tag{B.2}$$

where A_{0d} and A_{0cm} are, respectively, the differential and common-mode gains of the op amp at dc. For an op amp with a finite CMRR,

$$A_{0cm} = A_{0d}/\text{CMRR} \tag{B.3}$$

where CMRR is expressed in V/V (not in dB). In the macromodel of Fig. B.2, the voltage-controlled voltage sources E_{cm1} and E_{cm2} with gain constants of $A_{0cm}/2$ account for the finite CMRR while source E_d models A_{0d}.

6. **Unity-Gain Frequency (f_t).** From Eq. (2.44), the 3-dB frequency f_b and the unity-gain frequency (or gain-bandwidth product) f_t of an internally compensated op amp with an STC frequency response are related by

$$f_b = \frac{f_t}{A_{0d}} \tag{B.4}$$

As in Fig. B.1, the finite op-amp bandwidth is accounted for in the macromodel of Fig. B.2 by setting the corner frequency of the filter formed by resistor R_b and capacitor C_b (Eq. B.1) to equal the 3-dB frequency of the op amp, f_b.

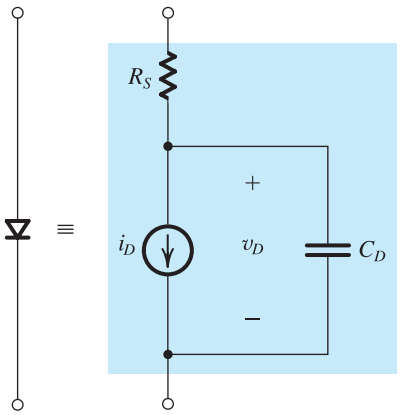
7. **Ouput Resistance (R_o).** The resistance seen at the output terminal of an op amp is the output resistance R_o.

The linear macromodels in Figs. B.1 and B.2 assume that the op-amp circuit is operating in its linear range and do not account for its nonideal performance when large signals are present at the output. Therefore, nonlinear effects, such as output saturation and slew rate, are not modeled.

Nonlinear Macromodel The linear macromodel in Fig. B.2 can be expanded to account for the op-amp nonlinearities. For example, the finite output voltage swing of the op amp can be modeled by placing limits on the output voltage of the voltage-controlled voltage source E_b. Details on how to incorporate this and other nonlinearities into macromodels for the op amp are dependent on the particular simulation tool used and can be found in their manuals. In general, ready-made robust macromodels that account for the nonlinear effects in an IC are provided by the op-amp manufacturers. Macromodels for many popular off-the-shelf ICs are included in many SPICE simulators or are available directly from the IC manufacturers.

B.1.2 The Diode Model

The large-signal SPICE model for the diode is shown in Fig. B.3. The static behavior is modeled by the exponential $i-v$ relationship. Here, for generality, a constant n is included in the exponent. It is known as the **emission cofficient**, and its value ranges from 1 to 2. In our study of the diode in Chapter 4, we assumed $n = 1$. The dynamic behavior is represented by the nonlinear capacitor C_D, which is the sum of the diffusion capacitance C_d and the junction capacitance C_j. The series resistance R_S represents the total resistance of the p and n regions on both sides of the junction. The value of this parasitic resistance is ideally zero, but it is



$$i_D = I_S\left(e^{v_D/nV_T} - 1\right)$$

$$C_D = C_d + C_j = \frac{\tau_T}{V_T} I_S\, e^{v_D/nV_T} + C_{j0}\Big/\left(1 - \frac{v_D}{V_0}\right)^m$$

Figure B.3 The SPICE diode model.

typically in the range of a few ohms for small-signal diodes. For small-signal analysis, SPICE uses the diode incremental resistance r_d and the incremental values of C_d and C_j.

Table B.1 provides a partial listing of the diode-model parameters used by SPICE, all of which should be familiar to the reader. But having a good device model solves only half of the modeling problem; the other half is to determine appropriate values for the model parameters. This is by no means an easy task. The values of the model parameters are determined using a combination of characterization of the device-fabrication process and specific measurements performed on the actual manufactured devices. Semiconductor manufacturers expend enormous effort and money to extract the values of the model parameters for their devices. For discrete diodes, the values of the SPICE model parameters can be determined from the diode data sheets, supplemented if needed by key measurements. Circuit simulators include in their libraries the model parameters of some of the popular off-the-shelf components. For instance, in Example PS4.1, we use the commercially available D1N418 *pn*-junction diode whose SPICE model parameters are readily available.

Table B.1 Parameters of the SPICE Diode Model (Partial Listing)

SPICE Parameter	Book Symbol	Description	Units
IS	I_S	Saturation current	A
N	n	Emission coefficient	
RS	R_S	Ohmic resistance	Ω
VJ	V_0	Built-in potential	V
CJ0	C_j0	Zero-bias depletion (junction) capacitance	F
M	m	Grading coefficient	
TT	τ_T	Transit time	s
BV	V_{ZK}	Breakdown voltage	V
IBV	I_{ZK}	Reverse current at V_{ZK}	A

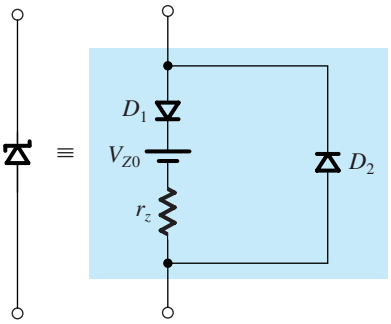


Figure B.4 Equivalent-circuit model used to simulate the zener diode in SPICE. Diode D_1 is ideal and can be approximated in SPICE by using a very small value for n (say $n = 0.01$).

B.1.3 The Zener Diode Model

The diode model in Fig. B.3 does not adequately describe the operation of the diode in the breakdown region. Hence, it does not provide a satisfactory model for zener diodes. However, the equivalent-circuit model shown in Fig B.4 can be used to simulate a zener diode in SPICE. Here, diode D_1 is an ideal diode that can be approximated in SPICE by using a very small value for n (say $n = 0.01$). Diode D_2 is a regular diode that models the forward-bias region of the zener (for most applications, the parameters of D_2 are of little consequence since zener diodes are rarely operated in the forward direction).

B.1.4 MOSFET Models

To simulate the operation of a MOSFET circuit, a simulator requires a mathematical model to represent the characteristics of the MOSFET. The relatively simple model we derived in Chapter 5 to represent the MOSFET is called the **square-law model** because of the quadratic i–v relationship in saturation. It works well for transistors with relatively *long* channels (e.g. over 1 μm). However, for devices with *short* channels, especially deep-submicron transistors, many physical effects that we neglected come into play, with the result that the derived first-order model no longer accurately represents the actual operation of the MOSFET (see Sections 5.4.1 & 17.4.5).

The simple square-law model is useful for understanding the basic operation of the MOSFET as a circuit element and is indeed used to obtain approximate pencil-and-paper circuit designs. However, more elaborate models, which account for short-channel effects, are required to be able to predict the performance of integrated circuits with a certain degree of precision prior to fabrication. Such models have indeed been developed and continue to be refined to more accurately represent the higher-order effects in short-channel transistors through a mix of physical relationships and empirical data. Examples include the Berkeley short-channel IGFET model (BSIM) and the EKV model, popular in Europe. Currently, semiconductor manufacturers select a MOSFET model and extract the values of the model parameters using both their knowledge of the details of the fabrication process and extensive measurements on a variety of fabricated MOSFETs. A lot of effort is expended extracting the model parameter values. The effort pays off when the performance of fabricated circuits is very close to that predicted by simulation, thus reducing the need for costly redesign.

Although the subject of advanced MOSFET modeling and short-channel effects is beyond the scope of this book, it is important to be aware of the limitations of the square-law model and of the availability of more accurate, but more complex, MOSFET models. Thus,

computer simulation becomes even more important when these complex device models are required to accurately analyze and design integrated circuits.

SPICE simulators provide the user with a choice of MOSFET models. The MOSFET model being used is indicated by a parameter called LEVEL. When LEVEL = 1, the simple square-law model (called the Shichman-Hodges model) is used, based on the MOSFET equations presented in Chapter 5. For simplicity, we will use this model to illustrate the description of the MOSFET model parameters in SPICE and to simulate the example circuits in SPICE. However, the reader is again reminded of the need to use a more sophisticated model to accurately predict circuit performance, especially for deep submicron transistors.

MOSFET Model Parameters Table B.2 provides a listing of some of the MOSFET model parameters used in the level-1 model. The reader should already be familiar with these parameters, except for a few, which are described next.

MOSFET Diode Parameters For the two reverse-biased diodes formed between each of the source and drain diffusion regions and the body (see Fig. 10.3), the saturation-current density is modeled in SPICE by the parameter JS. Furthermore, based on the parameters specified in Table B.2, SPICE will calculate the depletion-layer (junction) capacitances discussed in Section 10.1.1 as

$$C_{db} = \frac{\text{CJ}}{\left(1 + \dfrac{V_{DB}}{\text{PB}}\right)^{\text{MJ}}}\text{AD} + \frac{\text{CJSW}}{\left(1 + \dfrac{V_{DB}}{\text{PB}}\right)^{\text{MJSW}}}\text{PD} \tag{B.5}$$

$$C_{sb} = \frac{\text{CJ}}{\left(1 + \dfrac{V_{SB}}{\text{PB}}\right)^{\text{MJ}}}\text{AS} + \frac{\text{CJSW}}{\left(1 + \dfrac{V_{SB}}{\text{PB}}\right)^{\text{MJSW}}}\text{PS} \tag{B.6}$$

where AD and AS are the areas, and PD and PS are the perimeters of, respectively, the drain and source regions of the MOSFET. The first capacitance term in Eqs. (B.5) and (B.6) represents the depletion-layer (junction) capacitance over the bottom of the drain and source regions. The second capacitance term accounts for the depletion-layer capacitance along the sidewall (periphery) of these regions. Both terms are expressed using the formula developed in Section 3.6.1 (Eq. 3.47). The values of AD, AS, PD, and PS must be specified by the user based on the dimensions of the device being used.

MOSFET Dimension and Gate-Capacitance Parameters In a fabricated MOSFET, the effective channel length L_{eff} is shorter than the nominal (or drawn) channel length L (as specified by the designer) because the source and drain diffusion regions extend slightly under the gate oxide during fabrication. Furthermore, the effective channel width W_{eff} of the MOSFET is shorter than the nominal or drawn channel width W because of the sideways diffusion into the channel from the body along the width. In terms of the parameters specified in Table B.2,

$$L_{\text{eff}} = L - 2\text{LD} \tag{B.7}$$

$$W_{\text{eff}} = W - 2\text{WD} \tag{B.8}$$

In a manner analogous to using L_{ov} to denote LD, we will use the symbol W_{ov} to denote WD. Consequently, as indicated in Section 10.1.1, the gate-source capacitance C_{gs} and the

Table B.2 Parameters of the SPICE Level-1 MOSFET Model (Partial Listing)

SPICE Parameter	Book Symbol	Description	Units
		Basic Model Parameters	
LEVEL		MOSFET model selector	
TOX	t_{ox}	Gate-oxide thickness	m
COX	C_{ox}	Gate-oxide capacitance, per unit area	F/m^2
UO	μ	Carrier mobility	$cm^2/V{\cdot}s$
KP	k'	Process transconductance parameter	A/V^2
LAMBDA	λ	Channel-length modulation coefficient	V^{-1}
		Threshold Voltage Parameters	
VTO	V_{t0}	Zero-bias threshold voltage	V
GAMMA	γ	Body-effect parameter	$V^{1/2}$
NSUB	N_A, N_D	Substrate doping	cm^{-3}
PHI	$2\phi_f$	Surface inversion potential	V
		MOSFET Diode Parameters	
JS		Body-junction saturation-current density	A/m^2
CJ		Zero-bias body-junction capacitance, per unit area over the drain/source region	F/m^2
MJ		Grading coefficient, for area component	
CJSW		Zero-bias body-junction capacitance, per unit length along F/m the sidewall (periphery) of the drain/source region	
MJSW		Grading coefficient, for sidewall component	
PB	V_0	Body-junction built-in potential	V
		MOSFET Dimension Parameters	
LD	L_{ov}	Lateral diffusion into the channel from the source/drain diffusion regions	m
WD		Sideways diffusion into the channel from the body along the width	m
		MOS Gate-Capacitance Parameters	
CGBO		Gate-body overlap capacitance, per unit channel length	F/m
CGDO	C_{ov}/W	Gate-drain overlap capacitance, per unit channel width	F/m
CGSO	C_{ov}/W	Gate-source overlap capacitance, per unit channel width	F/m

gate-drain capacitance C_{gd} must be increased by an overlap component of, respectively,

$$C_{gs,ov} = W\,\text{CGSO} \tag{B.9}$$

and

$$C_{gd,ov} = W\,\text{CGDO} \tag{B.10}$$

Similarly, the gate-body capacitance C_{gb} must be increased by an overlap component of

$$C_{gb,ov} = L\,\text{CGBO} \tag{B.11}$$

There is a built-in redundancy in specifying the MOSFET model parameters in SPICE. For example, the user may specify the value of KP for a MOSFET or, alternatively, specify

Table B.3 Values of the Level-1 MOSFET Model Parameters for Exemplar CMOS Technologies[1]

	5-μm CMOS Process		0.5-μm CMOS Process		0.18-μm CMOS Process	
	NMOS	PMOS	NMOS	PMOS	NMOS	PMOS
LEVEL	1	1	1	1	1	1
TOX	8.50e-08	8.50e-08	9.50e-09	9.50e-09	4.08e-09	4.08e-09
UO	750	250	460	115	291	102
LAMBDA	0.01	0.03	0.1	0.2	0.08	0.11
GAMMA	1.4	0.65	0.5	0.45	0.3	0.3
VTO	1	−1	0.7	−0.8	0.5	−0.45
PHI	0.7	0.65	0.8	0.75	0.84	0.8
LD	7.00e-07	6.00e-07	8.00e-08	9.00e-08	10e-9	10e.9
JS	1.00e-06	1.00e-06	1.00e-08	5.00e-09	8.38e-6	4.00e-07
CJ	4.00e-04	1.80e-04	5.70e-04	9.30e-04	1.60e-03	1.00e-03
MJ	0.5	0.5	0.5	0.5	0.5	0.45
CJSW	8.00e-10	6.00e-10	1.20e-10	1.70e-10	2.04e-10	2.04e-10
MJSW	0.5	0.5	0.4	0.35	0.2	0.29
PB	0.7	0.7	0.9	0.9	0.9	0.9
CGBO	2.00e-10	2.00e-10	3.80e-10	3.80e-10	3.80e-10	3.50e-10
CGDO	4.00e-10	4.00e-10	4.00e-10	3.50e-10	3.67e-10	3.43e-10
CGSO	4.00e-10	4.00e-10	4.00e-10	3.50e-10	3.67e-10	3.43e-10

[1] We have created MOSFET models corresponding to the parameters above. They are available online at the textbook website.

TOX and UO and let SPICE compute KP as UO TOX. Similarly, GAMMA can be directly specified, or the physical parameters that enable SPICE to determine it can be specified (e.g., NSUB). In any case, *the user-specified values will always take precedence over (i.e., override) those values calculated by SPICE*. As another example, note that the user has the option of either directly specifying the overlap capacitances CGBO, CGDO, and CGSO or letting SPICE compute them as CGDO = CGSO = LD COX and CGBO = WD COX.

Table B.3 provides typical values for the level-1 MOSFET model parameters of a modern 0.18-μm CMOS technology and for older 0.5-μm and 5-μm CMOS technologies. The corresponding values for the minimum channel length $L_{\min}$, minimum channel width $W_{\min}$, and the maximum supply voltage $\left(V_{DD} + \left|V_{SS}\right|\right)_{\max}$ are as follows:

Technology	$L_{\min}$	$W_{\min}$	$(V_{DD} + \lvert V_{SS} \rvert)_{\max}$
5-μm CMOS	5 μm	12.5 μm	10 V
0.5-μm CMOS	0.5 μm	1.25 μm	3.3 V
0.18-μm CMOS	0.18 μm	0.22 μm	1.8 V

When simulating a MOSFET circuit, the user needs to specify both the values of the model parameters and the dimensions of each MOSFET in the circuit being simulated. At least the channel length L and width W must be specified. The areas AD and AS and the perimeters PD and PS need to be specified for SPICE to model the body-junction capacitances (otherwise, zero capacitances would be assumed). The exact values of these geometry parameters depend on the actual layout of the device (Appendix A). However, to estimate these dimensions, we will assume that a metal contact is to be made to each of the source and

drain regions of the MOSFET. For this purpose, typically, these diffusion regions must be extended *past* the end of the channel (i.e., in the L-direction in Fig. 5.1) by at least 2.75 L_{min}. Thus, the minimum area and perimeter of a drain/source diffusion region with a contact are, respectively,

$$\text{AD} = \text{AS} = 2.75 L_{\text{min}} W \tag{B.12}$$

and

$$\text{PD} = \text{PS} = 2 \times 2.75 L_{\text{min}} + W \tag{B.13}$$

Unless otherwise specified, we will use Eqs. (B.12) and (B.13) to estimate the dimensions of the drain/source regions in our examples.

Finally, we note that SPICE computes *the values for the parameters of the MOSFET small-signal model based on the dc operating point (bias point)*. These are then used by SPICE to perform the small-signal analysis (called "ac analysis").

B.1.5 The BJT Model

SPICE uses a general form of the BJT model that we discussed in Chapter 6 (Fig. 6.5). Known as the *transport* form of the **Ebers–Moll model**, it is shown in Fig. B.5. Here, the currents of the base–emitter diode (D_{BE}) and the base–collector diode (D_{BC}) are given, respectively, by

$$i_{BE} = \frac{I_S}{\beta_F}\left(e^{v_{BE}/n_F V_T} - 1\right) \tag{B.14}$$

and

$$i_{BC} = \frac{I_S}{\beta_R}\left(e^{v_{BC}/n_R V_T} - 1\right) \tag{B.15}$$

where n_F and n_R are the emission coefficients of the BEJ and BCJ, respectively. These coefficients are generalizations of the constant n of the *pn*-junction diode (Fig. B.3). (We have so far assumed $n_F = n_R = 1$). The parameters β_F and β_R are, respectively, the forward and reverse β of the BJT. The reverse β is the current gain obtained when the collector and emitter are interchanged and is much smaller than the forward β. In fact, $\beta_R \ll 1$.

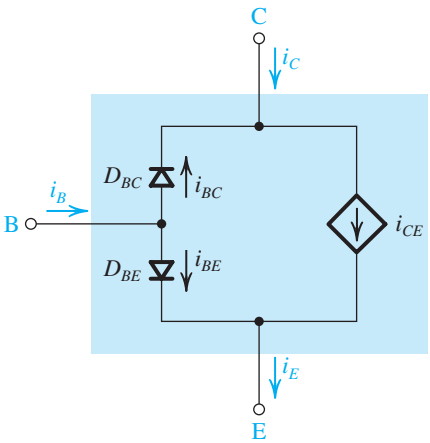


Figure B.5 The transport form of the Ebers–Moll model for an *npn* BJT.

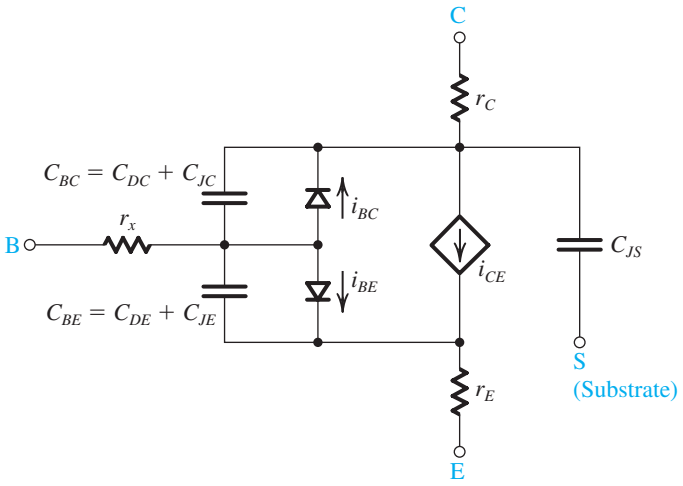


Figure B.6 The SPICE large-signal model for an *npn* BJT.

The controlled current-source i_{CE} in the transport model is defined as

$$i_{CE} = I_S\left(e^{v_{BE}/n_F V_T} - e^{v_{BC}/n_R V_T}\right) \tag{B.16}$$

Observe that i_{CE} represents the current component of i_C and i_E that arises as a result of the minority carrier diffusion across the base, or **carrier transport** across the base (hence the name transport model).

The transport model can account for the Early effect in a forward-biased BJT by including the factor $(1 - v_{BC}/V_A)$ in the expression for the transport current i_{CE} as follows:

$$i_{CE} = I_S\left(e^{v_{BE}/n_F V_T} - e^{v_{BC}/n_R V_T}\right)\left(1 - \frac{v_{BC}}{V_A}\right) \tag{B.17}$$

Figure B.6 shows the model used in SPICE. Here, resistors r_x, r_E, and r_C are added to represent the ohmic resistance of, respectively, the base, emitter, and collector regions. The dynamic operation of the BJT is modeled by two nonlinear capacitors, C_{BC} and C_{BE}. Each of these capacitors generally includes a diffusion component (i.e., C_{DC} and C_{DE}) and a depletion or junction component (i.e., C_{JC} and C_{JE}) to account for the charge-storage effects within the BJT (as described in Section 10.1.2). Furthermore, the BJT model includes a depletion junction capacitance C_{JS} to account for the collector–substrate junction in integrated-circuit BJTs, where a reverse-biased *pn* junction is formed between the collector and the substrate (which is common to all components of the IC).

For small-signal (ac) analysis, the SPICE BJT model is equivalent to the hybrid-π model of Fig. 7.26, but augmented with r_E, r_C, and (for IC BJTs) C_{JS}. Furthermore, the model includes a large resistance r_μ between the base and collector (in parallel with C_μ) to account for the dependence of i_1 on v_{CB}. The resistance r_μ is very large, typically greater than $10\beta r_o$.

Although Fig. B.6 shows the SPICE model for the *npn* BJT, the corresponding model for the *pnp* BJT can be obtained by reversing the direction of the currents and the polarity of the diodes and terminal voltages.

The SPICE Gummel–Poon Model of the BJT The BJT model described above lacks a representation of some second-order effects present in actual devices. One of the most important such effects is the variation of the current gains, β_F and β_R, with the current i_C. The Ebers–Moll model assumes β_F and β_R to be constant, thereby neglecting their current dependence (as depicted in Fig. 6.20). To account for this, and other second-order effects, SPICE uses a more accurate, yet more complex, BJT model called the Gummel–Poon model (named after H. K. Gummel and H. C. Poon, two pioneers in this field). This model is based on the relationship between the electrical terminal characteristics of a BJT and its base charge. It is beyond the scope of this book to delve into the model details. However, it is important for the reader to be aware of the existence of such a model.

In SPICE, the Gummel–Poon model automatically simplifies to the Ebers–Moll model when certain model parameters are not specified. Consequently, the BJT model to be used by SPICE need not be explicitly specified by the user (unlike the MOSFET case in which the model is specified by the LEVEL parameter). For discrete BJTs, the values of the SPICE model parameters can be determined from the data specified on the BJT data sheets, supplemented (if needed) by key measurements. For instance, in Example S.6.1, we will use the Q2N3904 *npn* BJT (from Fairchild Semiconductor) whose SPICE model is readily available. In fact, most SPICE simulators already include the SPICE model parameters for many of the commercially available discrete BJTs. For IC BJTs, the values of the SPICE model parameters are determined by the IC manufacturer (using both measurements on the fabricated devices and knowledge of the details of the fabrication process) and are provided to IC designers.

The SPICE BJT Model Parameters Table B.4 provides a listing of some of the BJT model parameters used in SPICE. The reader should be already familiar with these parameters. In the absence of a user-specified value for a particular parameter, SPICE uses a default value that typically results in the corresponding effect being ignored. For example, if no value is specified for the forward Early voltage (VAF), SPICE assumes that VAF $= \infty$ and does not account for the Early effect. Although ignoring VAF can be a serious issue in some circuits, the same is not true, for example, for the value of the reverse Early voltage (VAR).

The BJT Model Parameters BF and BR in SPICE Before leaving the SPICE model, a comment on β is in order. SPICE interprets the user-specified model parameters BF and BR as the *ideal maximum* values of the forward and reverse dc current gains, respectively, versus the operating current. These parameters are not equal to the constant-current-independent parameters $\beta_F(\beta_{dc})$ and β_R used in the Ebers–Moll model for the forward and reverse dc current gains of the BJT. SPICE uses a current-dependent model for β_F and β_R, and the user can specify other parameters (not shown in Table B.4) for this model. Only when such parameters are not specified, and the Early effect is neglected, will SPICE assume that β_F and β_R are constant and equal to BF and BR, respectively. Furthermore, SPICE computes values for both β_{dc} and β_{ac}, the two parameters that we generally assume to be approximately equal. SPICE then uses β_{ac} to perform small-signal (ac) analysis.

Table B.4 Parameters of the SPICE BJT Model (Partial Listing)

SPICE Parameter	Book Symbol	Description	Units
IS	I_S	Saturation current	A
BF	β_F	Ideal maximum forward current gain	
BR	β_R	Ideal maximum reverse current gain	
NF	n_F	Forward current emission coefficient	
NR	n_R	Reverse current emission coefficient	
VAF	V_A	Forward Early voltage	V
VAR		Reverse Early voltage	V
RB	r_x	Zero-bias base ohmic resistance	Ω
RC	r_C	Collector ohmic resistance	Ω
RE	r_E	Emitter ohmic resistance	Ω
TF	τ_F	Ideal forward transit time	s
TR	τ_R	Ideal reverse transit time	s
CJC	$C_{\mu 0}$	Zero-bias base–collector depletion (junction) capacitance	F
MJC	m_{BCJ}	Base–collector grading coefficient	
VJC	V_{0c}	Base–collector built-in potential	V
CJE	C_{je0}	Zero-bias base–emitter depletion (junction) capacitance	F
MJE	m_{BEJ}	Base–emitter grading coefficient	
VJE	V_{0e}	Base–emitter built-in potential	V
CJS		Zero-bias collector–substrate depletion (junction) capacitance	F
MJS		Collector–substrate grading coefficient	
VJS		Collector–substrate built-in potential	V

B.2 SPICE Examples

Example S.2.1

Performance of a Noninverting Amplifier

Consider an op amp with a differential input resistance of 2 MΩ, an input offset voltage of 1 mV, a dc gain of 100 dB, and an output resistance of 75 Ω. Assume the op amp is internally compensated and has an STC frequency response with a gain–bandwidth product of 1 MHz.

(a) Create a subcircuit model for this op amp in SPICE.
(b) Using this subcircuit, simulate the closed-loop noninverting amplifier in Fig. 2.12 with resistors $R_1 =$ 1 kΩ and $R_2 = 100$ kΩ to find:
  (i) Its 3-dB bandwidth $f_{3\text{dB}}$.
  (ii) Its output offset voltage V_{OSout}.
  (iii) Its input resistance R_{in}.
  (iv) Its output resistance R_{out}.
(c) Simulate the step response of the closed-loop amplifier, and measure its rise time t_r. Verify that this time agrees with the 3-dB frequency measured above.

Example S.2.1 *continued*

Solution

To model the op amp in SPICE, we use the equivalent circuit in Fig. B.2, but with $R_{id} = 2\ \text{M}\Omega$, $R_{icm} = \infty$ (open circuit), $I_{B1} = I_{B2} = 0$ (open circuit), $V_{OS} = 1\ \text{mV}$, $A_{0d} = 10^5\ \text{V/V}$, $A_{0cm} = 0$ (short circuit), and $R_o = 75\ \Omega$. Furthermore, we set $C_b = 1\ \mu\text{F}$ and $R_b = 15.915\ \text{k}\Omega$ to achieve an $f_t = 1\ \text{MHz}$.

To measure the 3-dB frequency of the closed-loop amplifier, we apply a 1-V ac voltage at its input, perform an ac-analysis simulation in SPICE, and plot its output versus frequency. The output voltage, plotted in Fig. B.7, corresponds to the gain of the amplifier because we chose an input voltage of 1 V. Thus, from Fig. B.7, the closed-loop amplifier has a dc gain of $G_0 = 100.9\ \text{V/V}$, and the frequency at which its gain drops to $G_0/\sqrt{2} = 71.35\ \text{V/V}$ is $f_{3\text{dB}} = 9.9\ \text{kHz}$, which agrees with Eq. (B.7).

The input resistance R_{in} corresponds to the reciprocal of the current drawn out of the 1-V ac voltage source used in the above ac-analysis simulation at 0.1 Hz. (Theoretically, R_{in} is the small-signal input resistance at dc. However, ac-analysis simulations must start at frequencies greater than zero, so we use 0.1 Hz to approximate the dc point.) Accordingly, R_{in} is found to be $2\ \text{G}\Omega$.

To measure R_{out}, we short-circuit the amplifier input to ground, inject a 1-A ac current at its output, and perform an ac-analysis simulation. R_{out} corresponds to the amplifier output voltage at 0.1 Hz and is found to be $76\ \text{m}\Omega$. Although an ac test voltage source could equally well have been used to measure the output resistance in this case, it is a good practice to attach a current source rather than a voltage source between the output and ground. This is because an ac current source appears as an open circuit when the simulator computes the dc bias point of the circuit while an ac voltage source appears as a short circuit, which can

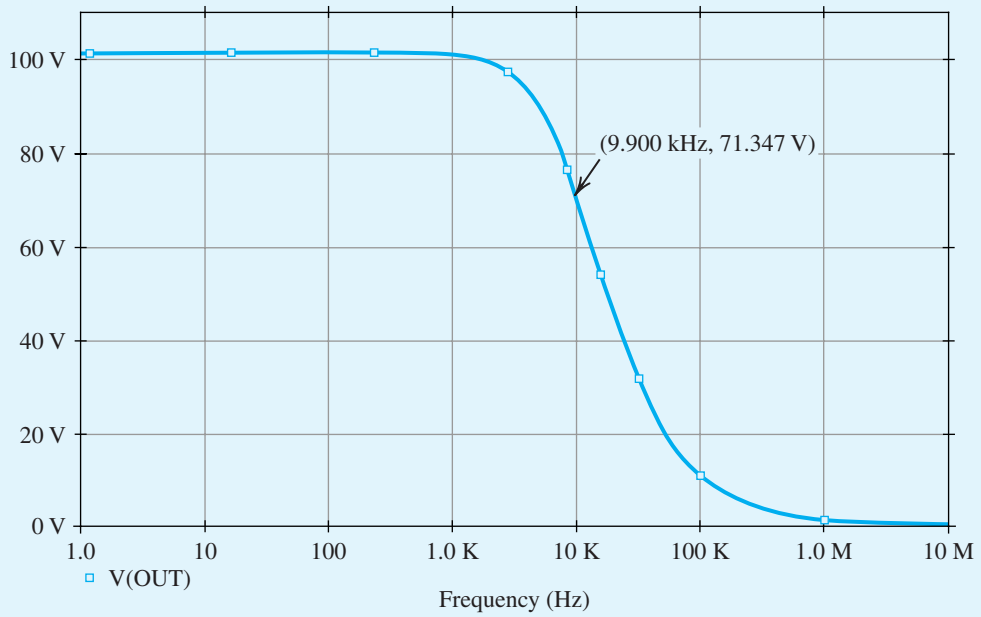


Figure B.7 Frequency response of the closed-loop amplifier in Example S.2.1.

erroneously force the dc output voltage to zero. For similar reasons, an ac test voltage source should be attached in series with the biasing dc voltage source for measuring the input resistance of a voltage amplifier.

A careful look at R_{in} and R_{out} of the closed-loop amplifier reveals that their values have, respectively, increased and decreased by a factor of about 1000, relative to the corresponding resistances of the op amp. Such a large input resistance and small output resistance are indeed desirable characteristics for a voltage amplifier. This improvement in the small-signal resistances of the closed-loop amplifier is a direct consequence of applying negative feedback (through resistors R_1 and R_2) around the open-loop op amp. We study negative feedback in Chapter 11, where we also learn how the improvement factor (1000 in this case) corresponds to the ratio of the open-loop op-amp gain (10^5) to the closed-loop amplifier gain (100).

From Eqs. (2.53) and (2.51), the closed-loop amplifier has an STC low-pass response given by

$$\frac{V_o(s)}{V_i(s)} = \frac{G_0}{1 + \dfrac{s}{2\pi f_{3\text{dB}}}}$$

As described in Appendix E, the response of such an amplifier to an input step of height V_{step} is given by

$$v_O(t) = V_{final}\left(1 - e^{-t/\tau}\right) \tag{B.18}$$

where $V_{final} = G_0 V_{step}$ is the final output-voltage value (i.e., the voltage value toward which the output is heading) and $\tau = 1/(2\pi f_{3\text{dB}})$ is the time constant of the amplifier. If we define $t_{10\%}$ and $t_{90\%}$ to be the time it takes for the output waveform to rise to, respectively, 10% and 90% of V_{final}, then from Eq. (B.18), $t_{10\%} \simeq 0.1\tau$ and $t_{90\%} \simeq 2.3\tau$. Therefore, the rise time t_r of the amplifier can be expressed as

$$t_r = t_{90\%} - t_{10\%} = 2.2\tau = \frac{2.2}{2\pi f_{3\text{dB}}}$$

Therefore, if $f_{3\text{dB}} = 9.9$ kHz, then $t_r = 35.4\ \mu$s. To simulate the step response of the closed-loop amplifier, we apply a step voltage at its input, using a piecewise-linear (PWL) source (with a very short rise time); then perform a transient-analysis simulation, and measure the voltage at the output versus time. In our simulation, we applied a 1-V step input, plotted the output waveform in Fig. B.8, and measured t_r to be 35.3 μs.

The linear macromodels in Figs. B.1 and B.2 assume that the op-amp circuit is operating in its linear range; they do not account for its nonideal performance when large signals are present at the output. Therefore, nonlinear effects, such as output saturation and slew rate, are not modeled. This is why, in the step response of Fig. B.8, we could see an output voltage of 100 V when we applied a 1-V step input. However, IC op amps are not capable of producing such large output voltages. Hence, a designer must be very careful when using these models.

It is important to point out that we also saw output voltages of 100 V or so in the ac analysis of Fig. B.7, where for convenience we applied a 1-V ac input to measure the gain of the closed-loop amplifier. So, would we see such large output voltages if the op-amp macromodel accounted for nonlinear effects (particularly output saturation)? The answer is yes, because in an ac analysis SPICE uses a linear model for nonlinear devices with the linear-model parameters evaluated at a bias point. Thus, we must keep in mind that the voltage magnitudes encountered in an ac analysis may not be realistic. In this case, the voltage and current ratios (e.g., the output-to-input voltage ratio as a measure of voltage gain) are of importance to the designer.

Example S.2.1 *continued*

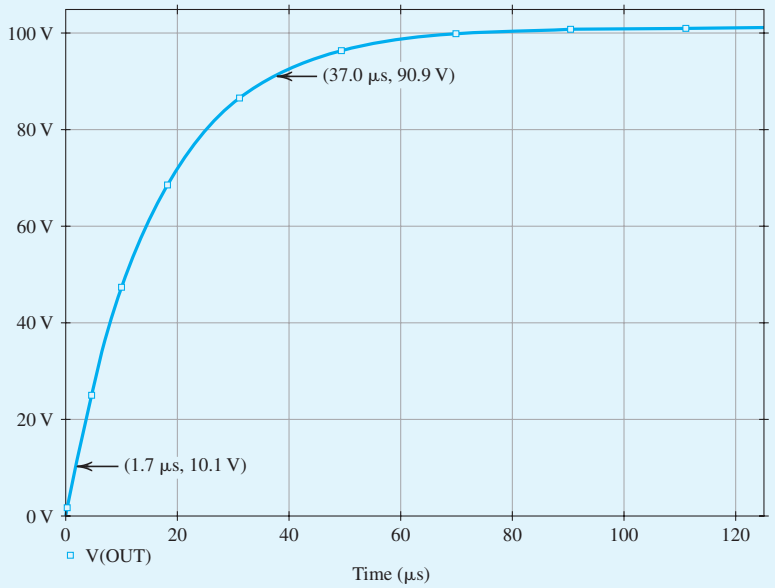


Figure B.8 Step response of the closed-loop amplifier in Example S.2.1.

Example S.2.2

Characteristics of the 741 Op Amp

Consider the µA741 op amp whose macromodel is available in SPICE. Use SPICE to plot the open-loop gain and hence determine f_t. Also, investigate the SR limitation and the output saturation of this op amp.

Solution

Figure B.9 shows the schematic capture used to simulate the frequency response of the µA741 op amp.[1] The µA741 part has seven terminals. Terminals 7 and 4 are, respectively, the positive and negative dc power-supply terminals of the op amp. The 741-type op amps are typically operated from ±15-V power supplies; therefore we connected the dc voltage sources $V_{CC} = +15$ V and $V_{EE} = -15$ V to terminals 7 and 4, respectively. Terminals 3 and 2 of the µA741 part correspond to the positive and negative input terminals, respectively, of the op amp. In general, as outlined in Section 2.1.3, the op-amp input signals are expressed as

$$v_{INP} = V_{CM} + \frac{V_d}{2}$$

$$v_{INN} = V_{CM} - \frac{V_d}{2}$$

[1]The reader is reminded that the netlist files of all SPICE examples in this book can be found on the text's website.

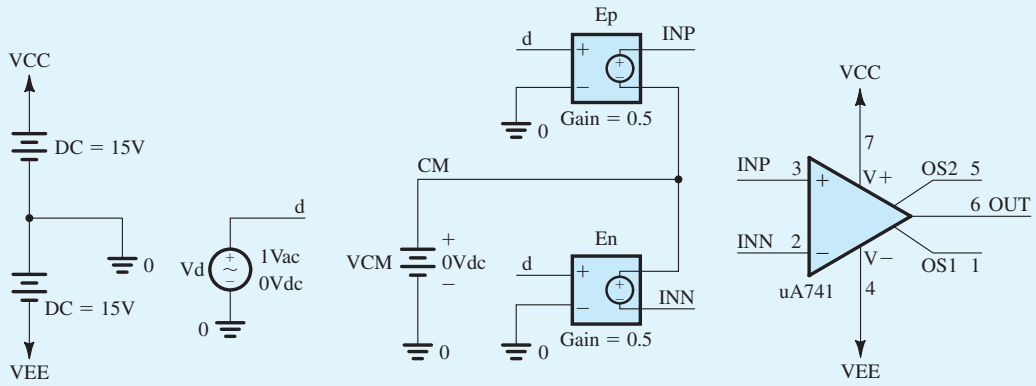


Figure B.9 Simulating the frequency response of the μA741 op-amp in Example S.2.2.

where v_{INP} and v_{INN} are the signals at, respectively, the positive- and negative-input terminals of the op amp with V_{CM} being the common-mode input signal (which sets the dc bias voltage at the op-amp input terminals) and V_d being the differential input signal to be amplified. The dc voltage source V_{CM} in Fig. B.9 is used to set the common-mode input voltage. Typically, V_{CM} is set to the average of the dc power-supply voltages V_{CC} and V_{EE} to maximize the available input signal swing. Hence, we set $V_{CM} = 0$. The voltage source V_d in Fig. B.9 is used to generate the differential input signal V_d. This signal is applied differentially to the op-amp input terminals using the voltage-controlled voltage sources E_p and E_n, whose gain constants are set to 0.5.

Terminals 1 and 5 of part μA741 are the offset-nulling terminals of the op amp (as depicted in Fig. 2.37). The offset-nulling characteristic of the op amp is not incorporated in this macromodel.

To measure f_t of the op amp, we set the voltage of source V_d to be 1-V ac, perform an ac-analysis simulation in SPICE, and plot the output voltage versus frequency as shown in Fig. B.10. Accordingly, the frequency at which the op-amp voltage gain drops to 0 dB is $f_t = 0.9$ MHz (which is close to the 1-MHz value reported in the data sheets for 741-type op amps).

To determine the slew rate of the μA741 op amp, we connect the op amp in a unity-gain configuration, as shown in Fig. B.11, apply a large pulse signal at the input with very short rise and fall times to cause slew-rate limiting at the output, perform a transient-analysis simulation in SPICE, and plot the output voltage as shown in Fig. B.12. The slope of the slew-rate limited output waveform corresponds to the slew-rate of the op amp and is found to be SR = 0.5 V/μs (which agrees with the value specified in the data sheets for 741-type op amps).

To determine the maximum output voltage of the μA741 op amp, we set the dc voltage of the differential voltage source V_d in Fig. B.9 to a large value, say +1 V, and perform a bias-point simulation in SPICE. The corresponding dc output voltage is the positive-output saturation voltage of the op amp. We repeat the simulation with the dc differential input voltage set to −1 V to find the negative-output saturation voltage. Accordingly, we find that the μA741 op amp has a maximum output voltage $V_{o\max} = 14.8$ V.

In these schematics (as shown in Fig. B.13), we use variable parameters to enter the values of the various circuit components. This allows one to investigate the effect of changing component values by simply changing the corresponding parameter values.

Example S.2.2 *continued*

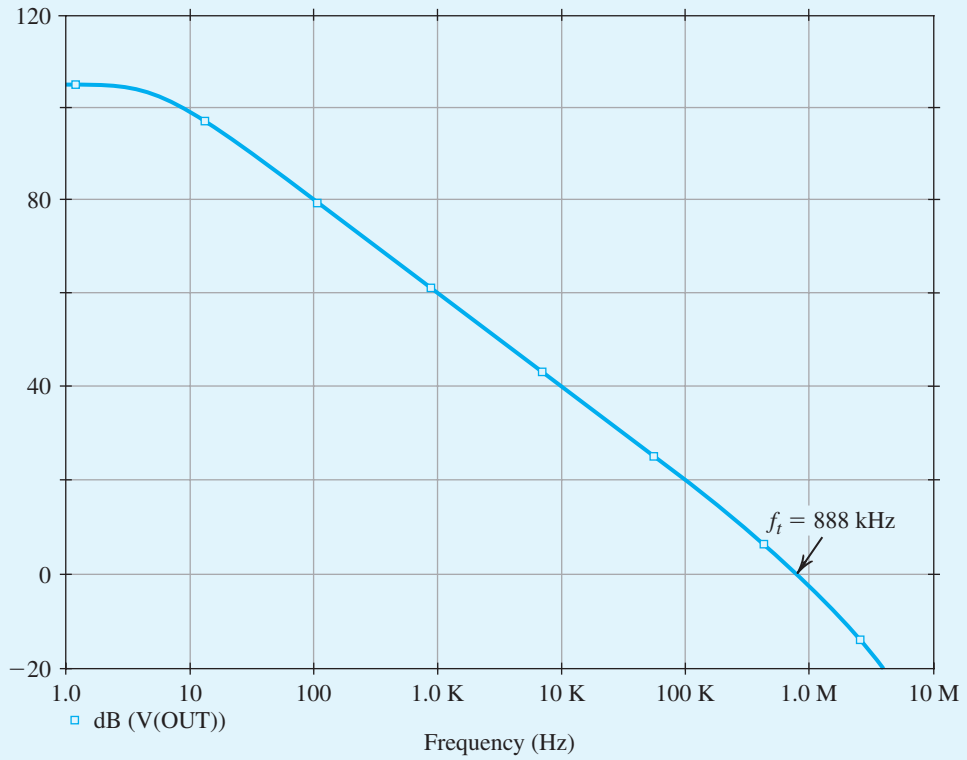


Figure B.10 Frequency response of the μA741 op amp in Example S.2.2.

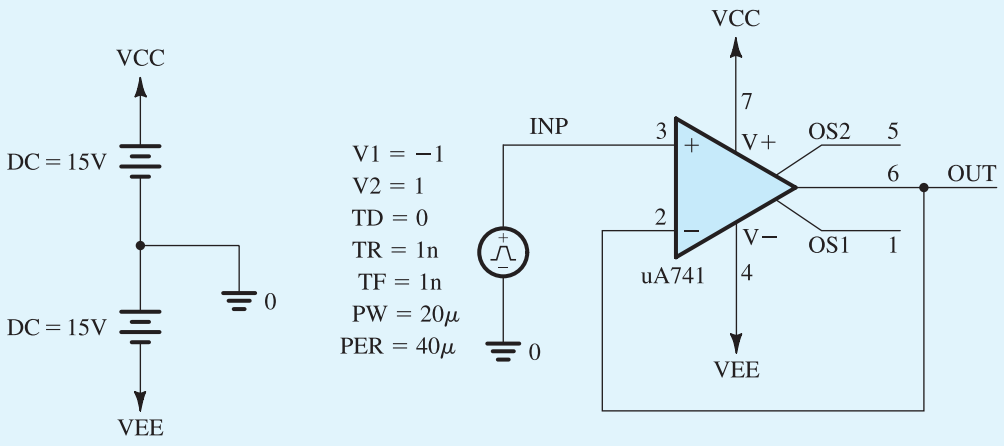


Figure B.11 Circuit for determining the slew rate of the μA741 op amp in Example S.2.2.

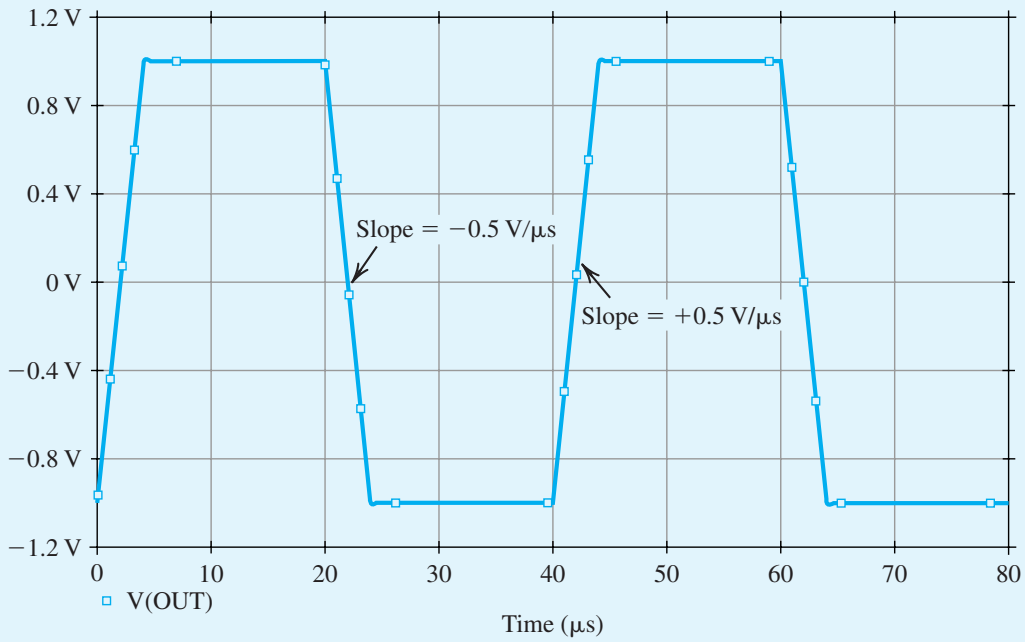


Figure B.12 Square-wave response of the μA741 op amp connected in the unity-gain configuration shown in Fig. B.11.

Example S.4.1

Design of a DC Power Supply

In this example, we will design a dc power supply using the rectifier circuit whose capture schematic is shown in Fig. B.13. This circuit consists of a full-wave diode rectifier, a filter capacitor, and a zener voltage regulator. The only perhaps puzzling component is the $R_{\text{isolation}}$, the 100-MΩ resistor between the secondary winding of the transformer and ground. This resistor is included to provide dc continuity and thus "keep SPICE happy"; it has little effect on circuit operation.

Let it be required that the power supply (in Fig. B.13) provide a nominal dc voltage of 5 V and be able to supply a load current I_{load} as large as 25 mA; that is, R_{load} can be as low as 200 Ω. The power supply is fed from a 120-V (rms) 60-Hz ac line. Note that in the SPICE schematic (Fig. B.13), we use a sinusoidal voltage source with a 169-V peak amplitude to represent the 120-V rms supply (as 120-V rms = 169-V peak). Assume the availability of a 5.1-V zener diode having $r_z = 10\,\Omega$ at $I_Z = 20$ mA (and thus $V_{Z0} = 4.9$ V), and that the required minimum current through the zener diode is $I_{Z\text{min}} = 5$ mA.

An approximate first-cut design can be obtained as follows: The 120-V (rms) supply is stepped down to provide 12-V (peak) sinusoids across each of the secondary windings using a 14:1 turns ratio for the

Example S.4.1 *continued*

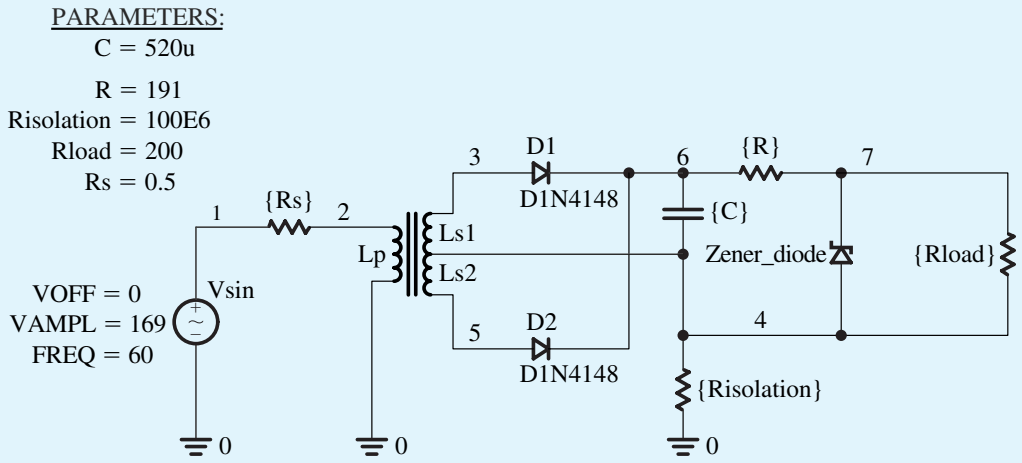


Figure B.13 Schematic capture of the 5-V dc power supply in Example S.4.1.

center-tapped transformer. The choice of 12 V is a reasonable compromise between the need to allow for sufficient voltage (above the 5-V output) to operate the rectifier and the regulator, while keeping the PIV ratings of the diodes reasonably low. To determine a value for R, we can use the following expression:

$$R = \frac{V_{C\min} - V_{Z0} - r_z I_{Z\min}}{I_{Z\min} + I_{L\max}}$$

where an estimate for $V_{C\min}$, the minimum voltage across the capacitor, can be obtained by subtracting a diode drop (say, 0.8 V) from 12 V and allowing for a ripple voltage across the capacitor of, say, $V_r = 0.5$ V. Thus, $V_{S\,\min} = 10.7$ V. Furthermore, we note that $I_{L\,\max} = 25$ mA and $I_{Z\min} = 5$ mA, and that $V_Z 0 = 4.9$ V and $r_z = 10\,\Omega$. The result is that $R = 191\,\Omega$.

Next, we determine C using a restatement of Eq. (4.33) with V_p/R replaced by the current through the 191-Ω resistor. This current can be estimated by noting that the voltage across C varies from 10.7 V to 11.2 V, and thus has an average value of 10.95 V. Furthermore, the desired voltage across the zener is 5 V. The result is $C = 520\,\mu$F.

Now, with an approximate design in hand, we can proceed with the SPICE simulation. For the zener diode, we use the model of Fig. B.4, and assume (arbitrarily) that D_1 has $I_S = 100$ pA and $n = 0.01$ while D_2 has $I_S = 100$ pA and $n = 1.7$. For the rectifier diodes, we use the commercially available 1N4148 type (with $I_S = 2.682$ nA, $n = 1.836$, $R_S = 0.5664\,\Omega$, $V_0 = 0.5$ V, $C_{j0} = 4$ pF, $m = 0.333$, $\tau_T = 11.54$ ns, $V_{ZK} = 100$ V, $I_{ZK} = 100\,\mu$A).

In SPICE, we perform a transient analysis and plot the waveforms of both the voltage v_C across the smoothing capacitor C and the voltage v_O across the load resistor R_{load}. The simulation results for $R_{\text{load}} = 200\,\Omega$ ($I_{\text{load}} \simeq 25$ mA) are presented in Fig. B.14. Observe that v_C has an average of 10.85 V and a ripple of ± 0.21 V. Thus, $V_1 = 0.42$ V, which is close to the 0.5-V value that we would expect from the chosen value of C. The output voltage v_O is very close to the required 5 V, with v_O varying between 4.957 V and 4.977 V for a ripple of only 20 mV. The variations of v_O with R_{load} are illustrated in Fig. B.15

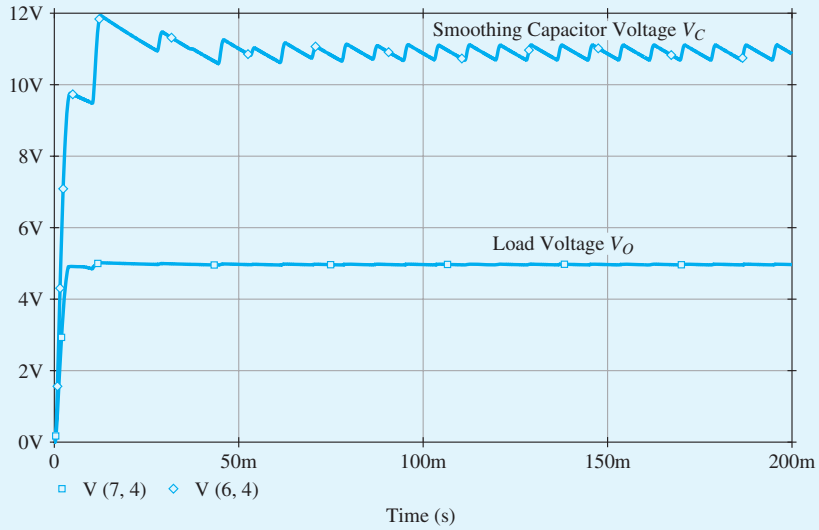


Figure B.14 The voltage v_c across the smoothing capacitor C and the voltage v_o across the load resistor $R_{load} = 200\,\Omega$ in the 5-V power supply of Example S.4.1.

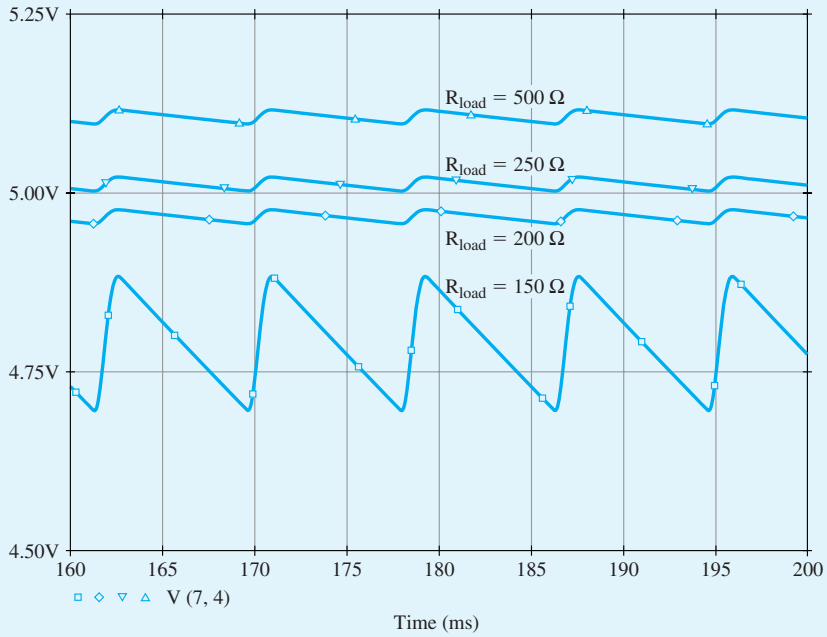


Figure B.15 The output-voltage waveform from the 5-V power supply (in Example S.4.1) for various load resistances: $R_{load} = 500\,\Omega$, $250\,\Omega$, $200\,\Omega$, and $150\,\Omega$. The voltage regulation is lost at a load resistance of $150\,\Omega$.

Example S.4.1 *continued*

for $R_{load} = 500\,\Omega$, $250\,\Omega$, $200\,\Omega$, and $150\,\Omega$. Accordingly, v_O remains close to the nominal value of 5 V for R_{load} as low as $200\,\Omega$($I_{load} \simeq 25$ mA). For $R_{load} = 150\,\Omega$ (which implies $I_{load} \simeq 33.3$ mA, greater than the maximum designed value), we see a significant drop in v_O (to about 4.8 V), as well as a large increase in the ripple voltage at the output (to about 190 mV). This is because the zener regulator is no longer operational; the zener has in fact cut off.

We conclude that the design meets the specifications, and we can stop here. Alternatively, we may consider using further runs of SPICE to help with the task of fine-tuning the design. For instance, we could consider what happens if we use a lower value of C, and so on. We can also investigate other properties of the present design (e.g., the maximum current through each diode) and ascertain whether this maximum is within the rating specified for the diode.

EXERCISE

B.1 Use SPICE to investigate the operation of the voltage doubler whose schematic capture is shown in Fig. EB.16(a). Specifically, plot the transient behavior of the voltages v_2 and v_{OUT} when the input is a sinusoid of 10-V peak and 1-kHz frequency. Assume that the diodes are of the 1N4148 type (with $I_S = 2.682$ nA, $n = 1.836$, $R_S = 0.5664\,\Omega$, $V_0 = 0.5$ V, $C_j0 = 4$ pF, $m = 0.333$, $\tau_T = 11.54$ ns, $V_{ZK} = 100$ V, $I_{ZK} = 100\,\mu$A).

Ans. The voltage waveforms are shown in Fig. B.16(b).

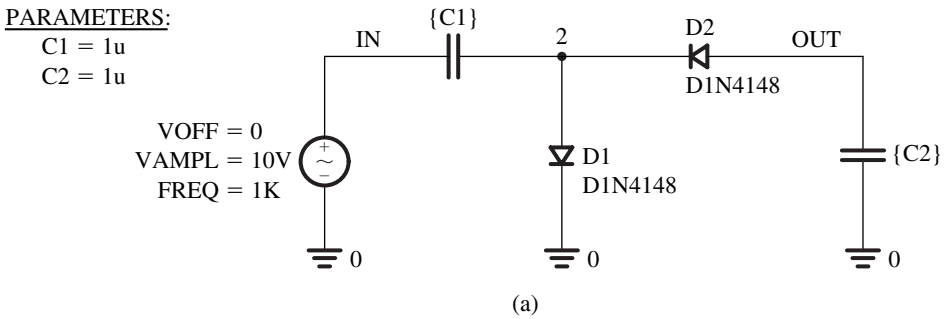
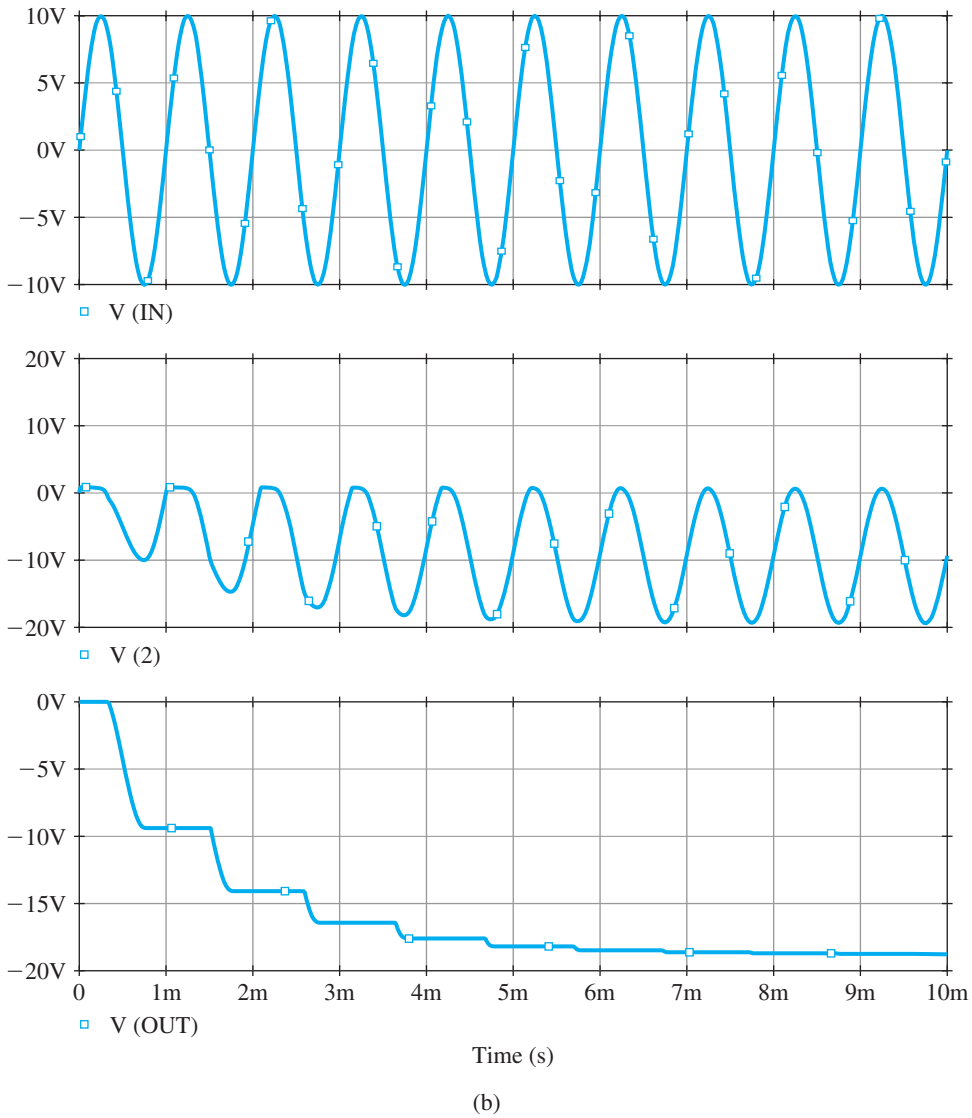


Figure EB.16 **(a)** Schematic capture of the voltage-doubler circuit in Exercise B.1. **(b)** Various voltage waveforms in the voltage-doubler circuit. The top graph displays the input sine-wave voltage signal, the middle graph displays the voltage across diode D_1, and the bottom graph displays the voltage that appears at the output.



(b)

Figure EB.16 *continued*

Example S.6.1

Dependence of the BJT β on the Bias Current

In this example, we use SPICE to simulate the dependence of β_{dc} on the collector bias current for the Q2N3904 discrete BJT (from Fairchild Semiconductor) whose model parameters are listed in Table B.5 and are available in SPICE. As shown in the schematic capture of Fig. B.17, the V_{CE} of the BJT is fixed using a constant voltage source (in this example, $V_{CE}=2$ V) and a dc current source I_B is applied at the base. To illustrate the dependence of β_{dc} on the collector current I_C, we perform a dc-analysis simulation in which the sweep variable is the current source I_B. The β_{dc} of the BJT, which corresponds to the ratio of the collector current I_C to the base current I_B, can then be plotted versus I_C, as shown in Fig. B.18. We see that to operate at the maximum value of β_{dc} (i.e., $\beta_{dc}=163$), at $V_{CE}=2$ V, the BJT must be biased at an $I_C=10$ mA. Since increasing the bias current of a transistor increases the power dissipation, it is clear from Fig. B.18 that the choice of current I_C is a trade-off between the current gain β_{dc} and the power dissipation. Generally speaking, the optimum I_C depends on the application and technology in hand. For example, for the Q2N3904 BJT operating at $V_{CE}=2$ V, decreasing I_C by a factor of 20 (from 10 mA to 0.5 mA) results in a drop in β_{dc} of about 25% (from 163 to 123).

Table B.5 Spice Model Parameters of the Q2N3904 Discrete BJT

IS=6.734F	XTI=3	EG=1.11	VAF=74.03	BF=416.4	NE=1.259	ISE=6.734F
IKF=66.78M	XTB=1.5	BR=.7371	NC=2	ISC=0	IKR=0	RC=1
CJC=3.638P	MJC=.3085	VJC=.75	FC=.5	CJE=4.493P	MJE=.2593	VJE=.75
TR=239.5N	TF=301.2P	ITF=.4	VTF=4	XTF=2	RB=10	

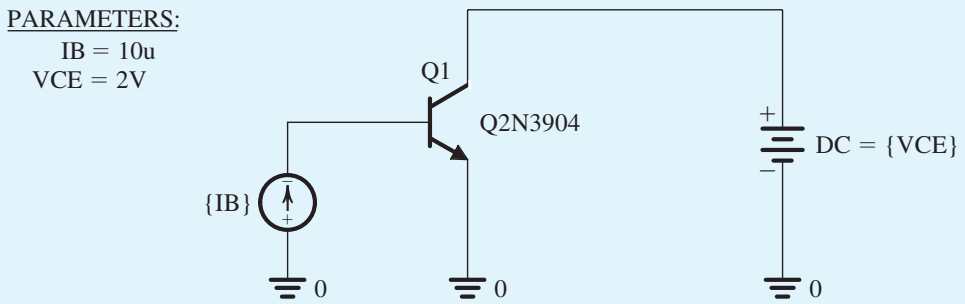


Figure B.17 The SPICE test bench used to demonstrate the dependence of β_{dc} on the collector bias current I_C for the Q2N3904 discrete BJT (Example S.6.1).

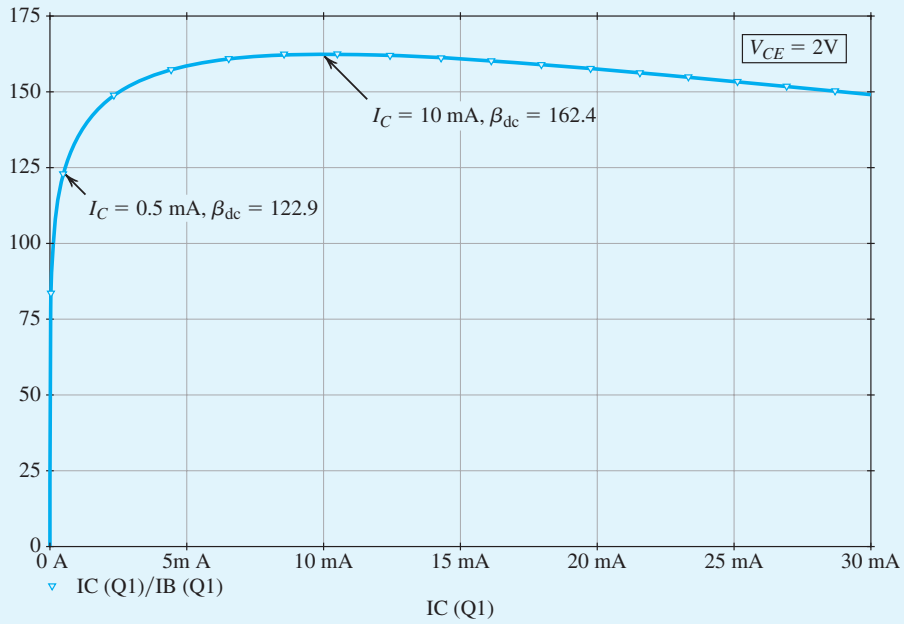


Figure B.18 Dependence of β_{dc} on I_C (at $V_{CE} = 2$ V) in the Q2N3904 discrete BJT (Example S.6.1).

Example S.7.1

The CS Amplifier

In this example, we will use SPICE to analyze and verify the design of the CS amplifier whose capture schematic is shown in Fig. B.17. Observe that the MOSFET has its source and body connected in order to cancel the body effect. We will assume a 0.5-μm CMOS technology for the MOSFET and use the SPICE level-1 model parameters listed in Table B.3. We will also assume a signal-source resistance $R_{sig} = 10\,\text{k}\Omega$, a load resistance $R_L = 50\,\text{k}\Omega$, and bypass and coupling capacitors of 10 μF. The targeted specifications for this CS amplifier are a midband gain $A_M = 10$ V/V and a maximum power consumption $P = 1.5$ mW. As should always be the case with computer simulation, we will begin with an approximate pencil-and-paper design. We will then use SPICE to fine-tune our design and to investigate the performance of the final design. In this way, maximum advantage and insight can be obtained from simulation.

With a 3.3-V power supply, the drain current of the MOSFET must be limited to $I_D = P/V_{DD} = 1.5\,\text{mW}/3.3\ \text{V} = 0.45\,\text{mA}$ to meet the power consumption specification. Choosing $V_{OV} = 0.3$ V (a typical value in low-voltage designs) and $V_{DS} = V_{DD}/3$ (to achieve a large signal swing at the output), the MOSFET can now be sized as

$$\frac{W}{L_{\text{eff}}} = \frac{I_D}{\frac{1}{2}k_n' V_{OV}^2(1+\lambda V_{DS})} = \frac{0.45\times 10^{-3}}{\frac{1}{2}(170.1\times 10^{-6})(0.3)^2[1+0.1(1.1)]} \simeq 53 \tag{B.19}$$

Example S.7.1 *continued*

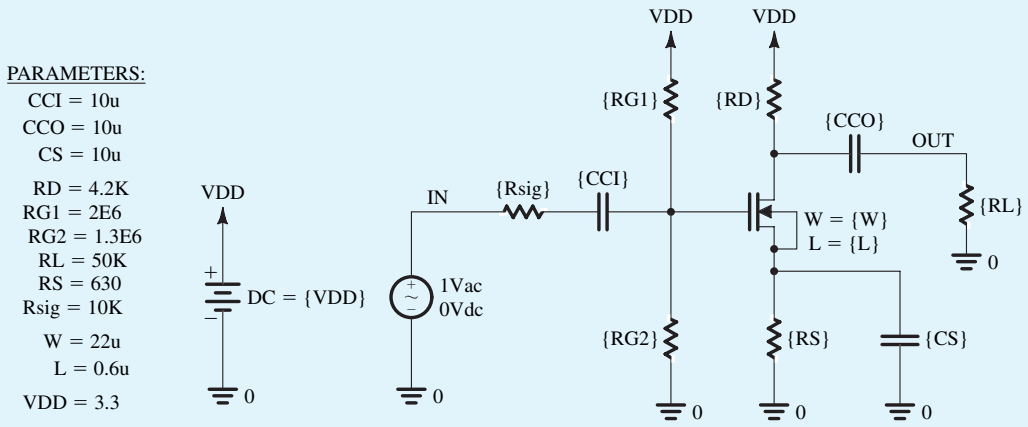


Figure B.19 Schematic capture of the CS amplifier in Example S.7.1.

where $k_n' = \mu_n C_{ox} = 170.1\,\mu\text{A/V}^2$ (from Table B.3). Here, L_{eff} rather than L is used to more accurately compute I_D. The effect of using W_{eff} rather than W is much less important because typically $W \gg W_{ov}$. Thus, choosing $L = 0.6\,\mu\text{m}$ results in $L_{\text{eff}} = L - 2L_{ov} = 0.44\,\mu\text{m}$ and $W = 23.3\,\mu\text{m}$. Note that we chose L slightly larger than L_{min}. This is a common practice in the design of analog ICs to minimize the effects of fabrication nonidealities on the actual value of L. As shown in the text, this is particularly important when the circuit performance depends on the matching between the dimensions of two or more MOSFETs (e.g., in the current-mirror circuits studied in Chapter 8).

Next, R_D is calculated based on the desired voltage gain:

$$|A_v| = g_m(R_D \| R_L \| r_o) = 10\,\text{V/V} \Rightarrow R_D \simeq 4.2\,\text{k}\Omega \tag{B.20}$$

where $g_m = 3.0$ mA/V and $r_o = 22.2\,\text{k}\Omega$. Hence, the output bias voltage is $V_O = V_{DD} - I_D R_D = 1.39$ V. An $R_S = (V_O - V_{DD}/3)/I_D = 630\,\Omega$ is needed to bias the MOSFET at a $V_{DS} = V_{DD}/3$. Finally, resistors $R_{G1} = 2\,\text{M}\Omega$ and $R_{G2} = 1.3\,\text{M}\Omega$ are chosen to set the gate bias voltage at $V_G = I_D R_S + V_{OV} + V_{tn} \simeq 1.29$ V. Using large values for these gate resistors ensures that both their power consumption and the loading effect on the input signal source are negligible. Note that we neglected the body effect in the expression for V_G to simplify our hand calculations.

We will now use SPICE to verify our design and investigate the performance of the CS amplifier. We begin by performing a bias-point simulation to verify that the MOSFET is properly biased in the saturation region and that the dc voltages and currents are within the desired specifications. Based on this simulation, we have decreased the value of W to $22\,\mu\text{m}$ to limit I_D to about 0.45 mA. Next, to measure the midband gain A_M and the 3-dB frequencies[4] f_L and f_H, we apply a 1-V ac voltage at the input, perform an ac-analysis simulation, and plot the output-voltage magnitude (in dB) versus frequency as shown in Fig. B.20. This corresponds to the magnitude response of the CS amplifier because we chose a 1-V input signal.[5] Accordingly, the midband gain is $A_M = 9.55$ V/V and the 3-dB bandwidth is $BW = f_H - f_L \simeq 122.1$ MHz. Figure B.20 further shows that the gain begins to fall off at about 300 Hz but flattens out again at about 10 Hz. This flattening in the gain at low frequencies is due to a real

transmission zero[6] introduced in the transfer function of the amplifier by R_S together with C_S. This zero occurs at a frequency $f_Z = 1/(2\pi R_S C_S) = 25.3$ Hz, which is typically between the break frequencies f_{P2} and f_{P3} derived in Section 10.8.2. So, let us now verify this phenomenon by resimulating the CS amplifier with a $C_S = 0$ (i.e., removing C_S) in order to move f_Z to infinity and remove its effect. The corresponding frequency response is plotted also in Fig. B.20. As expected, with $C_S = 0$, we do not observe any flattening in the low-frequency response of the amplifier. However, because the CS amplifier now includes a source resistor R_S, A_M has dropped by a factor of 2.6. This factor is approximately equal to $(1 + g_m R_S)$, as expected from our study of the CS amplifier with a source-degeneration resistance in Section 7.3.4. Note that the bandwidth BW has increased by approximately the same factor as the drop in gain A_M. As we will learn in Chapter 11 when we study negative feedback, the source-degeneration resistor R_S provides negative feedback, which allows us to trade off gain for wider bandwidth.

To conclude this example, we will demonstrate the improved bias stability achieved when a source resistor R_S is used. Specifically, we will change (in the MOSFET level-1 model for part NMOS0P5) the value of the zero-bias threshold voltage parameter VTO by $\pm 15\%$ and perform a bias-point simulation in SPICE. Table B.6 shows the corresponding variations in I_D and V_O for the case in which $R_S = 630\,\Omega$. For the case without source degeneration, we use an $R_S = 0$ in the schematic of Fig. B.19. Furthermore, to obtain the same I_D and V_O in both cases (for the nominal threshold voltage $V_{t0} = 0.7$ V), we use an $R_{G2} = 0.88$ MΩ to reduce V_G to around $V_{OV} + V_{tn} = 1$ V. The corresponding variations in the bias point are shown in Table B.6. Accordingly, we see that the source-degeneration resistor makes the bias point of the CS amplifier less sensitive to changes in the threshold voltage. In fact, the reader can show for the values displayed in Table B.6 that the variation in bias current ($\Delta I/I$) is reduced by approximately the same factor, $(1 + g_m R_S)$. However, unless a large bypass capacitor C_S is used, this reduced sensitivity comes at the expense of a reduction in the midband gain (as we observed in this example when we simulated the frequency response of the CS amplifier with a $C_S = 0$).

Table B.6 Variations in the Bias Point with the MOSFET Threshold Voltage

	$R_S = 630\,\Omega$		$R_S = 0$	
V_{tn0}	I_D (mA)	V_O (V)	I_D (mA)	V_O (V)
0.60	0.56	0.962	0.71	0.33
0.7	0.46	1.39	0.45	1.40
0.81	0.36	1.81	0.21	2.40

[4]No detailed knowledge of frequency-response calculations is required for this example; all that is needed is Section 7.5.1. Nevertheless, after the study of the frequency response of the CS amplifier in Sections 10.1 through 10.3, the reader will benefit by returning to this example and using SPICE to experiment further with the circuit.

[5]The reader should not be alarmed about the use of such a large signal amplitude. Recall that in a small-signal (ac) simulation, SPICE first finds the small-signal equivalent circuit at the bias point and then analyzes this linear circuit. Such ac analysis can, of course, be done with any ac signal amplitude. However, a 1-V ac input is convenient to use because the resulting ac output corresponds to the voltage gain of the circuit.

[6]Readers who have not yet studied poles and zeros can skip these few sentences.

Example S.7.1 *continued*

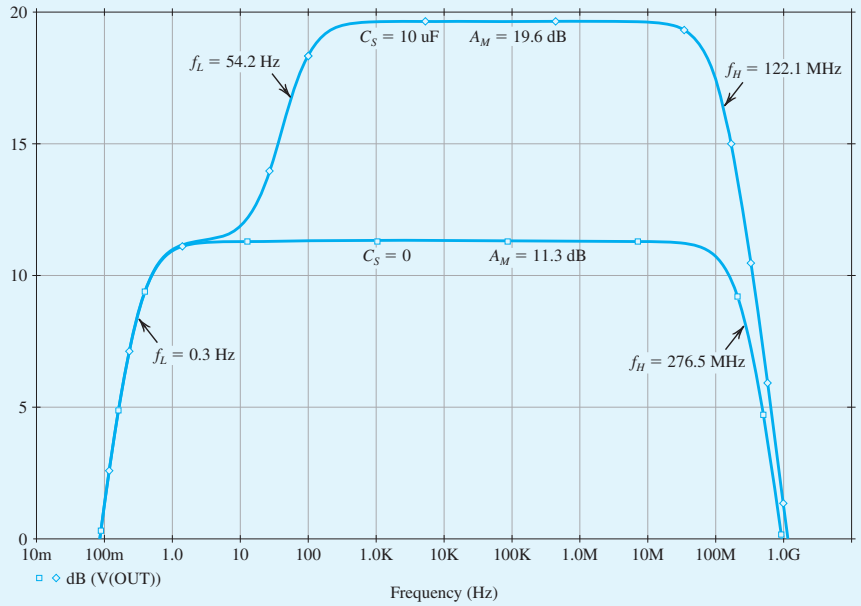


Figure B.20 Frequency response of the CS amplifier in Example S.7.1 with $C_S = 10\,\mu\text{F}$ and $C_S = 0$ (i.e., C_S removed).

Example S.7.2

The CE Amplifier with Emitter Resistance

In this example, we use SPICE to analyze and verify the design of the CE amplifier. A schematic capture of the CE amplifier is shown in Fig. B.23. We will use part Q2N3904 for the BJT and a ±5-V power supply. We will also assume a signal source resistor $R_{\text{sig}} = 10\,\text{k}\Omega$, a load resistor $R_L = 10\,\text{k}\Omega$, and bypass and coupling capacitors of 10 μF. To enable us to investigate the effect of including a resistance in the signal path of the emitter, a resistor R_{ce} is connected in series with the emitter bypass capacitor C_E. Note that the roles of R_E and R_{ce} are different. Resistor R_E is the **dc emitter-degeneration resistor** because it appears in the dc path between the emitter and ground. It is therefore used to help stabilize the bias point for the amplifier. The equivalent resistance $R_e = R_E \| R_{ce}$ is the **small-signal emitter-degeneration resistance** because it appears in the ac (small-signal) path between the emitter and ground and helps stabilize the gain of the amplifier. In this example, we will investigate the effects of both R_E and R_e on the performance of the CE amplifier. However, as should always be the case with computer simulation, we will begin with an approximate pencil-and-paper design. In this way, maximum advantage and insight can be obtained from simulation.

Based on the plot of β_{dc} versus I_C in Fig. B.20, a collector bias current I_C of 0.5 mA is selected for the BJT, resulting in $\beta_{\text{dc}} = 123$. This choice of I_C is a reasonable compromise between power dissipation and current gain. Furthermore, a collector bias voltage V_C of 0 V (i.e., at the mid–supply rail) is selected to

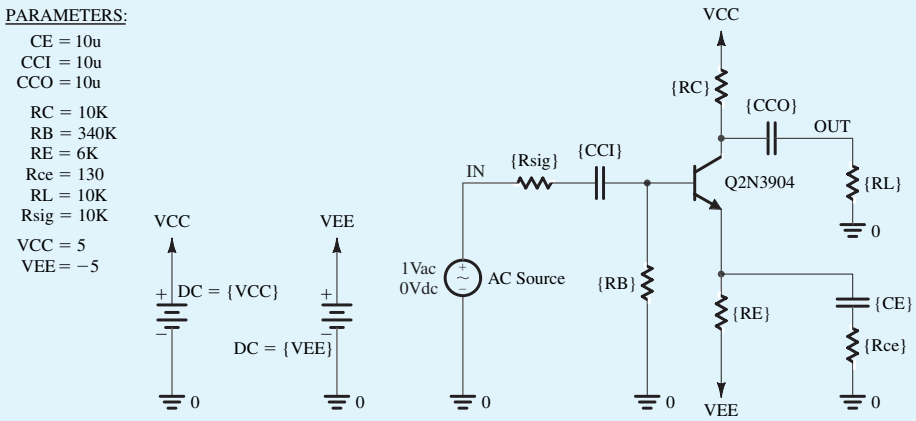


Figure B.21 Schematic capture of the CE amplifier in Example S.7.2.

achieve a high signal swing at the amplifier output. For $V_{CE} = 2$ V, the result is that $V_E = -2$ V requires bias resistors with values

$$R_C = \frac{V_{CC} - V_C}{I_C} = 10\,\text{k}\Omega$$

and

$$R_E = \frac{V_E - V_{EE}}{I_C} = 6\,\text{k}\Omega$$

Assuming $V_{BE} = 0.7$ V and using $\beta_{\text{dc}} = 123$, we can determine

$$R_B = -\frac{V_B}{I_B} = -\frac{0 - (V_{BE} + V_E)}{I_C/\beta_{\text{dc}}} = 320\,\text{k}\Omega$$

Next, the formulas of Section 7.3.4 can be used to determine the input resistance R_{in} and the midband voltage gain $|A_M|$ of the CE amplifier:

$$R_{\text{in}} = R_B \,\|\, (\beta_{\text{ac}} + 1)(r_e + R_e) \tag{B.21}$$

$$|A_M| = \left| -\frac{R_{\text{in}}}{R_{\text{sig}} + R_{\text{in}}} \times \frac{R_C \,\|\, R_L}{r_e + R_e} \right| \tag{B.22}$$

For simplicity, we will assume $\beta_{\text{ac}} \simeq \beta_{\text{dc}} = 123$, resulting in

$$r_e = \left(\frac{\beta_{\text{ac}}}{\beta_{\text{ac}} + 1}\right)\left(\frac{V_T}{I_C}\right) = 49.6\,\Omega$$

Thus, with no small-signal emitter degeneration (i.e., $R_{ce} = 0$), $R_{\text{in}} = 6.1\,\text{k}\Omega$ and $|A_M| = 38.2$ V/V. Using Eq. (B.22) and assuming R_B is large enough to have a negligible effect on R_{in}, it can be shown that the

Example S.7.2 *continued*

emitter-degeneration resistor R_e decreases the voltage gain $|A_M|$ by a factor of

$$\frac{1+\dfrac{R_e}{r_e}+\dfrac{R_{\text{sig}}}{r_\pi}}{1+\dfrac{R_{\text{sig}}}{r_\pi}}$$

Therefore, to limit the reduction in voltage gain to a factor of 2, we will select

$$R_e = r_e + \frac{R_{\text{sig}}}{\beta_{\text{ac}}+1} \tag{B.23}$$

Thus, $R_{ce} \simeq R_e = 130\,\Omega$. Substituting this value in Eqs. (B.21) and (B.22) shows that R_{in} increases from 6.1 kΩ to 20.9 kΩ while $|A_M|$ drops from 38.2 V/V to 18.8 V/V.

We will now use SPICE to verify our design and investigate the performance of the CE amplifier. We begin by performing a bias-point simulation to verify that the BJT is properly biased in the active region and that the dc voltages and currents are within the desired specifications. Based on this simulation, we have increased the value of R_B to 340 kΩ in order to limit I_C to about 0.5 mA while using a standard 1% resistor value. Next, to measure the midband gain A_M and the 3-dB frequencies[9] f_L and f_H, we apply a 1-V ac voltage at the input, perform an ac-analysis simulation, and plot the output-voltage magnitude (in dB) versus frequency as shown in Fig. B.22. This corresponds to the magnitude response of the CE amplifier because we chose a 1-V input signal.[10] Accordingly, with no emitter degeneration, the midband gain is $|A_M| = 38.5$ V/V = 31.7 dB and the 3-dB bandwidth is $BW = f_H - f_L = 145.7$ kHz. Using an R_{ce} of 130 Ω results in a drop in the midband gain $|A_M|$ by a factor of 2 (i.e., 6 dB). Interestingly, however, BW has now increased by approximately the same factor as the drop in $|A_M|$. As we learned in Chapter 11 in our study of negative feedback, the emitter-degeneration resistor R_{ce} provides negative feedback, which allows us to trade off gain for other desirable properties, such as a larger input resistance and a wider bandwidth.

To conclude this example, we will demonstrate the improved bias-point (or dc operating-point) stability achieved when an emitter resistor R_E is used. Specifically, we will increase/decrease the value of the parameter BF (i.e., the ideal maximum forward current gain) in the SPICE model for part Q2N3904 by a factor of 2 and perform a bias-point simulation. The corresponding change in BJT parameters (β_{dc} and β_{ac}) and bias-point (including I_C and CE) are presented in Table B.7 for the case of $R_E = 6$ kΩ. Note that β_{ac} is not equal to β_{dc} as we assumed, but is slightly larger. For the case without emitter degeneration, we will use $R_E = 0$ in the schematic of Fig. B.21. Furthermore, to maintain the same I_C and V_C in both cases at the values obtained for nominal BF, we use $R_B = 1.12$ MΩ to limit I_C to approximately 0.5 mA. The corresponding variations in the BJT bias point are also shown in Table B.7. Accordingly, we see that emitter degeneration makes the bias point of the CE amplifier much less sensitive to changes in β. However, unless a large bypass capacitor C_E is used, this reduced bias sensitivity comes at the expense of a reduction in the midband gain (as we observed in this example when we simulated the frequency response of the CE amplifier with an $R_e = 130\,\Omega$).

[9] No detailed knowledge of frequency-response calculations is required for this example; all that is needed is Section 7.4.2. Nevertheless, after the study of the frequency response of the CE amplifier in Sections 10.2 and 10.8, the reader will benefit by returning to this example to experiment further with the circuit using SPICE.

[10] The reader should not be alarmed about the use of such a large signal amplitude. Recall that in a small-signal (ac) simulation, SPICE first finds the small-signal equivalent circuit at the dc bias point and then analyzes this linear circuit. Such ac analysis can, of course, be done with any ac signal amplitude. However, a 1-V ac input is convenient to use because the resulting ac output corresponds to the voltage gain of the circuit.

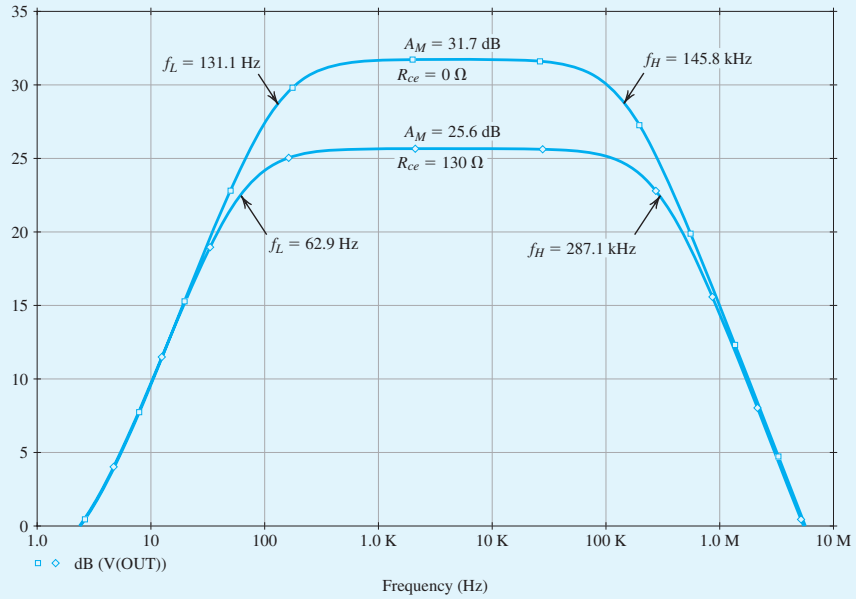


Figure B.22 Frequency response of the CE amplifier in Example S.7.2 with $R_{ce} = 0$ and $R_{ce} = 130\,\Omega$.

Table B.7 Variations in the Bias Point of the CE Amplifier with the SPICE Model-Parameter BF of BJT

	$R_E = 6$ k				$R_E = 0$			
BF (in SPICE)	β_{ac}	β_{dc}	I_C (mA)	V_C (V)	β_{ac}	β_{dc}	I_C (mA)	V_C (V)
208	106	94.9	0.452	0.484	109	96.9	0.377	1.227
416.4 (nominal value)	143	123	0.494	0.062	148	127	0.494	0.060
832	173	144	0.518	−0.183	181	151	0.588	−0.878

Example S.7.3

Design of a CMOS CS Amplifier

In this example, we will use SPICE to characterize a CS amplifier whose schematic capture is shown in Fig. B.23. We will assume a 0.18-μm CMOS technology for the MOSFET and use typical SPICE level-1 model parameters for this technology, as provided in Table B.3. We will also assume a signal-source resistance $R_{sig} = 10$ kΩ, a load resistance $R_L = 50$ kΩ, and bypass and coupling capacitors of 10 μF.

Example S.7.3 *continued*

The targeted specifications for this CS amplifier are a voltage gain $|A_v| = 10$ V/V and a maximum power consumption $P = 0.45$ mW. As should always be the case with computer simulation, we will begin with an approximate hand-analysis design. We will then use SPICE to fine-tune our design and to investigate the performance of the final design.

The amplifier specifications are summarized in Table B.8.

Hand Design

With a 1.8-V power supply, the drain current of the MOSFET must be limited to $I_D = P/V_{DD} = 0.45\,\text{mW}/1.8\,\text{V} = 0.25\,\text{mA}$ to meet the power consumption specification. Choosing $V_{OV} = 0.15$ V and $V_{DS} = V_{DD}/3 = 0.6$ V (to achieve a large signal swing at the output), the MOSFET can now be sized as

DEVICE PARAMETERS	
NAME	Q1:NMOS
W	15.48u
L	0.2u
KP	291u
LD	0.01u
VID	0.45
LAMBDA	0.08
GAMMA	0.3

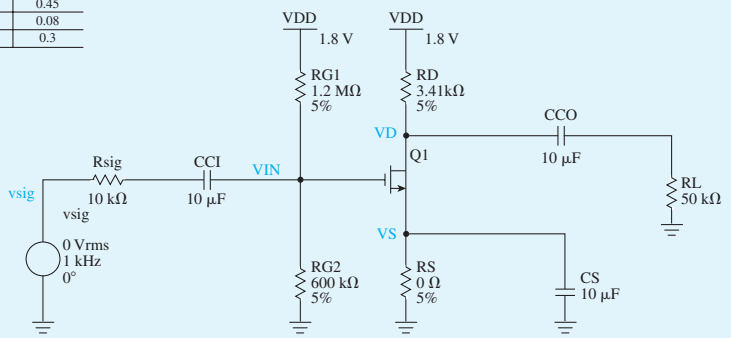


Figure B.23 Capture schematic of the CS amplifier.

Table B.8 CS Amplifier Specifications

Parameters	Value
Power	0.45 mW
R_{sig}	10 kΩ
R_L	50 kΩ
$\|A_v\|$	10 V/V
V_{DD}	1.8 V

$$\frac{W}{L_{\text{eff}}} = \frac{I_D}{\frac{1}{2}k_n' V_{OV}^2(1+\lambda V_{DS})} = \frac{250\times 10^{-6}}{\frac{1}{2}\times 246.2\times 10^{-2}\times 0.15^6\times(1+0.08\times 0.6)} \simeq 86$$

where $k_n' = \mu_n C_{\text{ox}} = 246.2\,\mu\text{A/V}^2$. Here, L_{eff} rather than L is used to more accurately compute I_D.

The effect of using W_{eff} instead of W is much less important, because typically $W \gg W_{ov}$. Thus, choosing $L = 0.200\,\mu\text{m}$ results in $L_{\text{eff}} = L - 2L_{ov} = 0.180\,\mu\text{m}$, and $W = 86 \times L_{\text{eff}} = 15.48\,\mu\text{m}$.

Note that we chose L slightly larger than L_{min}. This is a common practice in the design of analog ICs to minimize the effects of fabrication nonidealities on the actual value of L. As we have seen, this is particularly important when the circuit performance depends on the matching between the dimensions of two or more MOSFETs (e.g., in the current-mirror circuits studied in Chapter 8).

Next, R_D is calculated based on the desired voltage gain:

$$|A_v| = g_m(R_D \| R_L \| r_o) = 10\text{V/V} \Rightarrow R_D \simeq 3.41\ \text{k}\Omega$$

where

$$g_m = \frac{2I_D}{V_{OV}} = \frac{2 \times 0.25 \times 10^{-3}}{0.15} = 3.33\,\text{mA/V}$$

and

$$r_o = \frac{V_A}{I_D} = \frac{12.5}{0.25 \times 10^{-3}} = 50\,\text{k}\Omega$$

Hence, the dc bias voltage is $V_D = V_{DD} - I_D R_D = 0.9457\,\text{V}$.

To stabilize the bias point of the CS amplifier, we include a resistor in the source lead. In other words, to bias the MOSFET at $V_{DS} = V_{DD}/3$, we need an

$$R_s = \frac{V_S}{I_D} = \frac{(V_D - V_{DD}/3)}{I_D} = \frac{0.3475}{0.25 \times 10^{-3}} = 1.39\ \text{k}\Omega$$

However, as a result of including such a resistor, the gain drops by a factor of $(1 + g_m R_S)$. Therefore, we include a capacitor, C_S, to eliminate the effect of R_S on ac operation of the amplifier and gain.

Finally, choosing the current in the biasing branch to be $1\,\mu\text{A}$ gives $R_{G1} + R_{G2} = V_{DD}/1\,\mu\text{A} = 1.8\,\Omega$. Also, we know that

$$V_{GS} = V_{OV} + V_t = 0.15 + 0.45 = 0.6\,\text{V} \Rightarrow V_G = V_S + 0.6 = 0.3475 + 0.6 = 0.9475\,\text{V}$$

Hence,

$$\frac{R_{G2}}{R_{G1} + R_{G2}} = \frac{V_G}{V_{DD}} = \frac{0.9475}{1.8} \Rightarrow R_{G1} = 0.8525\,\text{M}\Omega, R_{G2} = 0.9475\ \text{M}\Omega$$

Using large values for these gate resistors ensures that both their power consumption and the loading effect on the input signal source are negligible.

Simulation

Amplifier Biasing We will now use SPICE to verify our design and investigate the performance of the CS amplifier. We begin by performing a bias-point simulation to verify that the MOSFET is properly biased in the saturation region and that the dc voltages and currents match the expected values. The results are shown in Fig. B.24.

Example S.7.3 *continued*

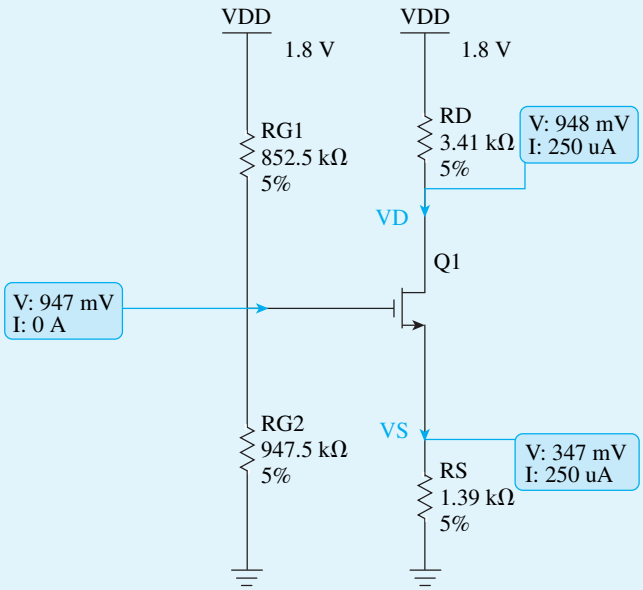


Figure B.24 DC bias-point analysis of the CS amplifier.

Amplifier Gain We can also verify if our design provides the desired gain. This can be done by performing transient response analysis. As can be seen from Fig. B.25, $|G_v| \simeq |A_v| \simeq 11$ V/V. Note the values of overall voltage gain G_v and A_v are close since $R_{in} = (R_{G1} \| R_{G2}) \gg R_{\text{sig}}$. In the case where the capacitor C_S is not included ($C_S = 0$), the gain drops by a factor of 5.63 (approximately equal to $1 + g_m R_S$) to 1.95. This is as expected from our study of the CS amplifier with a source-degeneration resistance.

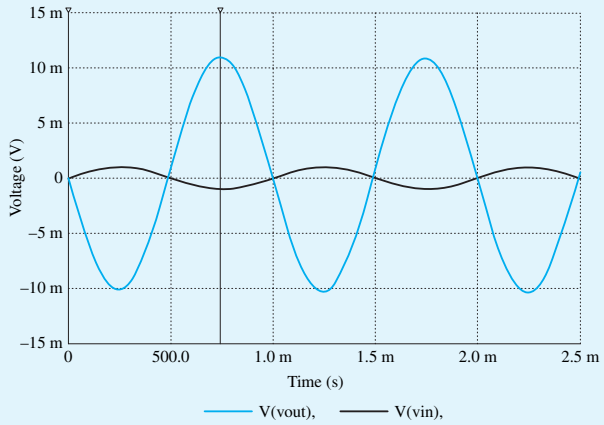


Figure B.25 A_v and G_v of the CS amplifier: transient analysis.

Investigating Amplifier Bias Stability We can also demonstrate the improved bias stability achieved when a source resistor R_S is used. Specifically, we change (in the MOSFET level-1 model) the value of the zero-bias threshold voltage parameter VTO by ± 0.1 V and perform bias-point simulation in SPICE. Table B.9 shows the corresponding variations in I_D and V_D for the case in which $R_S = 1.39$ kΩ. For the case without source degeneration, we use an $R_S = 0$ in the given schematic. Furthermore, to obtain the same I_D and V_D in both cases (for the nominal threshold voltage $V_{t0} = 0.45$ V), we use $R_{G1} = 1.2$ MΩ and $R_{G2} = 0.6$ MΩ.

Table B.9 Variations in VTO

VTO (V)	I_D(μA)	I_D % Change	V_D (V)	V_D % Change
		With $R_S = 1.39$ kΩ		
0.45	250	0	0.948	0
0.35	309	23.60%	0.748	−21.10%
0.55	192	−37.86%	1.14	20.25%
		Without R_S		
0.45	255.96	0	0.9292	0
0.35	492	96.80%	0.122	−87.13%
0.55	30.1	−90.26%	1.7	127.27%

Table B.10 Variations Due to Resistor Tolerances

	R_{G1} (MΩ)	R_{G2} (MΩ)	I_D (μA)	I_D % Change	V_D (V)	V_1 % Change
			With $R_S = 1.39$ kΩs			
Nominal	0.8525	0.9475	250	0	947.67	0
I_D low V_D high	0.895	0.9	223.86	−10.44%	1.037	9.39%
I_D high V_D low	0.81	0.995	276.1	10.46%	0.858	−9.41%
			Without R_S			
	R_{G1} (MΩ)	R_{G2} (MΩ)	I_D (μA)	I_D % Change	V_D (V)	V_D % Change
Nominal	1.2	0.6	255.96	0	0.9292	0
I_D low V_D high	1.26	0.57	143.28	−44.02%	1.311	41.44%
I_D high V_D low	1.14	0.63	398.62	55.74%	0.447	−52.47%

Also, Table B.10 shows the worst-case deviation of I_D and V_D values, when imposing 5% tolerance on the resistors that determine the gate voltage.

Example S.7.3 *continued*

Accordingly, we see that the source-degeneration resistor makes the bias point of the CS amplifier less sensitive to changes in the threshold voltage and the values of gate resistors. However, unless a large bypass capacitor C_S is used, this reduced sensitivity comes at the expense of a reduction in gain.

Largest Allowable Input Signal Swing Next, we wish to analyze this amplifier circuit to determine the largest allowable v_{sig} for which the transistor remains in saturation:

$$v_{DS} \geq v_{GS} - v_t$$

By enforcing this condition, with equality, at the point v_{GS} is maximum and v_{DS} is correspondingly minimum, we write:

$$v_{DS,\text{min}} \geq v_{GS,\text{max}} - v_{t0}$$

$$v_{DS} - |G_v| v_{\text{sig}} = V_{GS} + v_{\text{sig}} - v_{t0}$$

$$v_{\text{sig}} = \frac{V_{DS} - V_{GS} + V_{t0}}{(1 + |G_v|)} = \frac{0.9475 - 0.6 + 0.45}{11} = 72.5\,\text{mV}$$

If we increase the source signal's amplitude beyond approximately 73 mV, we can observe the distortion in the output signal, indicating that the MOSFET has entered the triode region.

Amplifier Linearity Finally, we can investigate the linearity of the designed amplifier. In this case, we use a triangular waveform and increase the amplitude of the signal until the output waveform begins to show nonlinear distortion (i.e., the rising and falling edges are no longer straight lines). Based on hand analysis, linearity holds as long as $v_{\text{in}} \ll 2V_{ov}$. According to the simulation results, linearity holds until v_{in} reaches the value of approximately 30 mV, which is one-tenth of the value of $2V_{ov}$.

Example S.8.1

The CS Amplifier with Active load

In this example, we will use SPICE to compute the dc transfer characteristic of the CS amplifier whose capture schematic is shown in Fig. B.27. We will assume a 5-μm CMOS technology for the MOSFETs and use parts NMOS5P0 and PMOS5P0 whose SPICE level-1 parameters are listed in Table B.3. To specify the dimensions of the MOSFETs in SPICE, we will use the multiplicative factor m together with the channel length L and the channel width W. The MOSFET parameter m, whose default value is 1, is used in SPICE to specify the number of MOSFETs connected in parallel. As depicted in Fig. B.28, a wide transistor with channel length L and channel width $m \times W$ can be implemented using m narrower transistors in parallel, each having a channel length L and a channel width W. Thus, neglecting the channel-length modulation effect, the drain current of a MOSFET operating in the saturation region can be expressed as

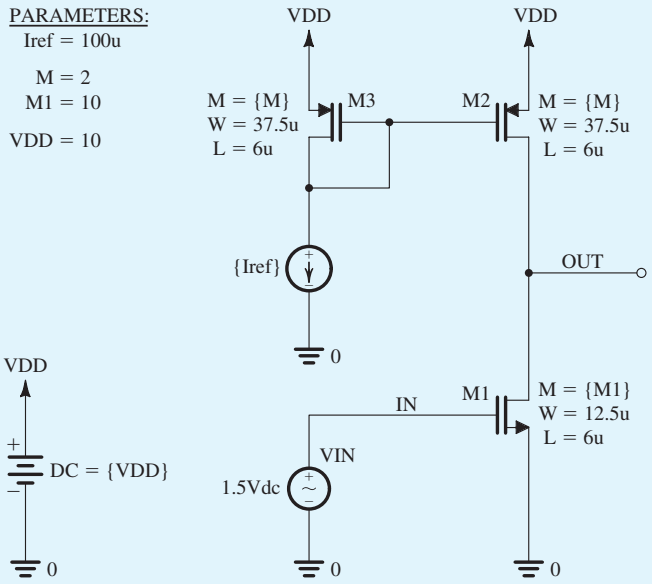


Figure B.26 Schematic capture of the CS amplifier in Example S.8.1.

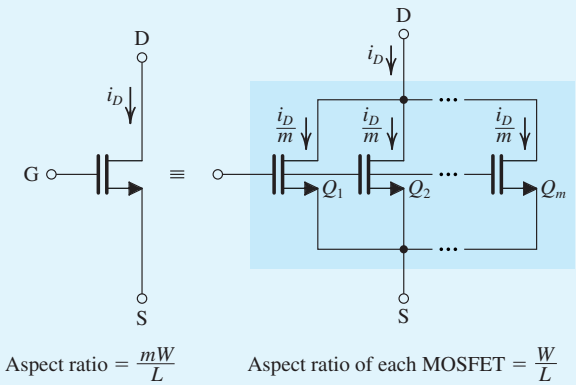


Figure B.27 Transistor equivalency.

$$I_D = \frac{1}{2}\mu C_{ox} m \frac{W}{L_{\text{eff}}} V_{OV}^2 \tag{B.24}$$

where L_{eff} rather than L is used to more accurately estimate the drain current.

The CS amplifier in Fig. B.27 is designed for a bias current of 100 μA assuming a reference current $I_{\text{ref}} = 100\,\mu\text{A}$ and $V_{DD} = 10$ V. The current mirror transistors M_2 and M_3 are sized for $V_{OV2} = V_{OV3} = 1$ V,

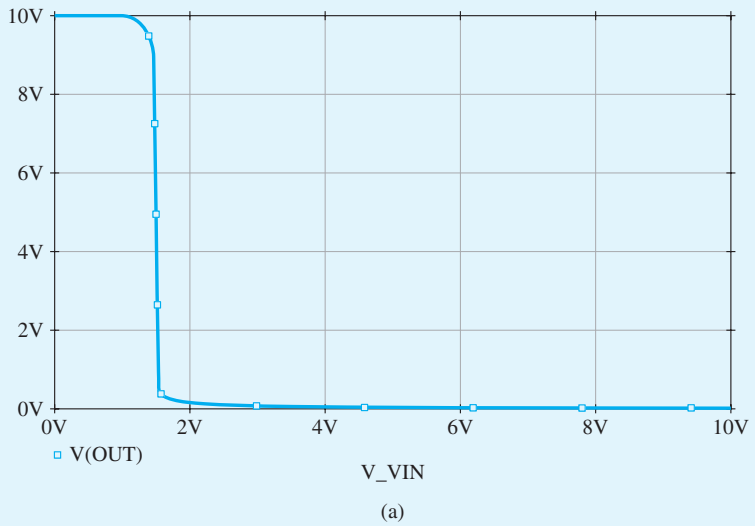
Example S.8.1 *continued*

while the input transistor M_1 is sized for $V_{OV1} = 0.5$ V. Note that a smaller overdrive voltage is selected for M_1 to achieve a larger voltage gain G_v for the CS amplifier, since

$$G_v = -g_{m1}R'_L = -g_{m1}(r_{o1} \parallel r_{o2}) = -\frac{2}{V_{OV1}}\left(\frac{V_{An}V_{Ap}}{V_{An}+V_{Ap}}\right) \tag{B.25}$$

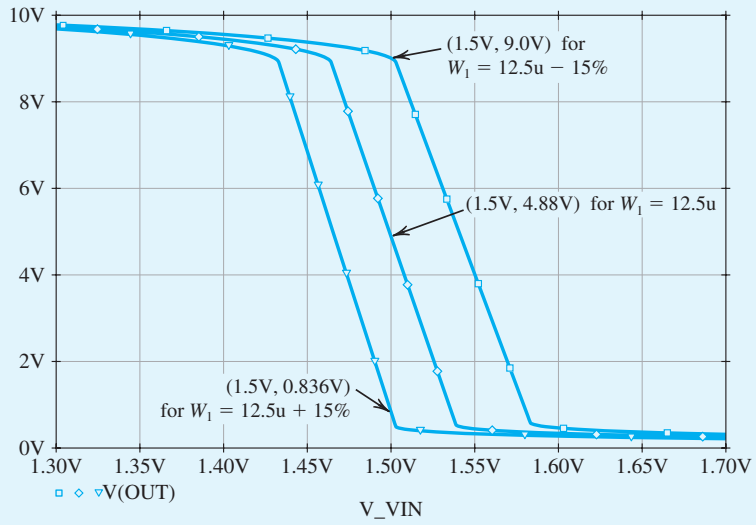
where V_{An} and V_{Ap} are the magnitudes of the Early voltages of, respectively, the NMOS and PMOS transistors. Unit-size transistors are used with $W/L = 12.5\ \mu m/6\ \mu m$ for the NMOS devices and $W/L = 37.5\ \mu m/6\ \mu m$ for the PMOS devices. Thus, using Eq. (B.24) together with the 5-μm CMOS process parameters in Table B.3, we find $m_1 = 10$ and $m_2 = m_3 = 2$ (rounded to the nearest integer). Furthermore, Eq. (B.25) gives $G_v = -100$ V/V.

To compute the dc transfer characteristic of the CS amplifier, we perform a dc analysis in SPICE with V_{IN} swept over the range 0 to V_{DD} and plot the corresponding output voltage V_{OUT}. Figure B.28(a) shows the resulting transfer characteristic. The slope of this characteristic (i.e., dV_{OUT}/dV_{IN}) corresponds to the gain of the amplifier. The high-gain segment is clearly visible for V_{IN} around 1.5 V. This corresponds to an overdrive voltage for M_1 of $V_{OV1} = V_{IN} - V_{tn} = 0.5$ V, as desired. To examine the high-gain region more closely, we repeat the dc sweep for V_{IN} between 1.3 V and 1.7 V. The resulting transfer characteristic is plotted in Fig. B.28 (b, middle curve). Using the graphical interface of SPICE, we find that the linear region of this dc transfer characteristic is bounded approximately by $V_{IN} = 1.465$ V and $V_{IN} = 1.539$ V. The corresponding values of V_{OUT} are 8.838 V and 0.573 V. These results are close to the expected values. Specifically, transistors M_1 and M_2 will remain in the saturation region and, hence, the amplifier will operate in its linear region if $V_{OV1} \le V_{OUT} \le V_{DD} - V_{OV2}$ or $0.5\ \text{V} \le V_{OUT} \le 9$ V. From the results above, the voltage gain G_v (i.e., the slope of the linear segment of the dc transfer characteristic) is approximately −112 V/V, which is reasonably close to the value obtained by hand analysis.



(a)

Figure B.28 **(a)** Voltage transfer characteristic of the CS amplifier in Example S.8.1. **(b)** Expanded view of the transfer characteristic in the high-gain region. Also shown are the transfer characteristics where process variations cause the width of transistor M_1 to change by +15% and –15% from its nominal value of $W_1 = 12.5\ \mu m$.



(b)

Figure B.28 *continued*

Note from the dc transfer characteristic in Fig. B.25(b) that for an input dc bias of $V_{IN} = 1.5$ V, the output dc bias is $V_{OUT} = 4.88$ V. This choice of V_{IN} maximizes the available signal swing at the output by setting V_{OUT} at the middle of the linear segment of the dc transfer characteristic. However, because of the high resistance at the output node (or, equivalently, because of the high voltage gain), this value of V_{OUT} is highly sensitive to the effect of process and temperature variations on the characteristics of the transistors. To illustrate this point, consider what happens when the width of M_1 (i.e., W_1, which is normally 12.5 μm) changes by ±15%. The corresponding dc transfer characteristics are shown in Fig. B.25(b). Accordingly, when $V_{IN} = 1.5$ V, V_{OUT} will drop to 0.84 V if W_1 increases by 15% and will rise to 9.0 V if W_1 decreases by 15%. In practical circuit implementations, this problem is circumvented by using negative feedback to accurately set the dc bias voltage at the output of the amplifier and, hence, to reduce the sensitivity of the circuit to process variations. We studied negative feedback in Chapter 11.

Example S.9.1

A Multistage Differential BJT Amplifier

The schematic capture of the multistage op-amp circuit analyzed in Example 9.7 is shown in Fig. B.29. Observe the manner in which the differential signal input V_d and the common-mode input voltage V_{CM} are applied. Such an input bias configuration for an op-amp circuit was presented and used

Example S.9.1 *continued*

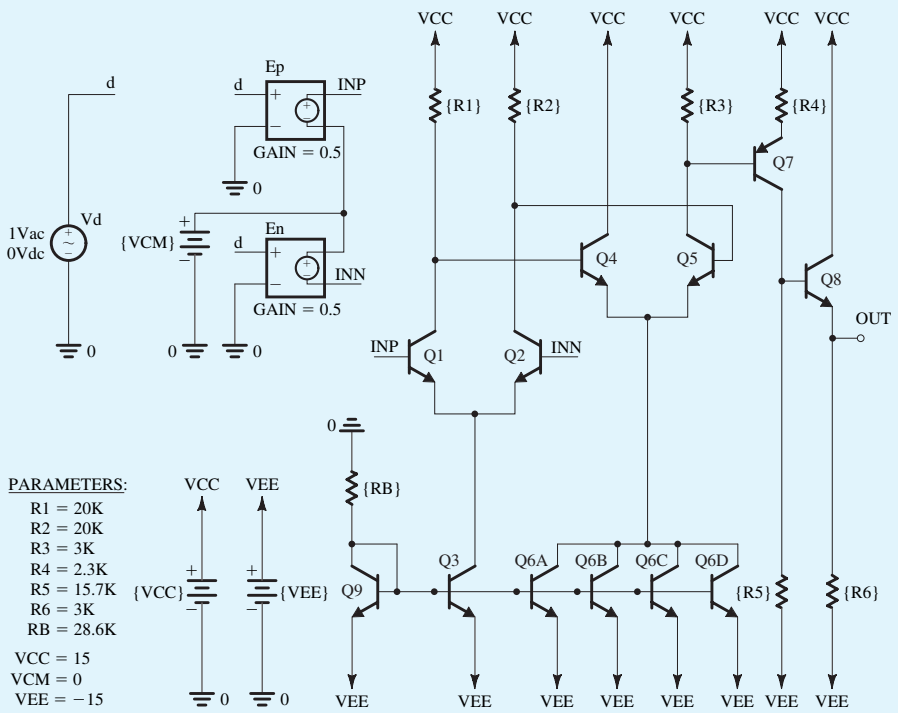


Figure B.29 Schematic capture of the op-amp circuit in Example 9.7.

in Example S.2.2. In the following simulations, we will use parts Q2N3904 and Q2N3906 (from Fairchild Semiconductor) for the *npn* and *pnp* BJTs, respectively. The model parameters of these discrete BJTs are listed in Table B.11

In SPICE, the common-mode input voltage V_{CM} of the op-amp circuit is set to 0 V (i.e., to the average of the dc power-supply voltages V_{CC} and V_{EE}) to maximize the available input signal swing. A bias-point simulation is performed to determine the dc operating point. Table B.12 summarizes the value of the dc collector currents as computed by SPICE and as calculated by the hand analysis in Example 9.7. Recall that our hand analysis assumed both β and the Early voltage V_A of the BJTs to be infinite. However, our SPICE simulations in Example S.6.1 (where we investigated the dependence of β on the collector current I_C) indicate that the Q2N3904 has $\beta \simeq 125$ at $I_C = 0.25\,\text{mA}$. Furthermore, its forward Early voltage (SPICE parameter VAF) is 74 V, as given in Table B.11. Nevertheless, we observe from Table B.12 that the largest error in the calculation of the dc bias currents is on the order of 20%. Accordingly, we can conclude that a quick hand analysis using gross approximations can still yield reasonable results for a preliminary estimate and, of course, hand analysis yields much insight into the circuit operation. In addition to the dc bias currents listed in Table B.12, the bias-point simulation in SPICE shows that the output dc offset (i.e., V_{OUT} when $V_d = 0$) is 3.62 V and that the input bias current I_{B1} is 2.88 μA.

To compute the **large-signal differential transfer characteristic** of the op-amp circuit, we perform a dc-analysis simulation in SPICE with the differential voltage input V_d swept over the range $-V_{EE}$ to

Table B.11 Spice Model Parameters of the Q2N3904 and Q2N3906 Discrete BJTs

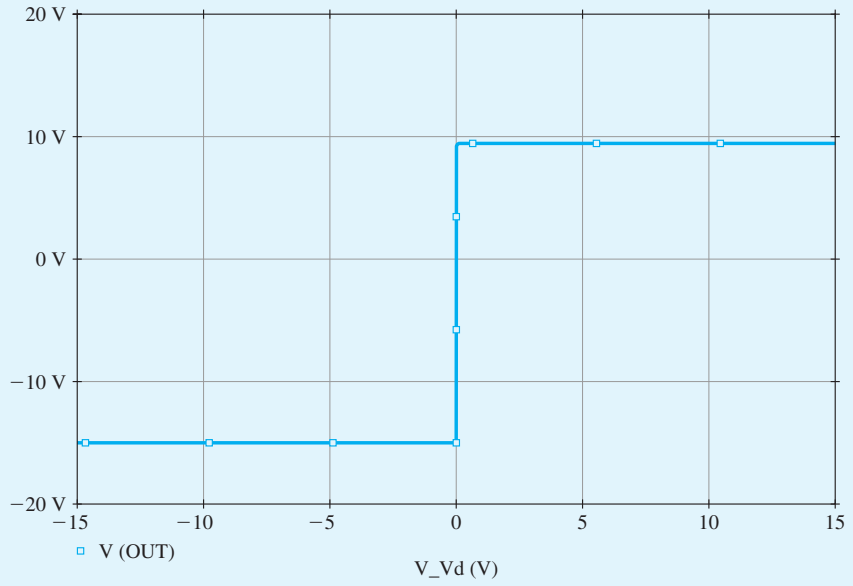
Q2N3904 Discrete BJT						
IS = 6.734f	XTI = 3	EG = 1.11	VAF = 74.03	BF = 416.4	NE = 1.259	ISE = 6.734f
IKF = 66.78m	XTB = 1.5	BR = .7371	NC = 2	ISC = 0	IKR = 0	RC = 1
CJC = 3.638p	MJC = .3085	VJC = .75	FC = .5	CJE = 4.493p	MJE = .2593	VJE = .75
TR = 239.5n	TF = 301.2p	ITF = .4	VTF = 4	XTF = 2	RB = 10	
Q2N3906 Discrete BJT						
IS = 1.41f	XTI = 3	EG = 1.11	VAF = 18.7	BF = 180.7	NE = 1.5	ISE = 0
IKF = 80m	XTB = 1.5	BR = 4.977	NC = 2	ISC = 0	IKR = 0	RC = 2.5
CJC = 9.728p	MJC = .5776	VJC = .75	FC = .5	CJE = 8.063p	MJE = .3677	VJE = .75
TR = 33.42n	TF = 179.3p	ITF = .4	VTF = 4	XTF = 6	RB = 10	

Table B.12 DC Collector Currents of the Op-Amp Circuit in Fig. B.29 as Computed by Hand Analysis (Example 9.7) and by SPICE

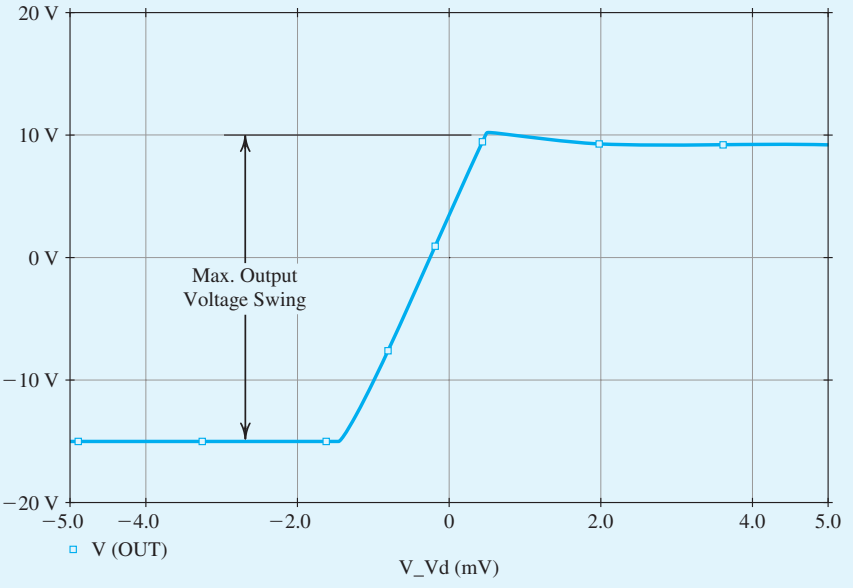
	Collector Currents (mA)		
Transistor	Hand Analysis (Example 9.7)	SPICE	Error (%)
Q_1	0.25	0.281	–11.0
Q_2	0.25	0.281	–11.0
Q_3	0.5	0.567	–11.8
Q_4	1.0	1.27	–21.3
Q_5	1.0	1.21	–17.4
Q_6	2.0	2.50	–20.0
Q_7	1.0	1.27	–21.3
Q_8	5.0	6.17	–18.9
Q_9	0.5	0.48	+4.2

$+V_{CC}$, and we plot the corresponding output voltage V_{OUT}. Figure B.30(a) shows the resulting dc transfer characteristic. The slope of this characteristic (i.e., dV_{OUT}/dV_d) corresponds to the differential gain of the amplifier. Note that, as expected, the high-gain region is in the vicinity of $V_d = 0\,\text{V}$. However, the resolution of the input-voltage axis is too coarse to yield much information about the details of the high-gain region. Therefore, to examine this region more closely, the dc analysis is repeated with V_d swept over the range $-5\,\text{mV}$ to $+5\,\text{mV}$ at increments of $10\,\mu\text{V}$. The resulting differential dc transfer characteristic is plotted in Fig. B.30(b). We observe that the linear region of the large-signal differential characteristic is bounded approximately by $V_d = -1.5\,\text{mV}$ and $V_d = +0.5\,\text{mV}$. Over this region, the output level changes from $V_{\text{OUT}} = -15\,\text{V}$ to about $V_{\text{OUT}} = +10\,\text{V}$ in a linear fashion. Thus, the output voltage swing for this amplifier is between $-15\,\text{V}$ and $+10\,\text{V}$, a rather asymmetrical range. A rough estimate for the differential gain of this amplifier can be obtained from the boundaries of the linear region as $A_d = [10 - (-15)]\,\text{V}/[0.5 - (-1.5)]\text{mV} = 12.5 \times 10^3\,\text{V/V}$. We also observe from Fig B.30(b) that $V_d \simeq -260\ \mu\text{V}$ when $V_{\text{OUT}} = 0$. Therefore, the amplifier has an input offset voltage V_{OS} of $+260\ \mu\text{V}$ (by convention, the negative value of the x-axis intercept of the large-signal differential transfer characteristic). This corresponds to an output offset voltage of $A_d V_{OS} \simeq \left(12.5 \times 10^3\right)(260\,\mu\text{V}) = 3.25\,\text{V}$,

Example S.9.1 *continued*



(a)



(b)

Figure B.30 **(a)** The large-signal differential transfer characteristic of the op-amp circuit in Fig. B.29. The common-mode input voltage V_{CM} is set to 0 V. **(b)** An expanded view of the transfer characteristic in the high-gain region.

which is close to the value found through the bias-point simulation. It should be emphasized that this offset voltage is inherent in the design and is not the result of component or device mismatches. Thus, it is usually referred to as a **systematic offset**.

Next, to compute the frequency response of the op-amp circuit[12] and to measure its differential gain A_d and its 3-dB frequency f_H in SPICE, we set the differential input voltage V_d to be a 1-V ac signal (with 0-V dc level), perform an ac-analysis simulation, and plot the output voltage magnitude $|V_{\text{OUT}}|$ versus frequency. Figure B.31(a) shows the resulting frequency response. Accordingly, $A_d = 13.96 \times 10^3$ V/V or 82.8 dB, and $f_H = 256.9$ kHz. Thus, this value of A_d is close to the value estimated using the large-signal differential transfer characteristic.

An approximate value of f_H can also be obtained using the expressions derived for the equivalent differential half-circuit in Chapter 10. Specifically,

$$f_H \simeq \frac{1}{2\pi R_{\text{eq}} C_{\text{eq}}} \tag{B.26}$$

where

$$C_{\text{eq}} = C_{\mu 2} + C_{\pi 5} + C_{\mu 5}\left[1 + g_{m5}\left(R_3 \| r_{o5} \| \left(r_{\pi 7} + (\beta + 1)R_4\right)\right)\right]$$

and

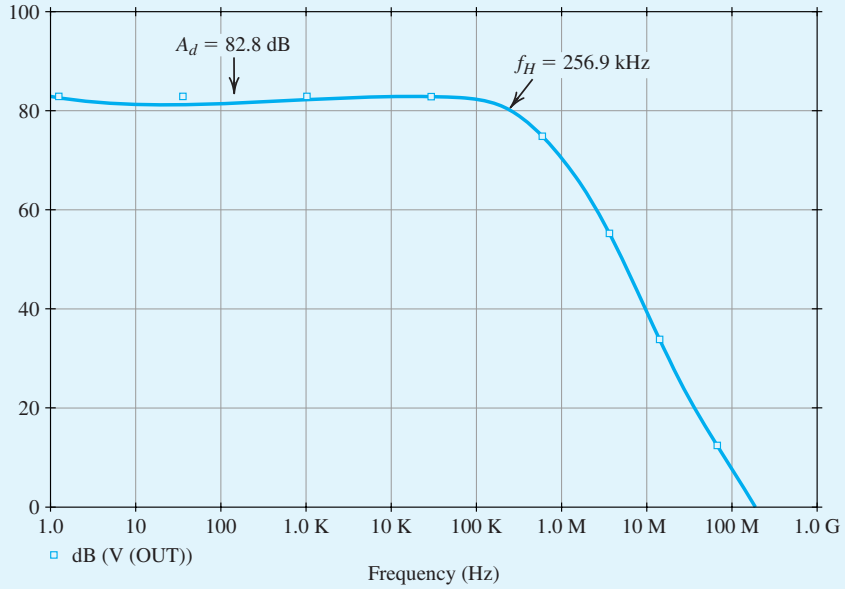
$$R_{\text{eq}} = R_2 | r_{o2} \| r_{\pi 5}$$

The values of the small-signal parameters as computed by SPICE can be found in the output file of a bias-point (or an ac-analysis) simulation. Using these values results in $C_{\text{eq}} = 338$ pF, $R_{\text{eq}} = 2.91$ kΩ, and $f_H = 161.7$ kHz. However, this approximate value of f_H is much smaller than the value computed by SPICE. The reason for this disagreement is that the foregoing expression for f_H was derived using the equivalent differential half-circuit concept. However, the concept is accurate only when it is applied to a symmetrical circuit. The op-amp circuit in Fig. B.29 is not symmetrical because the second gain stage formed by the differential pair Q_4–Q_5 has a load resistor R_3 in the collector of Q_5 only. To verify that the expression for f_H in Eq. (B.26) gives a close approximation for f_H in the case of a symmetric circuit, we insert a resistor R_3' (whose size is equal to R_3) in the collector of Q_4. Note that this will have only a minor effect on the dc operating point. The op-amp circuit with Q_4 having a collector resistor R_3' is then simulated in SPICE. Figure B.31(b) shows the resulting frequency response of this symmetric op amp, where $f_H = 155.7$ kHz. Accordingly, in the case of a perfectly symmetric op-amp circuit, the value of f_H in Eq. (B.29) closely approximates the value computed by SPICE. Comparing the frequency responses of the nonsymmetric (Fig. B.31a) and the symmetric (Fig. B.31b) op-amp circuits, we note that the 3-dB frequency of the op amp drops from 256.9 kHz to 155.7 kHz when resistor R_3' is inserted in the collector of Q_4 to make the op-amp circuit symmetrical. This is because, with a resistor R_3', the collector of Q_4 is no longer at signal ground and, hence, $C_{\mu 4}$ experiences the Miller effect. Consequently, the high-frequency response of the op-amp circuit is degraded.

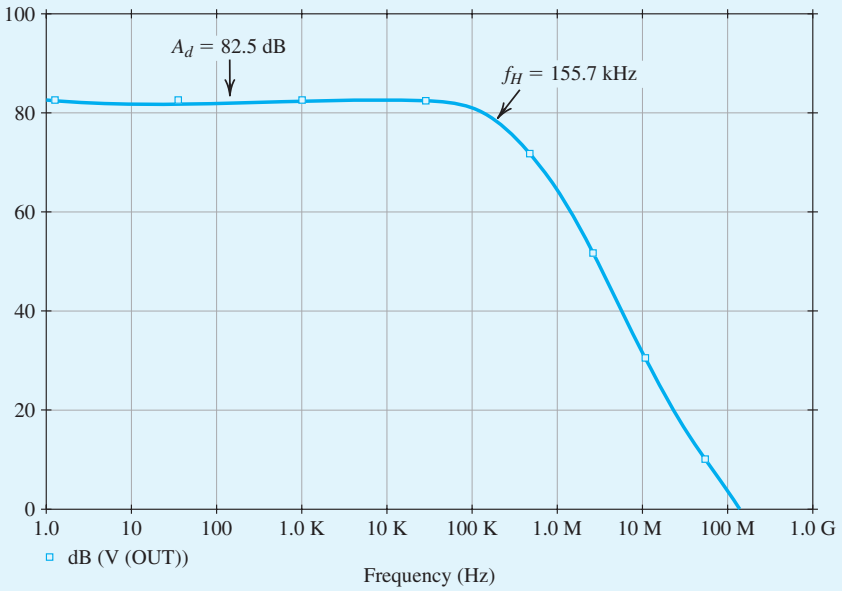
Observe that in the preceding ac-analysis simulation, owing to the systematic offset inherent in the design, the op-amp circuit is operating at an output dc voltage of 3.62 V. However, in an actual circuit implementation (with $V_{CM} = 0$), negative feedback is employed (see Chapters 2 and 11)

[12]This part of the example requires study of Chapter 10.

Example S.9.1 *continued*



(a)

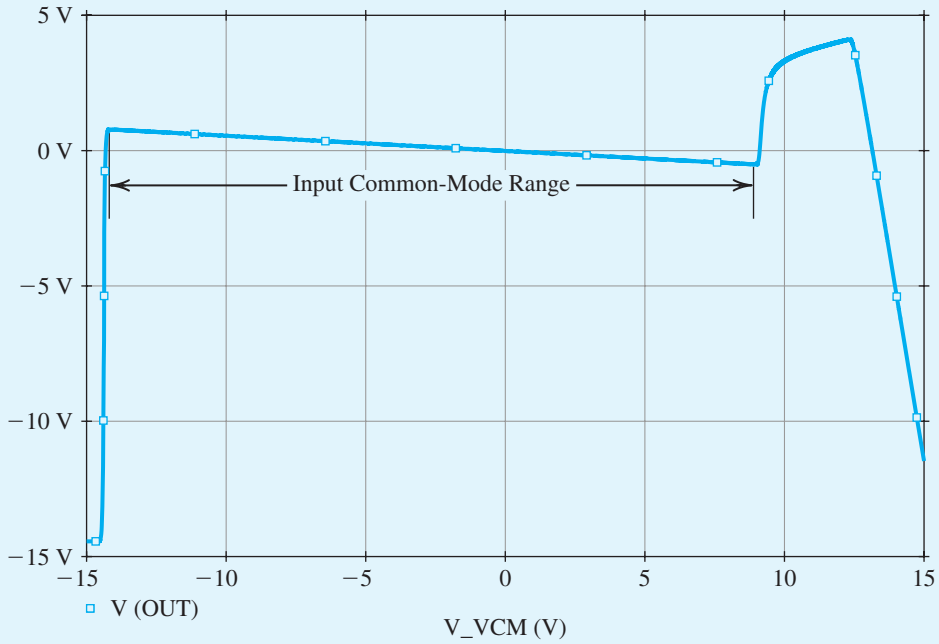


(b)

Figure B.31 Frequency response of **(a)** the op-amp circuit in Fig. B.29 and **(b)** the op-amp circuit in Fig. B.29 but with a resistor $R_3' = R_3$ inserted in the collector of Q_4 to make the op-amp circuit symmetrical.

and the output dc voltage is stabilized at zero. Thus, the small-signal performance of the op-amp circuit can be more accurately simulated by biasing the circuit so as to force operation at this level of output voltage. This can be easily done by applying a differential dc input of $-V_{OS}$. Superimposed on this dc input, we can apply an ac signal to perform an ac-analysis simulation for the purpose of, for example, computing the differential gain and the 3-dB frequency.

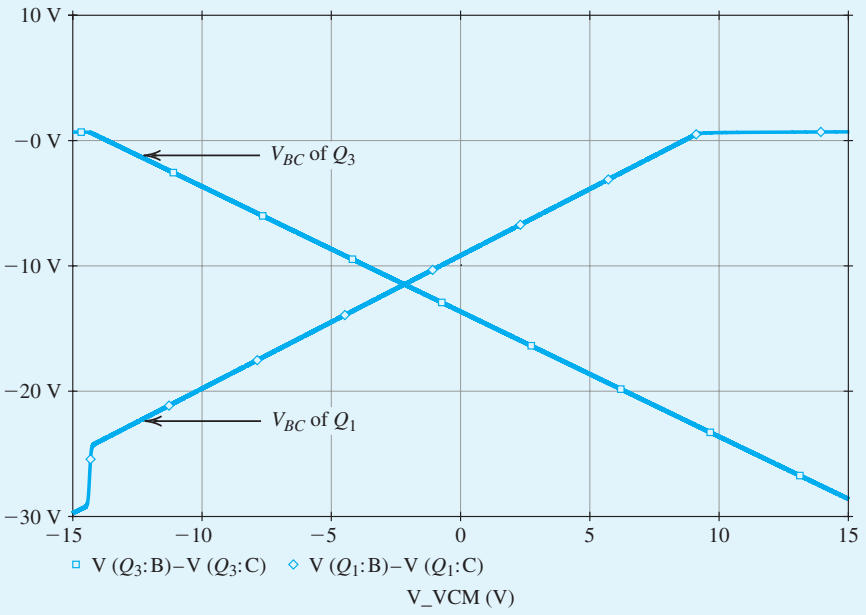
Finally, to compute the input common-mode range of the op-amp circuit in Fig. B.29, we perform a dc-analysis simulation in SPICE with the input common-mode voltage swept over the range $-V_{EE}$ to V_{CC}, while maintaining V_d constant at $-V_{OS}$ in order to cancel the output offset voltage (as discussed earlier) and, thus, prevent premature saturation of the BJTs. The corresponding output voltage V_{OUT} is plotted in Fig. B.32(a). From this common-mode dc transfer characteristic we find that the amplifier behaves linearly over the V_{CM} range –14.1 V to +8.9 V, which is therefore the **input common-mode range**. In Example 9.7, we noted that the upper limit of this range is determined by Q_1 and Q_2 saturating, whereas the lower limit is determined by Q_3 saturating. To verify this assertion, we requested SPICE to plot the values of the collector–base voltages of these BJTs versus the input common-mode voltage V_{CM}. The results are shown in Fig. B.32(b), from which we note that our assertion is indeed correct (recall that an *npn* BJT enters its saturation region when its base–collector junction becomes forward biased, i.e., $V_{BC} \geq 0$).



(a)

Figure B.32 **(a)** The large-signal common-mode transfer characteristic of the op-amp circuit in Fig. B.29. The differential input voltage V_d is set to $-V_{OS} = -260\,\mu\text{V}$ to prevent premature saturation. **(b)** The effect of the common-mode input voltage VCM on the linearity of the input stage of the op-amp circuit in Fig. B.29. The base–collector voltage of Q_1 and Q_3 is shown as a function of VCM. The input stage of the op-amp circuit leaves the active region when the base–collector junction of either Q_1 or Q_3 becomes forward biased (i.e., when VBC ≥ 0).

Example S.9.1 *continued*



(b)

Figure B.32 *continued*

Example S.9.2

The Two-Stage CMOS Op Amp

In this example, we will design the two-stage CMOS op amp whose schematic capture is shown in Fig. B.33. Once designed, the circuit's characteristics, such as the input common-mode range, the common-mode rejection ratio, the output-voltage range, and the input offset voltage will be evaluated.

The first stage is differential pair Q_1–Q_2 (which is actively loaded with the current mirror formed by Q_3 and Q_4), with bias current supplied by the current mirror formed by Q_8, and Q_5, which utilizes the reference bias current I_{REF}. The second stage consists of Q_6, which is a common-source amplifier actively loaded with the current source transistor Q_7.

For the design of this CMOS op amp, we will assume a 0.18-μm CMOS technology for the MOSFETs and use typical SPICE level-1 model parameters for this technology in Table B.3, excluding the intrinsic capacitance values. We will begin with an approximate hand-analysis design. We will then use SPICE to verify that the implemented circuit meets the specifications. The targeted specifications for this op amp

are a dc open-loop voltage gain $|A_v| = 2500$ V/V, with each of transistors Q_1, Q_2, Q_3, and Q_4 conducting a drain current of 100 μA.

To achieve the targeted specifications, a biasing current $I_{REF} = 200$ μA is used, and the transistors Q_5, Q_6, Q_7, and Q_8 will be sized so that they conduct the drain current of 200 μA. Also, the open-loop voltage gain for this design is the product of the voltage gains of the two stages. Accordingly, each stage is designed to contribute a voltage gain of –50 V/V, so as to achieve the specified open-loop voltage gain.

NMOS	
Vtn	0.5 V
|VAn|	12.5V
kn'	246.2μA/V^2
IREF	0.2mA
L	0.2 um
W	0.48 um
PMOS	
Vtp	–0.5 V
|VAp|	9V
kp'	–86.1 μA/V^2
IREF	0.1mA
L	0.2 um
W	0.64 um

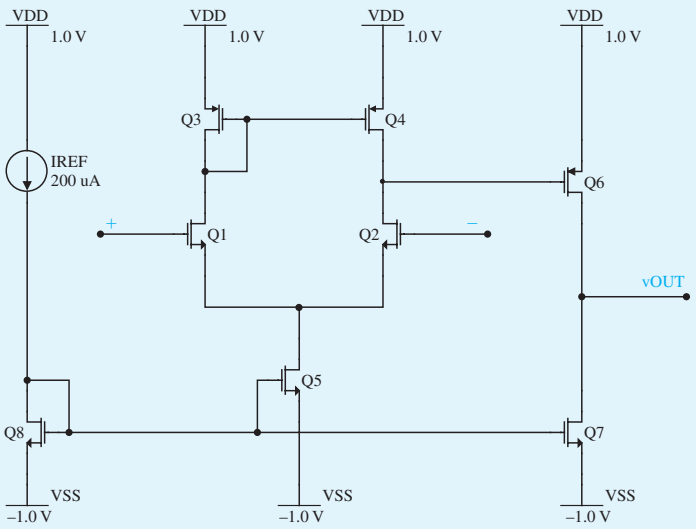


Figure B.33 Schematic capture of the two-stage CMOS op amp.

The amplifier specifications are summarized in Table B.13.

Table B.13 Two-Stage CMOS Op-Amp Specifications

Parameter	Value
$I_{(Q1,Q2,Q3,\text{and}\,Q4)}$	100 μA
$I_{(Q5,Q6,Q7,\text{and}\,Q8)}$	200 μA
$\|A_1\|$	50 V/V
$\|A_2\|$	50 V/V
V_{DD}	1 V
V_{SS}	–1 V

Hand Design For the design of this amplifier we choose $L = 0.200\,\mu$m, so we have $L_{\text{eff}} = 0.180\,\mu$m. For this channel length, and in 0.18-μm CMOS technology, the magnitudes of the Early voltages of the NMOS and PMOS transistors are $V_{An} = 12.5$ V and $|V_{Ap}| = 9$ V.

Example S.9.2 *continued*

The two-stage CMOS op amp in Fig. B.33 is equivalent to the one in Fig. 9.37, except that the first stage is an NMOS differential amplifier and the second stage is a PMOS common source. Note that the differential voltage gain of the first stage can be expressed using Eq. (9.140) as:

$$A_1 = -g_{m1}(r_{o2} \| r_{o4})$$

Hence,

$$A_1 = -\frac{2}{V_{OV1}}\left(\frac{V_{An}|V_{Ap}|}{V_{An}+|V_{Ap}|}\right)$$

resulting in

$$V_{OV1} = -\frac{2}{A_1}\left(\frac{V_{An}|V_{Ap}|}{V_{An}+|V_{Ap}|}\right) = -\frac{2}{(-50)}\left(\frac{12.5\times 9}{12.5+9}\right) \simeq 0.21\,\text{V}$$

Also, the voltage gain of the second stage is provided by Eq. (9.141) as

$$A_2 = -g_{m6}(r_{o6} \| r_{o7})$$

Therefore,

$$A_2 = -\frac{2}{V_{OV6}}\left(\frac{V_{An}|V_{Ap}|}{V_{An}+|V_{Ap}|}\right)$$

resulting in

$$V_{OV6} = -\frac{2}{A_2}\left(\frac{V_{An}|V_{Ap}|}{V_{An}+|V_{Ap}|}\right) = -\frac{2}{(-50)}\left(\frac{12.5\times 9}{12.5+9}\right) \simeq 0.21\,\text{V}$$

For this example, all transistors are sized for an overdrive voltage of 0.21 V. Furthermore, to simplify the design procedure, we ignore the channel-length modulation effect. As a result, using unit-size NMOS transistors with $W_n/L_n = 0.64\,\mu\text{m}/0.2\,\mu\text{m}$, and unit-size PMOS transistors with $W_n/L_n = 0.48\,\mu\text{m}/0.2\,\mu\text{m}$, the corresponding multiplicative factor m for each transistor can be calculated by rounding to the nearest integer value which is calculated as m:

$$m = \frac{I_D}{\frac{1}{2}k'\left(\frac{W}{L_{\text{eff}}}\right)V_{OV}^2}$$

Table B.14 summarizes the relevant information and the calculated m values for each transistor.

Table B.14 Transistor Sizes

Transistor	I_D (μA)	V_{OV} (V)	W (μm)	L_{eff} (μm)	k' (μA/V^2)	m
1	100	0.21	0.48	0.18	246.2	7
2	100	0.21	0.48	0.18	246.2	7
3	100	0.21	0.64	0.18	86.1	15
4	100	0.21	0.64	0.18	86.1	15
5	200	0.21	0.48	0.18	246.2	14
6	200	0.21	0.64	0.18	86.1	30
7	200	0.21	0.48	0.18	246.2	14
8	200	0.21	0.48	0.18	246.2	14

Simulation

Verifying A_v Based on the simulation results we read $|A_1| = 57$ V/V, $|A_2| = 58.6$ V/V, $|A_v| = 3340$ V/V, $I_{(Q1,Q2,Q3,\text{and}Q4)} = 97\,\mu$A, $I_{Q5} = 194\,\mu$A, $I_{(Q6,Q7)} = 202\,\mu$A, and $I_{Q8} = 200\,\mu$A. These values are somewhat different from the targeted specifications. The deviations can be attributed to the fact that we rounded the values of m to the nearest integer and ignored the effect of channel-length modulation, that is, the term $(1+\lambda V_{DS})$, when calculating the multiplicative factor. To get closer to our targeted specifications, we may use the obtained V_{DS} values for each transistor, from the original design, to estimate new multiplicative factor values by taking the term $(1+\lambda V_{DS})$ into account. Table B.15 shows the revised multiplicative factor values.

Table B.15 Revised Transistor Multiplicative Factors

Transistor	m
1	6
2	6
3	14
4	14
5	13
6	26
7	13
8	13

The simulation results show $|A_1| = 54$ V/V, $|A_2| = 58.2$ V/V, $|A_v| = 3145$ V/V, $I_{(Q1,Q2,Q3\text{ and }Q4)} = 103\,\mu$A, $I_{Q5} = 206\,\mu$A, $I_{(Q6,Q7)} = 205\,\mu$A, and $I_{Q8} = 200\,\mu$A, from which we see that the voltage gains are closer to the targeted specifications.

One should note that the discrepancies between the hand-design and simulation results in this simulation example are more apparent because errors in each stage add up.

Next, we will explore some important characteristics of the designed two-stage CMOS op amp.

Input Common-Mode Range The upper limit of the input common-mode range is the value of input voltage at which Q_1 and Q_2 leave the saturation region. This occurs when the input voltage exceeds the drain voltage of Q_1 by $V_{tn} = 0.5$ V. Since the drain of Q_1 is at $1-(0.21+0.5) = 0.29$ V, then the upper limit of the input common-mode range is $v_{ICM\max} = 0.29+0.5 = 0.79$ V.

The lower limit of the input common-mode range is the value of input voltage at which Q_5 leaves the saturation region. Since for Q_5 to operate in saturation the voltage across it (i.e., V_{DS5}) should at least be equal to the overdrive voltage at which it is operating (i.e., 0.21 V), the highest voltage permitted at the drain of Q_5 should be –0.79 V. It follows that the lowest value of v_{ICM} should be $v_{ICM\min} = -0.08$ V.

To verify the results using the simulation tool, we swept the input common-mode voltage v_{ICM} from –1 V to 1 V and plotted the resulting v_{GD} of Q_1 and Q_5. As can be seen from Fig. B.34, both transistors Q_1 and Q_5 stay in saturation for the input common-mode range of $-0.08\text{V} \le v_{ICM} \le 0.79$ V, as indicated by cursors.

Example S.9.2 *continued*

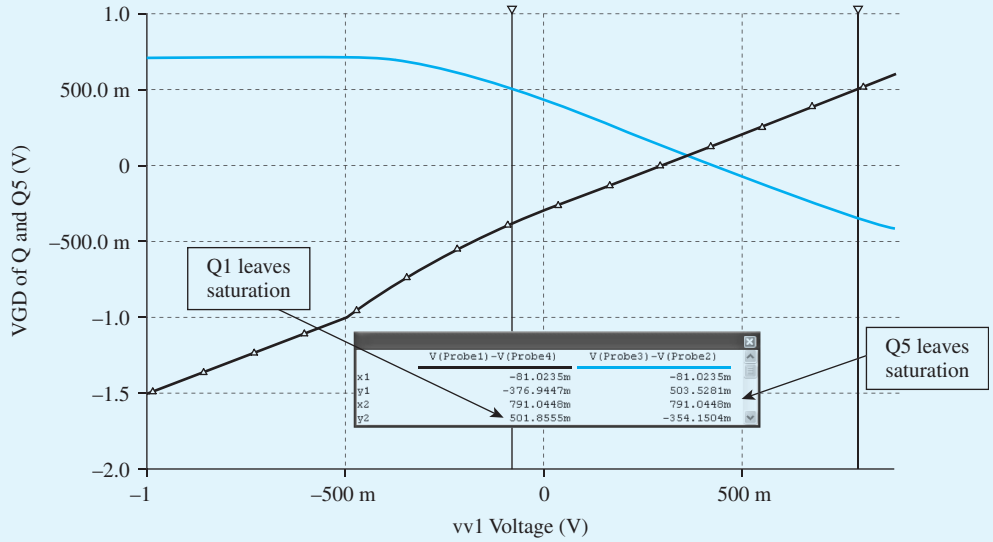


Figure B.34 Input common–mode range of the two-stage CMOS op amp.

Common-Mode Rejection Ratio (CMRR) of the First Stage The value of the CMRR of the first stage (the active-loaded MOS differential amplifier) is determined from Eq. (9.138). Note that the value of R_{SS} in the provided equation corresponds to the output resistance of Q_5 (i.e., r_{o5}). Thus,

$$\text{CMRR} \equiv \frac{|A_1|}{|A_{cm}|} = \frac{50}{1/2g_{m3}r_{o5}} = 100g_{m3}r_{o5} = 100\frac{2\times 100\times 10^{-6}}{0.21}\frac{12.5}{200\times 10^{-6}} = 5952.4 = 75.5\ \text{dB}$$

Using the simulation tool, the value of CMRR is calculated by dividing the previously obtained A_1 value (54 V/V) by the common-mode gain of the first stage. This yields

$$\text{CMRR} \equiv \frac{|A_1|}{|A_{cm}|} = \frac{54}{78\times 10^{-3}} = 6923 = 76.8\,\text{dB}$$

Output Voltage Range The lowest allowable output voltage is the value at which Q_7 leaves the saturation region, which is $-V_{SS}+V_{OV7} = -1+0.21 = 0.79$ V. The highest allowable output voltage is the value at which Q_6 leaves saturation, which is $V_{DD}-|-V_{OV6}| = 1-0.21 = 0.79$ V. Thus, the output-voltage range is –0.79 V to 0.79 V.

To verify the calculated output voltage range, we swept the input voltage from –2 mV to 2 mV (we used a small input voltage due to high gain). As can be seen from Fig. B.35, the output level changes from –0.795 V to 0.784 V, a rather symmetrical range. Therefore, the simulation results confirm our hand-analysis calculations.

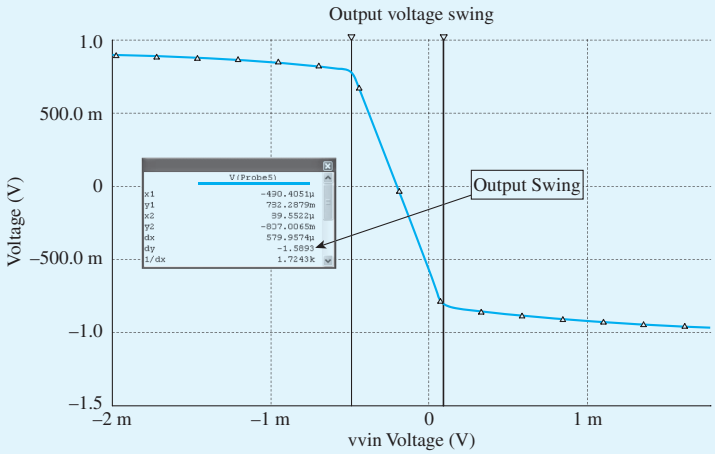


Figure B.35 Output-voltage range of the two-stage CMOS op amp.

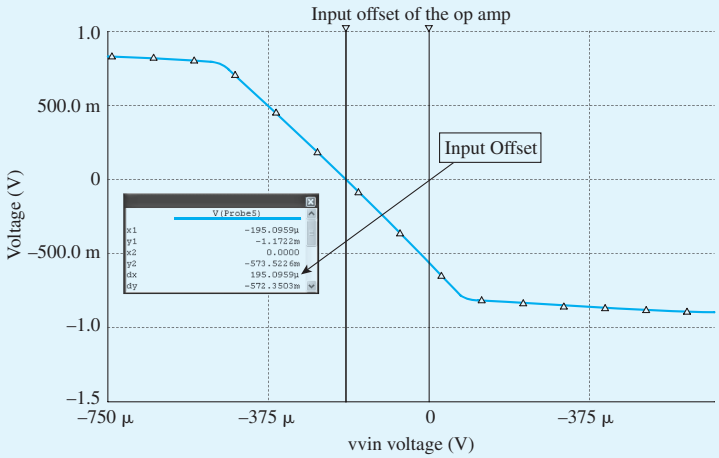


Figure B.36 Input offset voltage of the two-stage CMOS op amp.

Input Offset Voltage Although, theoretically, there should be no systematic offset, we do observe an output offset voltage V_O.

If we apply a voltage $-V_{OS}$ between the input terminals of the differential amplifier, the output voltage should be reduced to zero. This equivalency can be verified using the simulation tool When both the input terminals are grounded, the probe at the output reads the dc voltage 0.574 V. Also, when we apply the voltage $V_{OS} = (0.574/3145) \simeq 183\,\mu\text{V}$, between the input terminals, the output voltage is reduced to zero (Fig. B.36). Hence, the op amp has an input offset voltage of $V_{OS} = 195\,\mu\text{V}$, which approximately corresponds to an output offset voltage of $V_O = 0.574$ V.

Example S.10.1

Frequency Response of the CMOS CS and the Folded-Cascode Amplifiers

In this example, we will use SPICE to compute the frequency response of both the CS and the folded-cascode amplifiers whose schematic capture diagrams are shown shortly in Figs. B.37 and B.39, respectively. We will assume that the dc bias levels at the output of the amplifiers are stabilized using negative feedback. However, before performing a small-signal analysis (an ac-analysis simulation) in SPICE to measure the frequency response, we will perform a dc analysis (a bias-point simulation) to verify that all MOSFETs are operating in the saturation region and, hence, ensure that the amplifier is operating in its linear region.

In the following, we will assume a 0.5-μm CMOS technology for the MOSFETs and use parts NMOSOP5 and PMOSOP5 whose SPICE level-1 model parameters are listed in Table B.3. To specify the dimensions of the MOSFETs in SPICE, we will use the multiplicative factor m, together with the channel length L and channel width W (as we did in Example S.7.1).

The CMOS CS Amplifier

The CS amplifier circuit in Fig. B.37 is identical to the one shown in Fig. 7.4, except that a current source is connected to the source of the input transistor M_1 to set its drain current I_{D1} independently of its drain voltage V_{D1}. Furthermore, in our SPICE simulations, we used an impractically large bypass capacitor C_S of 1 F. This sets the source of M_1 at approximately signal ground during the ac-analysis

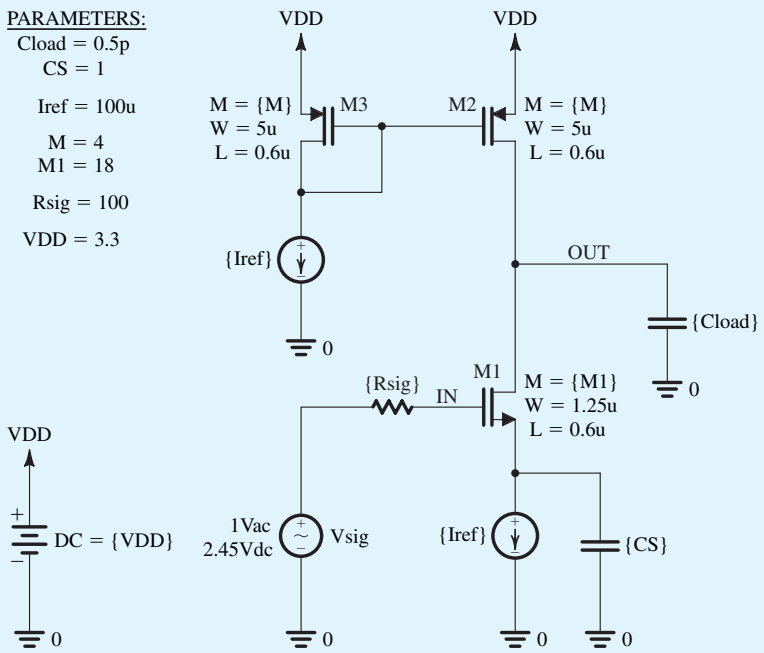


Figure B.37 Schematic capture of the CS amplifier in Example S.10.1

simulation. Accordingly, the CS amplifier circuits in Figs. 7.4 and B.37 are equivalent for the purpose of frequency-response analysis. In Chapter 9, we found out, in the context of studying the differential pair, how the goals of this biasing approach for the CS amplifier are realized in practical IC implementations.

The CS amplifier in Fig. B.37 is designed assuming a reference current $I_{ref} = 100\,\mu A$ and $V_{DD} = 3.3$ V. The current-mirror transistors, M_2 and M_3, are sized for $V_{OV2} = V_{OV3} = 0.3$ V, while the input transistor M_1 is sized for $V_{OV1} = 0.15$ V. Unit-size transistors are used with $W/L = 1.25\ \mu m/0.6\ \mu m$ for the NMOS devices and $W/L = 5\ \mu m/0.6\ \mu m$ for the PMOS devices. Thus, using the square law $I_D - V_{OV}$ of the MOSFET together with the 0.5-μm CMOS process parameters in Table B.3, we find $m_1 = 18$ and $m_2 = m_3 = 4$. Furthermore, Eq. (B.25) gives $G_v = -44.4$ V/V for the CS amplifier.

In the SPICE simulations of the CS amplifier in Fig. B.37, the dc bias voltage of the signal source is set such that the voltage at the source terminal of M_1 is $V_{S1} = 1.3$ V. This requires the dc level of V_{sig} to be $V_{OV1} + V_{tn1} + V_{S1} = 2.45$ V because $V_{tn1} \simeq 1$ V as a result of the body effect on M_1. The reasoning behind this choice of V_{S1} is that, in a practical circuit implementation, the current source that feeds the source of M_1 is realized using a cascode current mirror such as the one in Fig. 8.30. In this case, the minimum voltage required across the current source (i.e., the minimum V_{S1}) is $V_t + 2V_{OV} = 1.3$ V, assuming $V_{OV} = 0.3$ V for the current-mirror transistors.

A bias-point simulation is performed in SPICE to verify that all MOSFETs are biased in the saturation region. Next, to compute the frequency response of the amplifier, we set the ac voltage of the signal source to 1 V, perform an ac-analysis simulation, and plot the output voltage magnitude versus frequency. Figure B.38(a) shows the resulting frequency response for $R_{sig} = 100\,\Omega$ and $R_{sig} = 1$ MΩ. In both cases, a load capacitance of $C_{load} = 0.5$ pF is used. The corresponding values of the 3-dB frequency f_H of the amplifier are given in Table B.16.

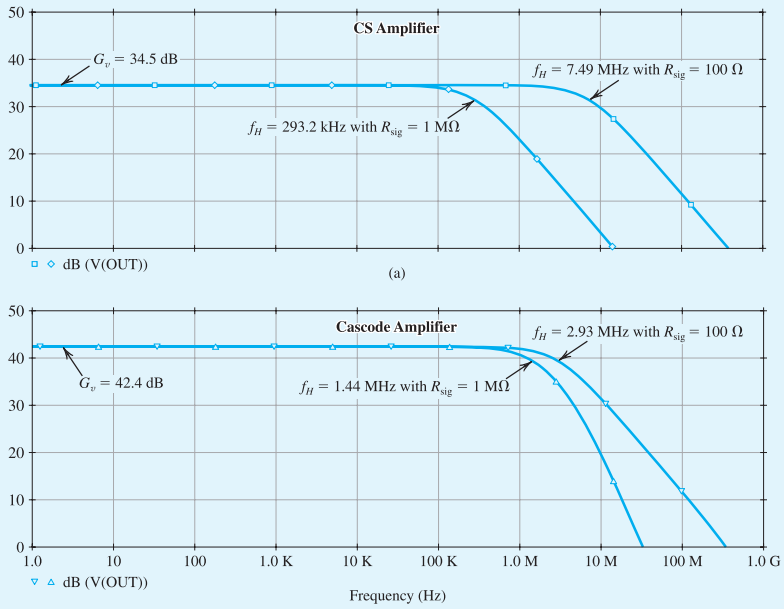


Figure B.38 Frequency response of **(a)** the CS amplifier and **(b)** the folded-cascode amplifier in Example S.10.1, with $R_{sig} = 100\,\Omega$ and $R_{sig} = 1$ MΩ.

Example S.10.1 *continued*

Table B.16 Dependence of the 3-dB Bandwidth f_H on R_{sig} for the CS and the Folded-Cascode Amplifiers in Example S.10.1

	f_H	
R_{sig}	CS Amplifier	Folded-Cascode Amplifier
100 Ω	7.49 MHz	2.93 MHz
1 MΩ	293.2 kHz	1.44 MHz

Observe that f_H drops when R_{sig} is increased. This is anticipated from our study of the high-frequency response of the CS amplifier in Section 10.2. Specifically, as R_{sig} increases, the pole

$$f_{p,\text{in}} = \frac{1}{2\pi}\frac{1}{R_{sig}C_{in}} \tag{B.27}$$

formed at the amplifier input will have an increasingly significant effect on the overall frequency response of the amplifier. As a result, the effective time constant τ_H in Eq. (10.80) increases and f_H decreases. When R_{sig} becomes very large, as it is when $R_{sig} = 1$ MΩ, a dominant pole is formed by R_{sig} and C_{in}. This results in

$$f_H \simeq f_{p,\text{in}} \tag{B.28}$$

To estimate $f_{p,\text{in}}$, we need to calculate the input capacitance C_{in} of the amplifier. Using Miller's theorem, we have

$$\begin{aligned} C_{in} &= C_{gs1} + C_{gd1}(1 + g_{m1}R_L') \\ &= \left(\frac{2}{3}m_1W_1L_1C_{ox} + C_{gs,ov1}\right) + C_{gd,ov1}(1 + g_{m1}R_L') \end{aligned} \tag{B.29}$$

where

$$R_L' = r_{o1} \parallel r_{o2} \tag{B.30}$$

Thus, C_{in} can be calculated using the values of C_{gs1} and C_{gd1}, which are computed by SPICE and can be found in the output file of the bias-point simulation. Alternatively, C_{in} can be found using Eq. (B.29) with the values of the overlap capacitances $C_{gs,ov1}$ and $C_{gsd,ov1}$ calculated using the process parameters in Table B.4 (as described in Eqs. B.9 and B.10); that is:

$$C_{gs,ov1} = m_1W_1CGSO \tag{B.31}$$

$$C_{gd,ov1} = m_1W_1CGDO \tag{B.32}$$

This results in $C_{in} = 0.53$ pF when $|G_v| = g_{m1}R_L' = 53.2$ V/V. Accordingly, using Eqs. (B.27) and (B.28), $f_H = 300.3$ kHz when $R_{sig} = 1$ MΩ, which is close to the value computed by SPICE.

The Folded-Cascode Amplifier

The folded-cascode amplifier circuit in Fig. B.39 is equivalent to the one in Fig. 13.8, except that a current source is placed in the source of the input transistor M_1 (for the same dc-biasing purpose as in the case of

the CS amplifier). Note that, in Fig. B.39, the PMOS current mirror M_3–M_4 and the NMOS current mirror M_5–M_6 are used to realize, respectively, current sources I_1 and I_2 in the circuit of Fig. 13.8. Furthermore, the current transfer ratio of mirror M_3–M_4 is set to 2 (i.e., $m_3/m_4 = 2$). This results in $I_{D3} \simeq 2I_{\text{ref}}$. Hence, transistor M_2 is biased at $I_{D2} = I_{D3} - I_{D1} = I_{\text{ref}}$. The gate bias voltage of transistor M_2 is generated using the diode-connected transistors M_7 and M_8. The size and drain current of these transistors are set equal to those of transistor M_2. Therefore, ignoring the body effect,

$$V_{G2} = V_{DD} - V_{SG7} - V_{SG8} \simeq V_{DD} - 2(|V_{tp}| + |V_{OVp}|)$$

where V_{OVp} is the overdrive voltage of the PMOS transistors in the amplifier circuit. These transistors have the same overdrive voltage because their I_D/m is the same. Thus, such a biasing configuration results in $V_{SG2} = |V_{tp}| + |V_{OVp}|$ as desired, while setting $V_{SD3} = |V_{tp}| + |V_{OVp}|$ to improve the bias matching between M_3 and M_4.

The folded-cascode amplifier in Fig. B.39 is designed assuming a reference current $I_{\text{ref}} = 100\ \mu\text{A}$ and $V_{DD} = 3.3$ V (similar to the case of the CS amplifier). All transistors are sized for an overdrive voltage of 0.3 V, except for the input transistor M_1, which is sized for $V_{OV1} = 0.15$ V. Thus, since $I_D = \frac{1}{2}\mu m C_{ox} m \left(W/L_{\text{eff}}\right) V_{OV}^2$, all the MOSFETs in the amplifier circuit are designed using $m = 4$, except for $m_1 = 18$.

The midband voltage gain of the folded-cascode amplifier in Fig. B.39 can be expressed as

$$G_v = -g_{m1} R_{\text{out}} \tag{B.33}$$

where

$$R_{\text{out}} = R_{\text{out2}} \,\|\, R_{\text{out5}} \tag{B.34}$$

is the output resistance of the amplifier. Here, R_{out2} is the resistance seen looking into the drain of the cascode transistor M_2, while R_{out5} is the resistance seen looking into the drain of the current-mirror transistor M_5.

$$R_{\text{out2}} \simeq \left(g_{m2} r_{o2}\right) R_{s2} \tag{B.35}$$

where

$$R_{s2} = r_{o1} \,\|\, r_{o3} \tag{B.36}$$

is the effective resistance at the source of M_2. Furthermore,

$$R_{\text{out5}} = r_{o5} \tag{B.37}$$

Thus, for the folded-cascoded amplifier in Fig. B.39,

$$R_{\text{out}} \simeq r_{o5} \tag{B.38}$$

and

$$G_v \simeq -g_{m1} r_{o5} = -2\frac{V_{An}}{V_{OV1}} \tag{B.39}$$

Using the 0.5-μm CMOS parameters, this gives $R_{\text{out}} = 100$ kΩ and $G_v = -133$ V/V. Therefore, R_{out} and hence $|G_v|$ of the folded-cascode amplifier in Fig. B.39 are larger than those of the CS amplifier in Fig. B.37 by a factor of 3.

Figure B.38(b) shows the frequency response of the folded-cascode amplifier as computed by SPICE for the cases of $R_{\text{sig}} = 100\ \Omega$ and $R_{\text{sig}} = 1$ MΩ. The corresponding values of the 3-dB frequency f_H of the amplifier are given in Table B.16. Observe that when R_{sig} is small, f_H of the folded-cascode amplifier is lower than that of the CS amplifier by a factor of approximately 2.6, approximately equal to the factor by

Example S.10.1 *continued*

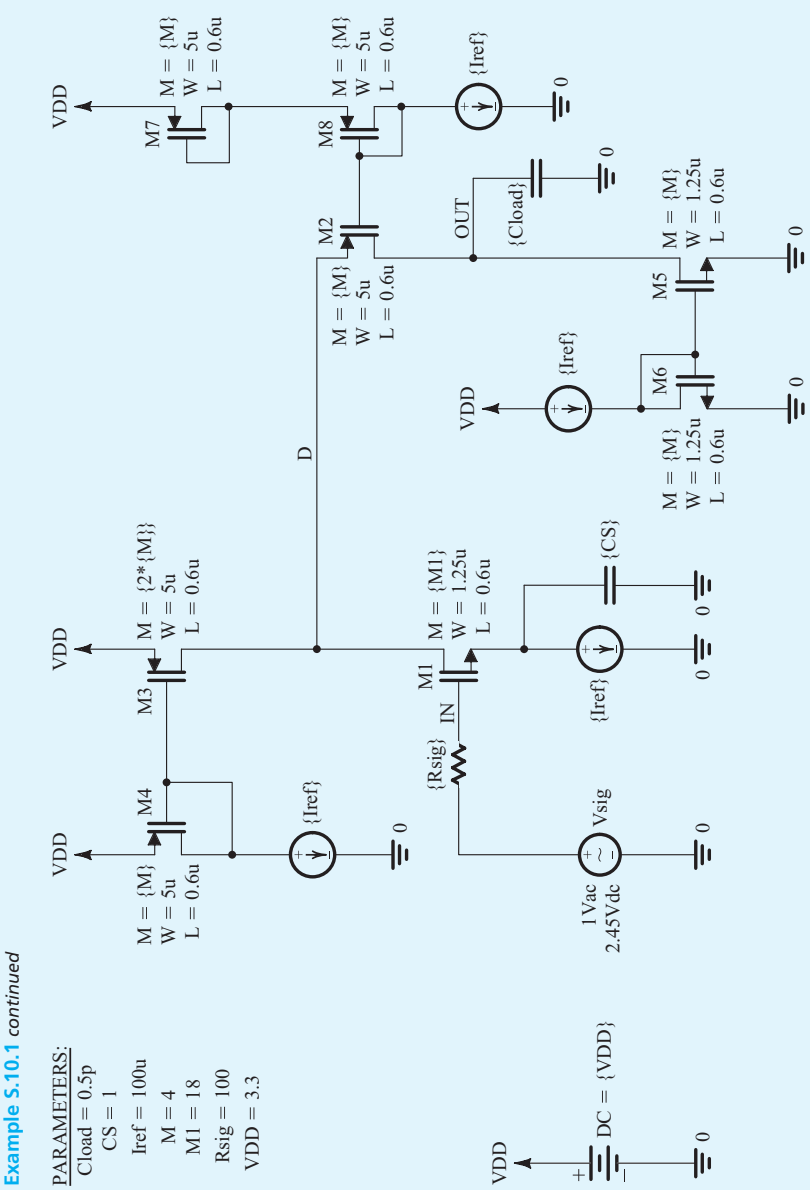


Figure B.39 Schematic capture of the of folded-cascode amplifier in Example S.10.1.

which the gain is increased. This is because when R_{sig} is small, the frequency response of both amplifiers is dominated by the pole formed at the output node, that is,

$$f_H \simeq f_{p,\text{out}} = \frac{1}{2\pi}\frac{1}{R_{\text{out}}C_{\text{out}}} \tag{B.40}$$

Since the output resistance of the folded-cascode amplifier is larger than that of the CS amplifier (by a factor of approximately 3, as found through the hand analysis above) while their output capacitances are approximately equal, the folded-cascode amplifier has a lower f_H in this case.

On the other hand, when R_{sig} is large, f_H of the folded-cascode amplifier is much higher than that of the CS amplifier. This is because, in this case, the effect of the pole at $f_{p,\text{in}}$ on the overall frequency response of the amplifier becomes significant. Since, due to the Miller effect, C_{in} of the CS amplifier is much larger than that of the folded-cascode amplifier, its f_H is much lower in this case. To confirm this point, observe that C_{in} of the folded-cascode amplifier can be estimated by replacing R_L' in Eq. (B.29) with the total resistance R_{d1} between the drain of M_1 and ground. Here,

$$R_{d1} = r_{o1} \| r_{o3} \| R_{\text{in2}} \tag{B.41}$$

where R_{in2} is the input resistance of the common-gate transistor M_2 and can be obtained using an approximation of the relationship in Eq. (8.51) as

$$R_{\text{in2}} \simeq \frac{r_{o2} + r_{o5}}{g_{m2}r_{o2}} \tag{B.42}$$

Thus,

$$R_{d1} \simeq r_{o1} \| r_{o3} \| \left(\frac{r_{o2} + r_{o5}}{g_{m2}r_{o2}} \simeq \frac{2}{g_{m2}} \right) \tag{B.43}$$

Therefore, R_{d1} is much smaller than R_L' in Eq. (B.30). Hence, C_{in} of the folded-cascode amplifier in Fig. B.39 is indeed much smaller than that of the CS amplifier in Fig. B.37. This confirms that the folded-cascode amplifier is much less impacted by the Miller effect and, therefore, can achieve a much higher f_H when R_{sig} is large.

The midband gain of the folded-cascode amplifier can be significantly increased by replacing the current mirror M_5–M_6 with a current mirror having a larger output resistance, such as the cascode current mirror in Fig. 8.36 whose output resistance is approximately $g_m r_o^2$. In this case, however, R_{in2} and hence R_{d1} increase, causing an increased Miller effect and a corresponding reduction in f_H.

Finally, it is interesting to observe that the frequency response of the folded-cascode amplifier, shown in Fig. B.38(b), drops beyond f_H at approximately –20 dB/decade when $R_{\text{sig}} = 100\,\Omega$ and at approximately -40 dB/decade when $R_{\text{sig}} = 1$ MΩ. This is because when R_{sig} is small, the frequency response is dominated by the pole at $f_{p,\text{out}}$. However, when R_{sig} is increased, $f_{p,\text{in}}$ is moved closer to $f_{p,\text{out}}$ and both poles contribute to the gain falloff.

Example S.10.2

Frequency Response of the Discrete CS Amplifier

In this example, we will investigate the frequency response of the CS amplifier of Example S.7.3. By using SPICE to perform "ac analysis" on the designed CS amplifier, we are able to measure the midband gain A_M and the 3-dB frequencies f_L and f_H, and to plot the output-voltage magnitude (in dB) versus frequency. Figure B.40 shows the schematic capture of the CS amplifier.

DEVICE PARAMETERS	
NAME	Q1:NMOS
W	15.48u
L	0.2u
KP	246u
LD	0.01u
VT0	0.45
LAMBDA	0.08
GAMMA	0.3

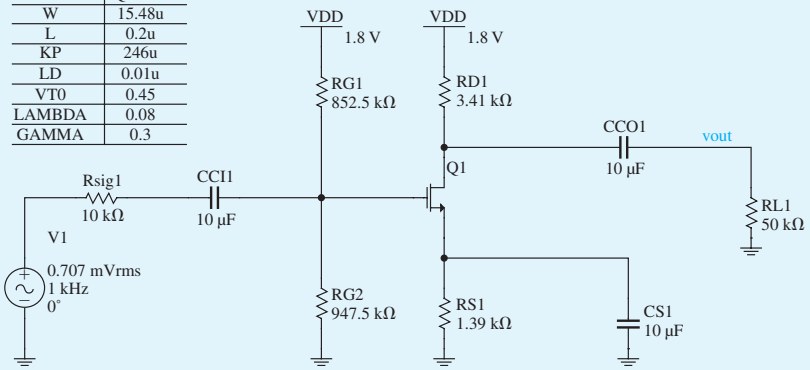


Figure B.40 Schematic capture of discrete CS amplifier.

Hand Analysis

Midband Gain The midband gain of this CS amplifier can be determined as follows:

$$A_M = \frac{R_{\text{in}}}{R_{\text{in}} + R_{\text{sig}}}\left[g_m\left(R_D \parallel R_L\right)\right]$$

$$R_{\text{in}} = \left(R_{G1} \parallel R_{G2}\right) = 852.5 \times 10^3 \parallel 947.5 \times 10^3 = 448.75 \times 10^3\ \Omega$$

$$g_m = 3.33\,\text{mA/V}$$

$$A_M = \frac{448.75 \times 10^3}{448.75 \times 10^3 + 10 \times 10^3}\left[3.33 \times 10^{-3}\left(3.41 \times 10^3 \parallel 50 \times 10^3\right)\right] \simeq 10\,\text{V/V}$$

Low-Frequency Poles and Zero We know from Section 10.8.2 that the low-frequency poles are as follows:

$$f_{P1} = \frac{1}{2\pi \times C_{CI}\left(R_{\text{sig}} + R_{\text{in}}\right)} = \frac{1}{2\pi \times 10 \times 10^{-6}\left[\left(10 \times 10^3\right) + 448.75 \times 10^3\right]}$$

$$f_{P1} = 0.0347\,\text{Hz}$$

$$f_{P2} = \frac{1}{2\pi \times C_{CO}\left(R_D + R_L\right)} = \frac{1}{2\pi \times 10 \times 10^{-6}\left(3.41 \times 10^3\right) + \left(50 \times 10^3\right)}$$

$$f_{P2} = 0.30\,\text{Hz}$$

$$f_{P3} = \frac{1}{2\pi \times C_S}\left(g_m + \frac{1}{R_S}\right) = \frac{1}{2\pi \times 10 \times 10^{-6}}\left[\left(3.33 \times 10^{-3}\right) + \frac{1}{1.39 \times 10^3}\right]$$

$$f_{P3} = 64.4\,\text{Hz}$$

And the location of the real transmission zero is determined as

$$f_Z = \frac{1}{2\pi \times C_S R_S} = \frac{1}{2\pi \times \left(10 \times 10^{-6}\right)\left(1.39 \times 10^{3}\right)}$$

$$f_Z = 11.45\,\text{Hz}$$

Upon observing the relative magnitude of each of the poles, we can conclude that f_{P3} will determine f_L, the lower 3-dB frequency of the amplifier gain,

$$f_L \simeq f_{P3} \simeq 11.45\ \text{Hz}$$

High-Frequency Rolloff The high-frequency rolloff of the amplifier gain is caused by the MOSFET internal capacitance. The typical values for 0.180 μm CMOS technology are given in Table B.3. We know from Section 10.2 that

$$f_H = \frac{1}{2\pi \times C_{\text{in}} R'_{\text{sig}}}$$

$$R'_{\text{sig}} = 10 \times 10^{3} \,\|\, 448.75 \times 10^{3} = 9.78 \times 10^{3}$$

$$C_{\text{in}} = W\left\{C_{gs0} + C_{gd0}\left[1 + g_m\left(R_L \,\|\, R_L\right)\right]\right\}$$

Note that C_{gs0} and C_{gd0} are per-unit-width values provided in the models.

$$C_{\text{in}} = (15.48 \times 10^{-6}) \times (0.3665 \times 10^{-9}) \times \left[1 + 1 + 3.33 \times 10^{-3}(50 \times 10^{3} \,\|\, 3.41 \times 10^{-3})\right]$$

$$C_{\text{in}} = 0.716\,\text{fF}$$

$$f_H = \frac{1}{2\pi \times 0.716 \times 10^{-15} \times 9.78 \times 10^{3}}$$

$$f_H \simeq 191\,\text{MHz}$$

Now we can determine the bandwidth, BW, of the CS amplifier:

$$BW = f_H - f_L$$

$$BW \simeq f_H = 191\ \text{MHz}$$

Simulation

Figure B.41 shows the magnitude plot of the frequency response of this CS amplifier.

Based on the simulation results, the midband gain is $A_M = 9.80$ V/V. Also, $f_L = 60.8$ Hz and $f_H = 192.2$ MHz, resulting in 3-dB bandwidth of $BW = f_L - f_H = 192.2$ MHz. Figure B.41 further shows that (moving toward the left) the gain begins to fall off at about 300 Hz, but flattens out again at about 12.2 Hz. This flattening in the gain at low frequencies is due to a real transmission zero introduced in the transfer function of the amplifier by R_S together with C_S, with a frequency $f_Z - = 1/2\pi R_S C_S = 11.45$ Hz. Students are encouraged to investigate this relationship by using the simulation tool to modify the values of R_S and C_S and observing the corresponding change in the zero frequency. Note this value of zero is typically between the break frequencies f_{P2} and f_{P3}.

Example S.10.2 *continued*

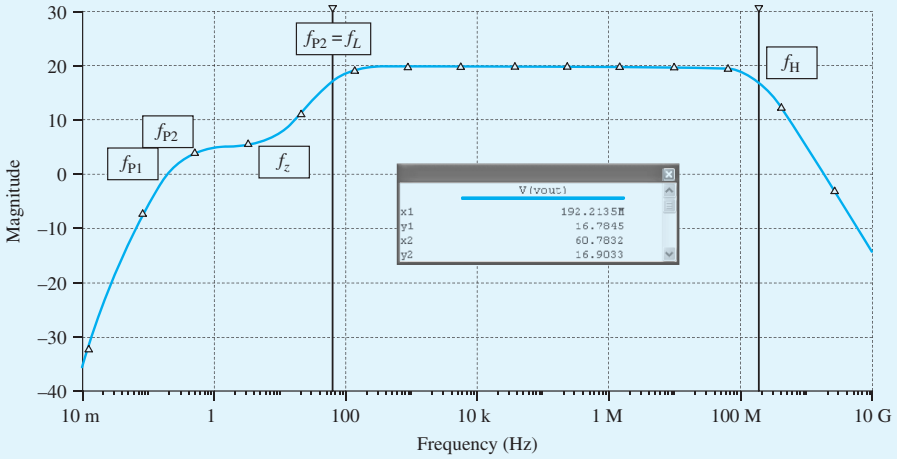


Figure B.41 Frequency response of the CS amplifier.

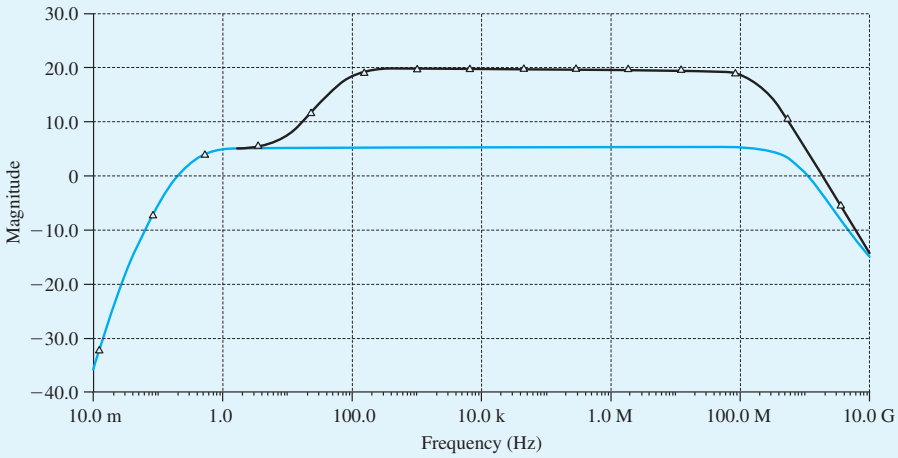


Figure B.42 Frequency response of the CS amplifier with $C_S = 10\ \mu F$ and $C_S = 0$.

We can further verify this phenomenon by resimulating the CS amplifier with a $C_S = 0$ (i.e., removing C_S) in order to move f_Z to infinity and remove its effect. The corresponding frequency response is plotted in Fig. B.42. As expected with $C_S = 0$, we do not observe any flattening in the low-frequency response of the amplifier. However, because the CS amplifier now includes a source resistor R_S, the value of A_M has dropped by a factor of 5.4. This factor is approximately equal to $(1 + g_m R_S)$, as expected from our study of the CS amplifier with a source-degeneration resistance. Note that the bandwidth BW has increased by approximately the same factor as the drop in gain A_M. As we learned in Chapter 11 in our study of negative feedback, the source-degeneration resistor R_S provides negative feedback, which allows us to trade off gain for a wider bandwidth.

Example S.11.1

Determining the Loop Gain of a Feedback Amplifier

This example illustrates the use of SPICE to compute the loop gain $A\beta$. For this purpose, we shall use the shunt–series feedback amplifier shown in Fig. B.43.

To compute the loop gain, we set the input signal V_s to zero, and we choose to break the feedback loop between the collector of Q_1 and the base of Q_2. However, in breaking the feedback loop, we must ensure that the following two conditions that existed prior to breaking the feedback loop do not change: (1) the dc bias situation and (2) the ac signal termination.

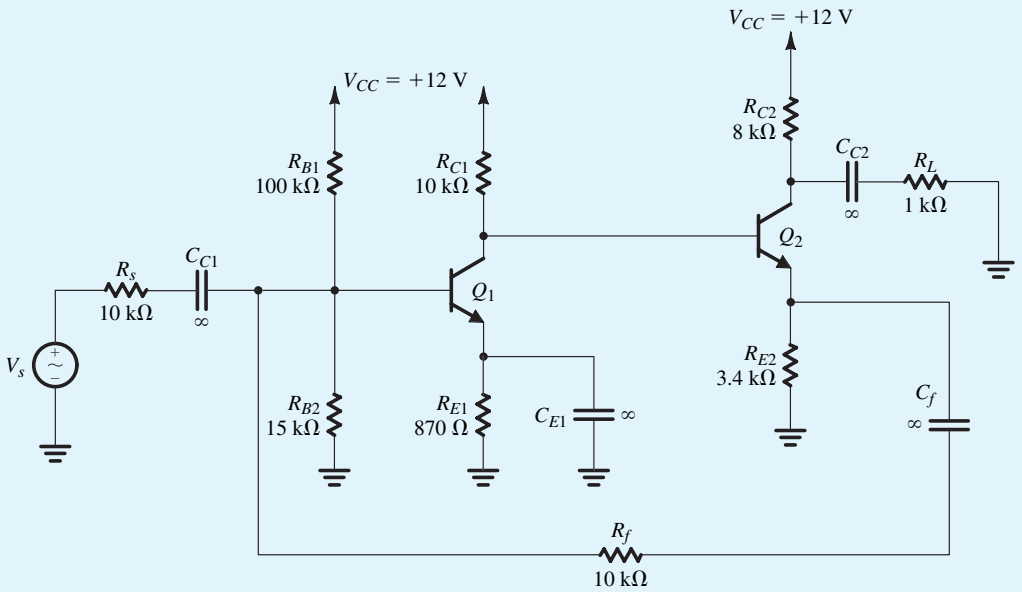


Figure B.43 Circuit of the shunt–series feedback amplifier in Example S.11.1.

To break the feedback loop without disturbing the dc bias conditions of the circuit, we insert a large inductor L_{break}, as shown in Fig. B.44(a). Using a value of, say, $L_{\text{break}} = 1$ GH will ensure that the loop is opened for ac signals while keeping dc bias conditions unchanged.

To break the feedback loop without disturbing the signal termination conditions, we must load the loop output at the collector of Q_1 with a termination impedance Z_t whose value is equal to the impedance seen looking into the loop input at the base of Q_2. Furthermore, to avoid disturbing the dc bias conditions, Z_t must be connected to the collector of Q_1 via a large coupling capacitor. However, it is not always easy to determine the value of the termination impedance Z_t. So, we will describe two simulation methods to compute the loop gain without explicitly determining Z_T.

Example S.11.1 *continued*

Method 1 *Using the open-circuit and short-circuit transfer functions*

Note that to avoid the problem of terminating the loop when we open it, the loop gain can be expressed as

$$A\beta = -1 \Big/ \left(\frac{1}{T_{oc}} + \frac{1}{T_{sc}} \right)$$

where T_{oc} is the open-circuit voltage transfer function and T_{sc} is the short-circuit voltage transfer function.

The circuit for determining T_{oc} is shown in Fig. B.44(b). Here, an ac test signal voltage V_t is applied to the loop input at the base of Q_2 via a large coupling capacitor (having a value of, say, 1 kF) to avoid disturbing the dc bias conditions. Then,

$$T_{oc} = \frac{V_{oc}}{V_t}$$

where V_{oc} is the ac open-circuit output voltage at the collector of Q_1.

In the circuit for determining T_{sc} (Fig. B.44b), an ac test signal current I_t is applied to the loop input at the base of Q_2. Note that a coupling capacitor is not needed in this case because the ac current source appears as an open circuit at dc, and, hence, does not disturb the dc bias conditions.

The loop output at the collector of Q_1 is ac short-circuited to ground via a large capacitor C_{to}. Then,

$$T_{sc} = \frac{I_{sc}}{I_t}$$

where I_{sc} is the ac short-circuit output current at the collector of Q_1.

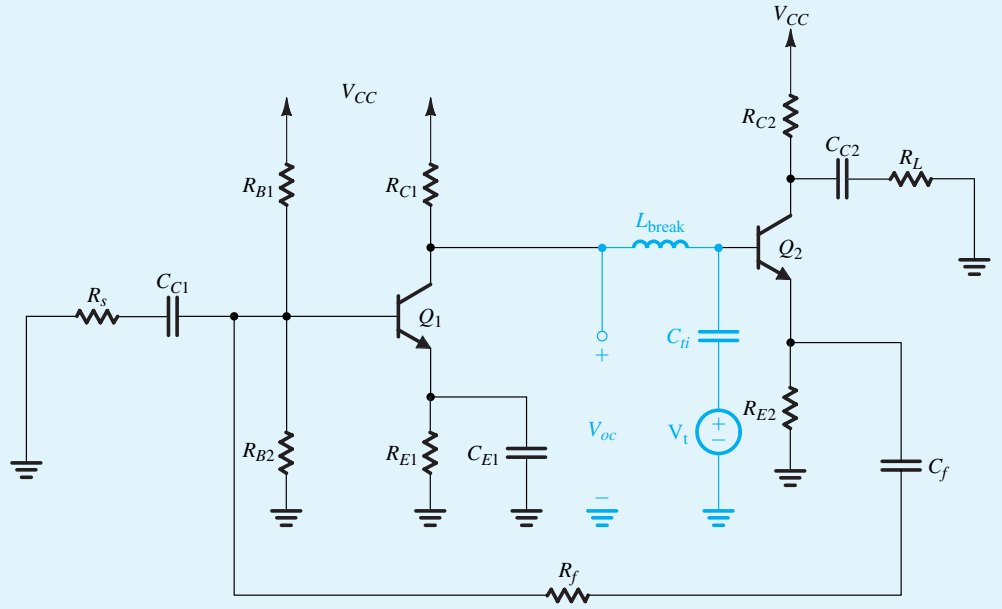
Method 2 *Using a replica circuit*

As shown in Fig. B.45, a replica of the feedback amplifier circuit can be simply used as a termination impedance. Here, the feedback loops of both the amplifier circuit and the replica circuit are broken using a large inductor L_{break} to avoid disturbing the dc bias conditions. The loop output at the collector of Q_1 in the amplifier circuit is then connected to the loop input at the base of Q_2 in the replica circuit via a large coupling capacitor C_{to} (again, to avoid disturbing the dc bias conditions). Thus, for ac signals, the loop output at the collector of Q_1 in the amplifier circuit sees an impedance equal to that seen before the feedback loop is broken. Accordingly, we have ensured that the conditions that existed in the amplifier circuit prior to breaking the loop have not changed.

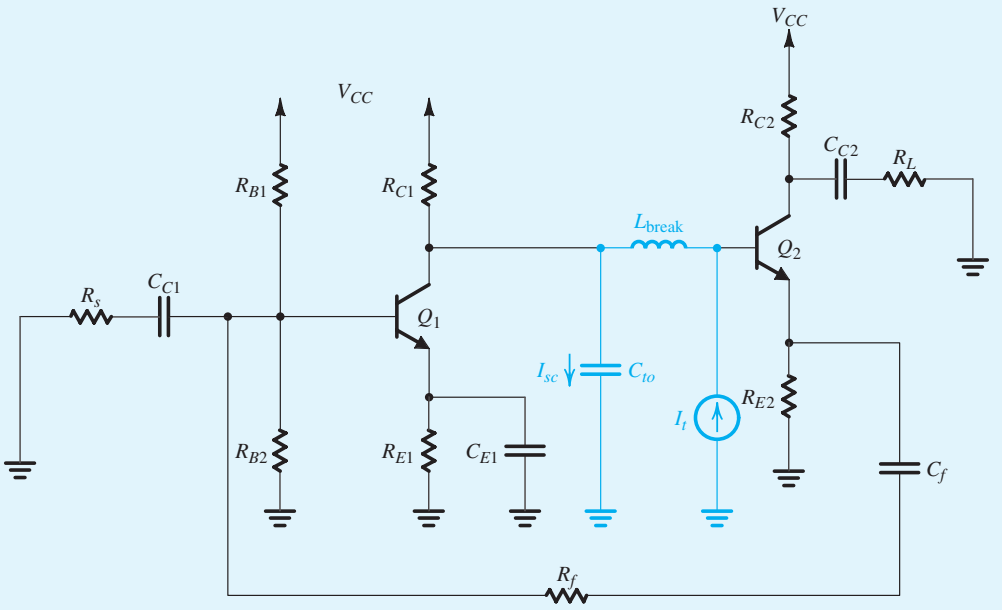
Next, to determine the loop gain $A\beta$, we apply an ac test signal voltage V_t via a large coupling capacitor C_{ti} to the loop input at the base of Q_2 in the amplifier circuit. Then, as described in Chapter 11,

$$A\beta = -\frac{V_r}{V_t}$$

where V_r is the ac returned signal at the loop output at the collector of Q_1 in the amplifier circuit.



(a)



(b)

Figure B.44 Circuits for simulating **(a)** the open-circuit voltage transfer function T_{oc} and **(b)** the short-circuit current transfer function T_{sc} of the feedback amplifier in Fig. B.43 for the purpose of computing its loop gain.

Example S.11.1 *continued*

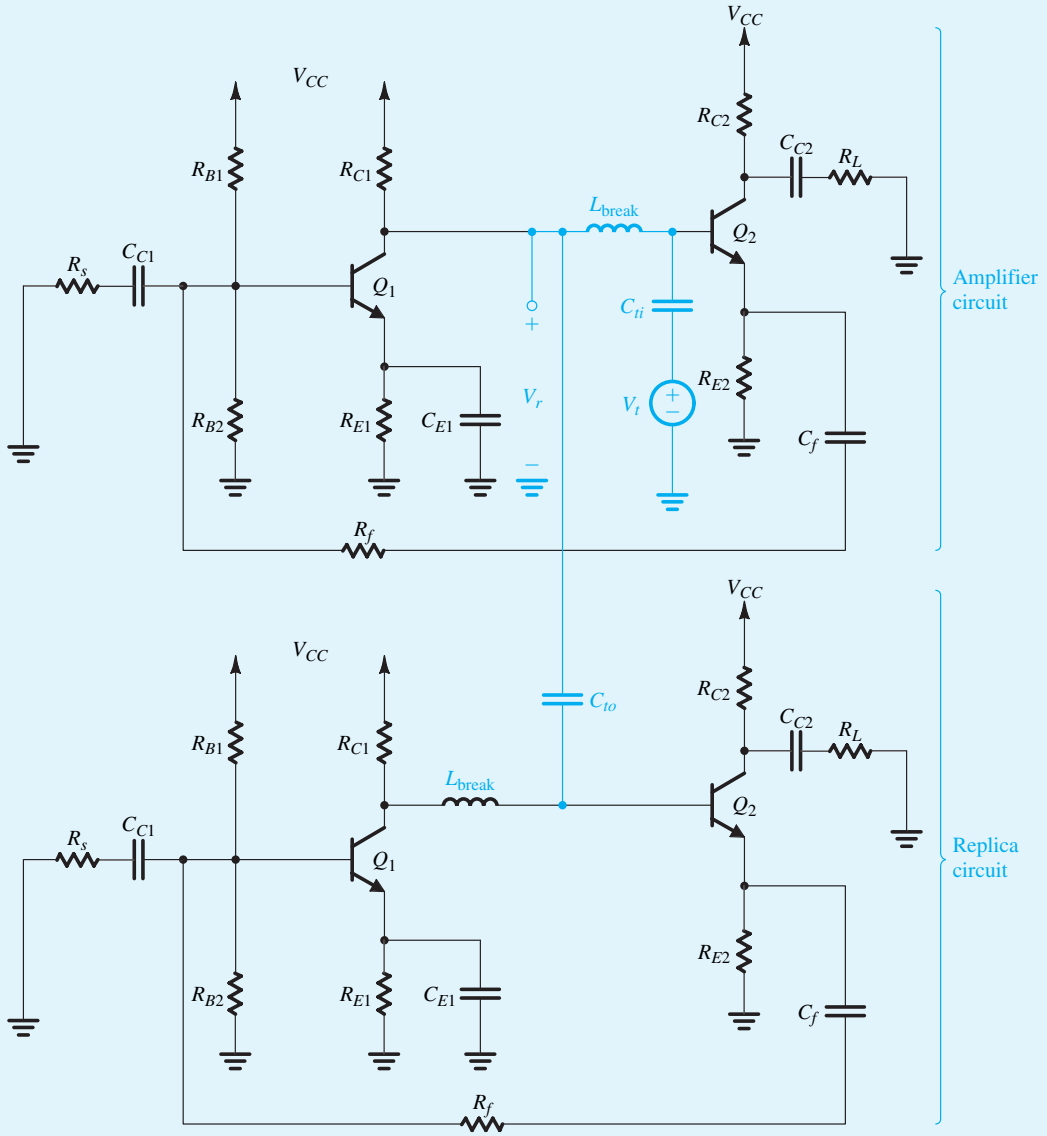


Figure B.45 Circuit for simulating the loop gain of the feedback amplifier circuit in Fig. B.43 using the replica-circuit method.

To compute the loop gain $A\beta$ of the feedback amplifier circuit in Fig. B.43 using SPICE, we choose to simulate the circuit in Fig. B.45. In the SPICE simulations, we used part Q2N3904 (whose SPICE model is given in Table B.6) for the BJTs, and we set L_{break} to be 1 GH and the coupling and bypass capacitors to be 1 kF. The magnitude and phase of $A\beta$ are plotted in Fig. B.46, from which we see that the feedback amplifier has a gain margin of 53.7 dB and a phase margin of 88.7°.

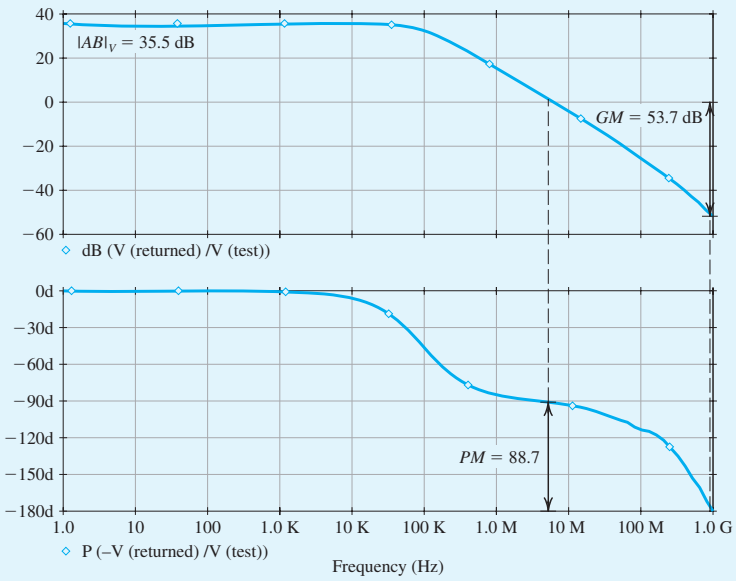


Figure B.46 **(a)** Magnitude and **(b)** phase of the loop gain $A\beta$ of the feedback amplifier circuit in Fig. B.43.

Example S.11.2

A Two-Stage CMOS Op Amp with Series–Shunt Feedback

In this example, we will investigate the effect of applying a series–shunt feedback to the two-stage CMOS op amp whose schematic capture is shown in Fig. B.47.

The first stage is a differential pair Q_1–Q_2 (which is actively loaded with the current mirror formed by Q_3 and Q_4) with bias current supplied by a current mirror formed by Q_8 and Q_5, which utilizes the reference bias current I_{REF}. The second stage consists of Q_6, which is a common-drain amplifier actively loaded with a current source load (transistor Q_7).

For the implementation of this CMOS op amp, we will use a 0.18-μm CMOS technology for the MOSFETs and typical SPICE level-1 model parameters for this technology, including the intrinsic capacitance values. The targeted specifications are an unloaded dc open-loop voltage gain $|A_v| = 50$ V/V, and closed-loop voltage gain $|A_f| = 10$ V/V, with each of transistors Q_1, Q_2, Q_3, and Q_4 biased at a drain current of 100 μA.

To achieve the targeted specifications, a biasing current $I_{REF} = 200$ μA is used, and the transistors Q_5, Q_6, Q_7, and Q_8 will be sized to conduct drain currents of 200 μA. The dc open-loop voltage gain for this amplifier is the product of the voltage gains of the two stages. Since the gain of the second stage (source follower) is approximately 1 V/V, the first stage must be designed to provide the full voltage gain of 50 V/V to achieve the specified open-loop voltage gain.

Example S.11.2 *continued*

The amplifier specifications are summarized in Table B.17.

Hand Design

Design of the Two-Stage Op Amp The first stage of this CMOS op amp is identical to the first stage of the op amp we designed in Example S.9.2, to which the reader is referred. Also, transistors Q_6 and Q_7 are sized to provide the bias current of 200 μA in the second stage.

NMOS	
Vtn	0.5 V
|VAn|	12.5V
kn'	246.2uA/V^2
L	0.2 um
W	0.48 um

PMOS	
Vtp	–0.5 V
|VAp|	9V
kp'	–86.1 μA/V^2
L	0.2 um
W	0.64 um

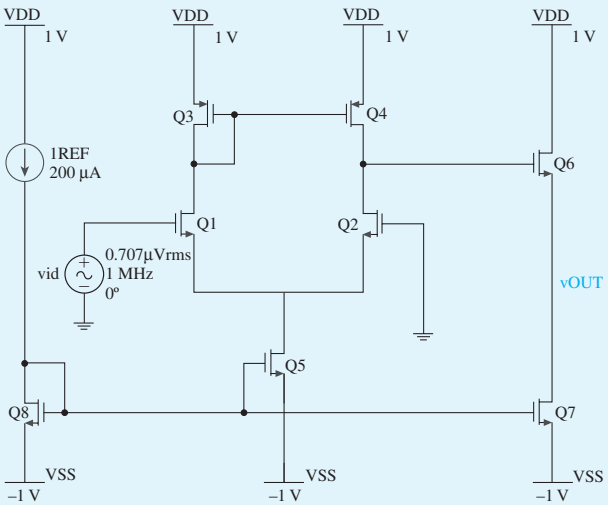


Figure B.47 Schematic capture of the two-stage CMOS op amp.

Table B.17 Two-Stage CMOS Op-Amp Specifications

Parameters	Value
$I_{(Q1,Q2,Q3,\text{and}\,Q4)}$	100 μA
$I_{(Q5,Q6,Q7,\text{and}\,Q8)}$	200 μA
$\|A_1\|$	50 V/V
$\|A_2\|$	1 V/V
$\|A_f\|$	10 V/V
V_{DD}	1 V
V_{SS}	−1 V

As a result, using unit-size NMOS transistors with $W_n/L_n = 0.48\,\mu\text{m}/0.20\,\mu\text{m}$, and unit-size PMOS transistors with $W_p/L_p = 0.64\,\mu\text{m}/0.20\,\mu\text{m}$, the corresponding multiplicative factor m for each transistor can be calculated as found in Example S.9.2 (with the difference here that Q_6 and Q_7 have the same dimensions). Table B.18 summarizes the relevant information and the calculated m values for the transistor.

Table B.18 Transistor Sizes

Transistor	I_D (μA)	m
1	100	6
2	100	6
3	100	14
4	100	14
5	200	13
6	200	13
7	200	13
8	200	13

Design of the Feedback Network First we need to determine the value of the feedback factor β for this series–shunt feedback amplifier. The β network can be implemented using a voltage divider, as shown in Fig. B.48. The resistor values are chosen large enough (in comparison to the output resistance of the designed two-stage op amp) to minimize the effect of loading. Therefore, effectively,

$$A \simeq A_v$$

where A is the open-loop gain of the amplifier (with loading). Now we can calculate the required feedback factor, β, as follows:

$$|A_f| = \frac{A_v}{1 + A_v\beta} = \frac{50}{1 + 50\beta} = 10\,\text{V/V}$$
$$\beta = 0.08$$

The resistor values of this voltage divider are selected to provide voltage divisions of 0.08 ($R_1 = 92\ \text{k}\Omega$ and $R_2 = 8\ \text{k}\Omega$).

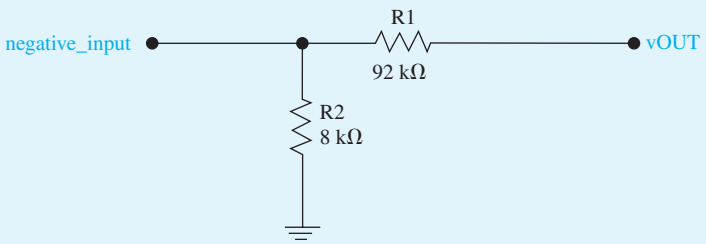


Figure B.48 β Network.

Simulation

Now we will simulate our designed circuit to verify our hand design and study the effect of feedback on the dc-gain, bandwidth, and output resistance of the amplifier.

Example S.11.2 *continued*

Verifying A_v The schematic capture of the two-stage CMOS amplifier is in Fig. B.49. We can verify the dc voltage gain of this amplifier by performing frequency-response analysis.

As can be seen from Fig. B.49, $|A_v| = 35.0\,\text{dB} \simeq 56.2\,\text{V/V}$, which is close to the targeted specification.

Verifying A The schematic capture of the A-circuit is given in Fig. B.50.

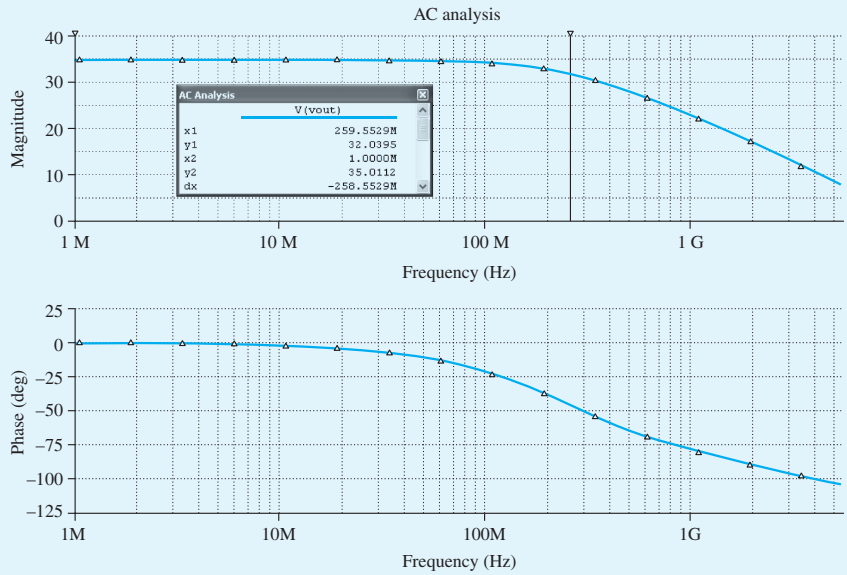


Figure B.49 Frequency response of the two-stage CMOS op-amp amplifier.

NMOS	
Vtn	0.5 V
|VAn|	12.5V
kn'	246.2μA/V^2
L	0.2 um
W	0.48 um

PMOS	
Vtp	−0.5 V
|VAp|	9V
kp'	−86.1 uA/V^2
L	0.2 um
W	0.64 um

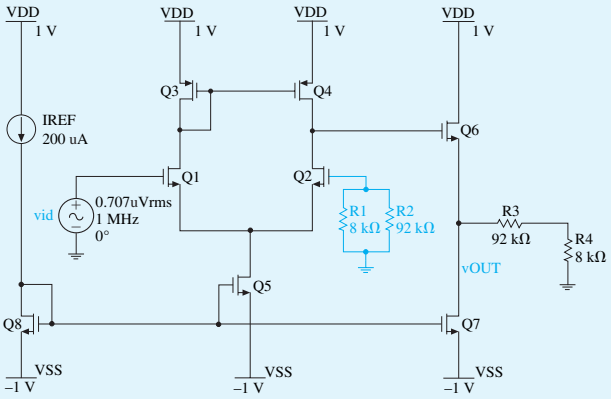


Figure B.50 Schematic capture of the A-circuit.

We can verify the open-loop voltage gain of this circuit by performing a frequency-response analysis. As can be seen from Fig. B.51, $|A| = 34.9$ dB $\simeq 55.6$ V/V, which is close to the value of A_v. This supports our assumption of $A \simeq A_v$.

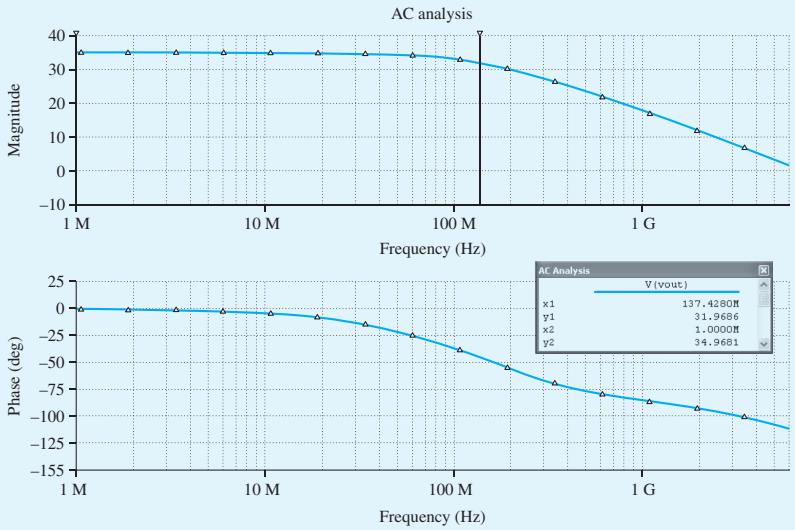


Figure B.51 Schematic capture of the *A*-circuit.

NMOS	
Vtn	0.5 V
|VAn|	12.5V
kn'	246.2uA/V^2
L	0.2 um
W	0.48 um

PMOS	
Vtp	–0.5 V
|VAp|	9V
kp'	–86.1 uA/V^2
L	0.2 um
W	0.64 um

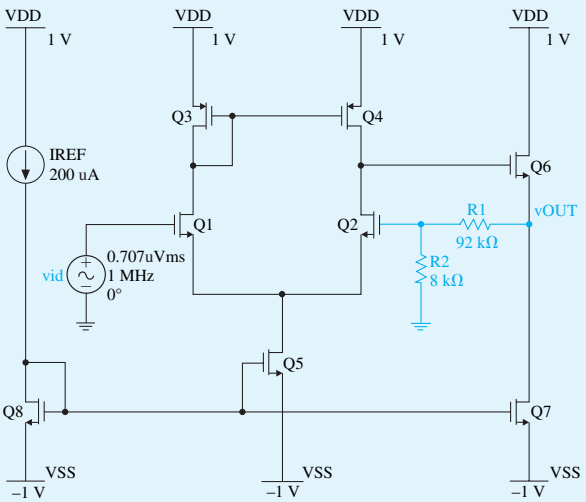


Figure B.52 Schematic capture of the closed-loop circuit.

Example S.11.2 *continued*

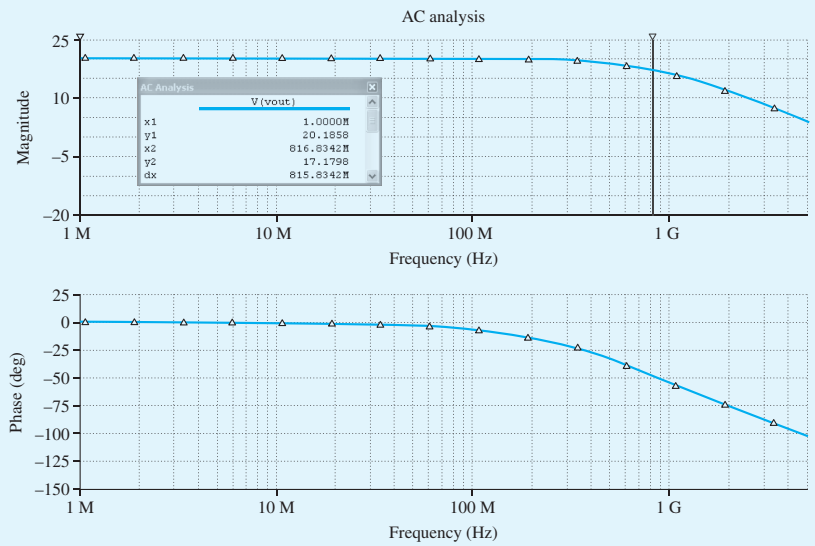


Figure B.53 Frequency response of the closed-loop circuit.

Verifying A_f The schematic capture of the closed-loop circuit is given in Fig. B.52. As can be seen from this schematic, the β-network establishes a series connection at the input and a shunt connection at the output of the original two-stage CMOS op amp.

We can verify the closed-loop voltage gain by performing a frequency-response analysis. As can be seen from Fig. B.53, $|A_f| = 20.2$ dB $\simeq 10.2$ V/V, which is close to the targeted specification for A_f.

Investigating the Effect of Feedback In addition to the frequency-response analysis, which provided information on the dc voltage gain and the 3-dB bandwidth, we used SPICE to find the output resistances of the open-loop and closed-loop circuits Table B.19 summarizes our findings for open-loop (A-circuit) and closed-loop circuits.

It can be seen from Table B.19 that the series–shunt feedback connection causes the dc voltage gain and the output resistance of the circuit to decrease by a factor of 5.5, while the 3-dB bandwidth increases by approximately the same factor. This factor is equal to $1 + A\beta$, the amount of the feedback. This is as expected and corresponds to what we learned in Chapter 11.

Table B.19 Effect of Feedback on Gain, 3-dB Bandwidth, and Output Resistor

Circuit	Gain (V/V)	3-dB Bandwidth (MHz)	R_{out} (Ω)
Open loop	55.6	137	492.6
Closed loop	10.2	816	89.3

Example S.12.1

Class B BJT Output Stage

We investigate the operation of the class B output stage whose schematic capture is shown in Fig. B.54. For the power transistors, we use the discrete BJTs MJE243 and MJE253 (from ON Semiconductor),[13] which are rated for a maximum continuous collector current $I_{Cmax} = 4$ A and a maximum collector–emitter voltage of $V_{CEmax} = 100$ V. To permit comparison with the hand analysis performed in Example 12.3, in the simulation, we use component and voltage values identical (or close) to those of the circuit designed in Example 12.3. Specifically, we use a load resistance of 8 Ω, an input sine-wave signal of 17.9-V peak and 1-kHz frequency, and 23-V power supplies. In SPICE, a transient-analysis simulation is performed over the interval 0 ms to 3 ms, and the waveforms of various node voltages and branch currents are plotted. In this example, the graphical interface of SPICE is utilized to compute various power-dissipation values.

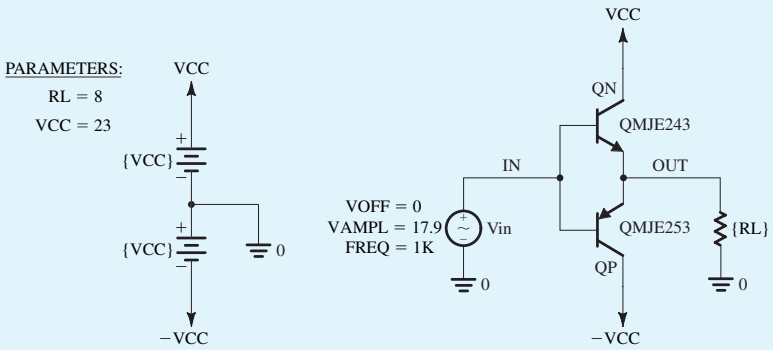


Figure B.54 Capture schematic of the class B output stage in Example S.12.1.

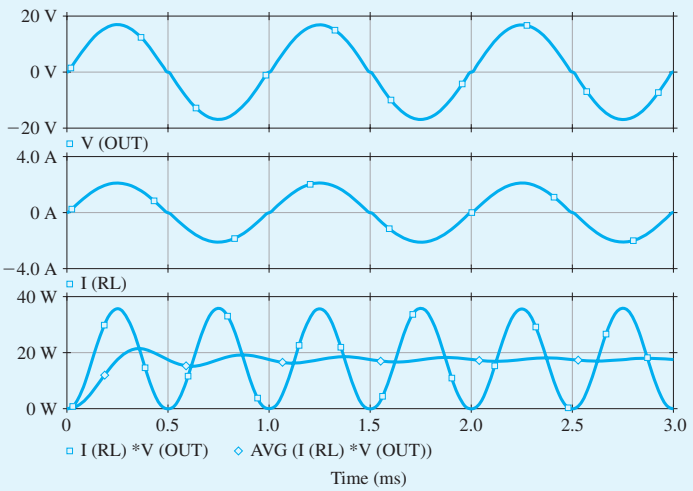


Figure B.55 Several waveforms associated with the class B output stage (shown in Fig. B.54) when excited by a 17.9-V, 1-kHz sinusoidal signal. The upper graph displays the voltage across the load resistance, the middle graph displays the load current, and the lower graph displays the instantaneous and average power dissipated by the load.

Example S.12.1 *continued*

Some of the resulting waveforms are displayed in Fig. B.55. The upper and middle graphs show the load voltage and current, respectively. The peak voltage amplitude is 16.9 V, and the peak current amplitude is 2.1 A. If one looks carefully, one can observe that both exhibit crossover distortion. The bottom graph displays the instantaneous and the average power dissipated in the load resistance as computed by multiplying the voltage and current values to obtain the instantaneous power, and taking a running average for the average load power P_L. The transient behavior of the average load power, which eventually settles into a quasi-constant steady state of about 17.6 W, is an artifact of the SPICE algorithm used to compute the running average of a waveform.

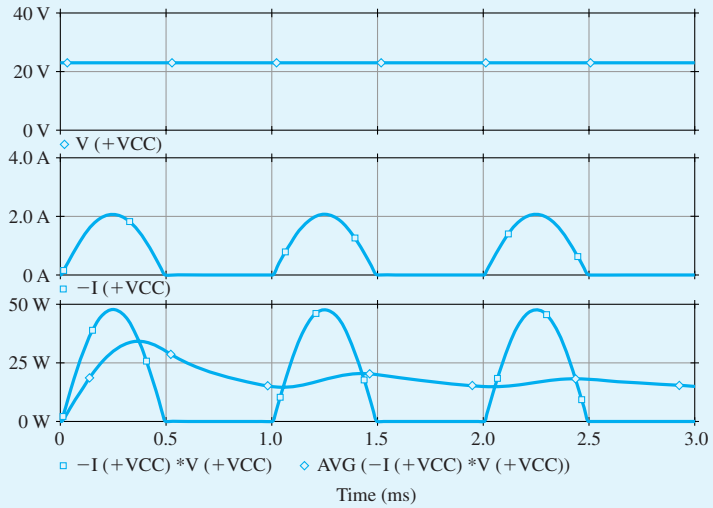


Figure B.56 The voltage (upper graph), current (middle graph), and instantaneous and average power (bottom graph) supplied by the positive voltage supply ($+V_{CC}$) in the circuit of Fig. B.54.

The upper two graphs of Fig. B.56 show the voltage and current waveforms, respectively, of the positive supply, $+V_{CC}$. The bottom graph shows the instantaneous and average power supplied by $+V_{CC}$. Similar waveforms can be plotted for the negative supply, $-V_{CC}$. The average power provided by each supply is found to be about 15 W, for a total supply power P_S of 30 W. Thus, the power-conversion efficiency can be computed to be

$$\eta = P_L/P_S = \frac{17.6}{30} \times 100\% = 58.6\%$$

Figure B.57 shows plots of the voltage, current, and power waveforms associated with transistor Q_P. Similar waveforms can be obtained for Q_N. As expected, the voltage waveform is a sinusoid, and the current waveform consists of half-sinusoids. The waveform of the instantaneous power, however, is rather unusual. It indicates the presence of some distortion as a result of driving the transistors rather hard. This can be verified by reducing the amplitude of the input signal. Specifically, when the amplitude is reduced to about 17 V, the "dip" in the power waveform vanishes. The average power dissipated in each of Q_N and Q_P are found to be approximately 6 W.

Observe that the results obtained using SPICE are quite close to those obtained by hand calculation in Example 12.3.

To investigate the crossover distortion further, we present in Fig. B.58 a plot of the voltage transfer characteristic (VTC) of the class B output stage. This plot is obtained through a dc-analysis simulation with v_{IN} swept over the range –10 V to +10 V in 1.0-mV increments. From it, we determine that the slope of the VTC is nearly unity and that the dead band extends from –0.60 V to +0.58 V. The effect of the crossover distortion can be quantified by performing a Fourier analysis on the output voltage waveform in SPICE. This analysis decomposes the waveform generated through a transient analysis into its

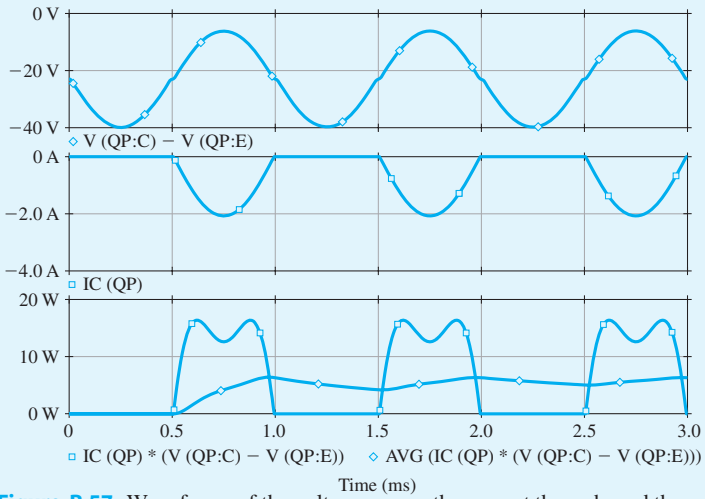


Figure B.57 Waveforms of the voltage across, the current through, and the power dissipated in the *pnp* transistor Q_P of the output stage shown in Fig. B.54.

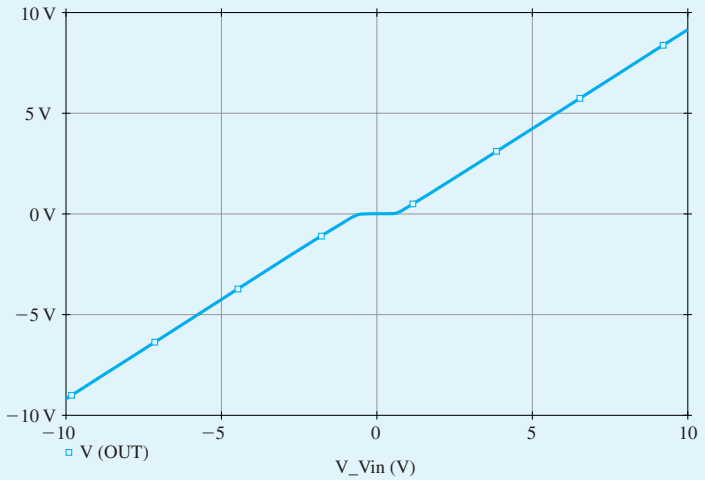


Figure B.58 Transfer characteristic of the class B output stage of Fig. B.54.

Example S.12.1 *continued*

Fourier-series components. Further, SPICE computes the total harmonic distortion (THD) of the output waveform. The results obtained from the simulation output file are shown on the next page.

These Fourier components are used to plot the line spectrum shown in Fig. B.59. We note that the output waveform is rather rich in odd harmonics and that the resulting THD is rather high (2.14%).

```
FOURIER COMPONENTS OF TRANSIENT RESPONSE V(OUT)

DC COMPONENT = -1.525229E-02
```

HARMONIC NO	FREQUENCY (HZ)	FOURIER COMPONENT	NORMALIZED COMPONENT	PHASE (DEG)	NORMALIZED PHASE (DEG)
1	1.000E+03	1.674E+01	1.000E+00	-2.292E-03	0.000E+00
2	2.000E+03	9.088E-03	5.428E-04	9.044E+01	9.044E+01
3	3.000E+03	2.747E-01	1.641E-02	-1.799E+02	-1.799E+02
4	4.000E+03	4.074E-03	2.433E-04	9.035E+01	9.036E+01
5	5.000E+03	1.739E-01	1.039E-02	-1.799E+02	-1.799E+02
6	6.000E+03	5.833E-04	3.484E-05	9.159E+01	9.161E+01
7	7.000E+03	1.195E-01	7.140E-03	-1.800E+02	-1.799E+02
8	8.000E+03	5.750E-04	3.435E-05	9.128E+01	9.129E+01
9	9.000E+03	9.090E-02	5.429E-03	-1.800E+02	-1.799E+02
10	1.000E+04	3.243E-04	1.937E-05	9.120E+01	9.122E+01

```
TOTAL HARMONIC DISTORTION = 2.140017E+00 PERCENT
```

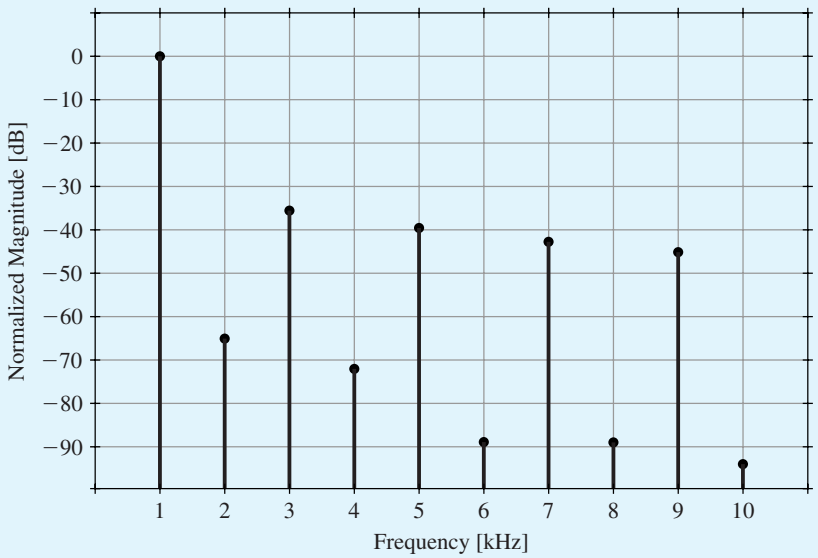


Figure B.59 Fourier-series components of the output waveform of the class B output stage in Fig. B.54.

Example S.13.1

Frequency Compensation of the Two-Stage CMOS Op Amp

In this example, we will use SPICE to aid in designing the frequency compensation of the two-stage CMOS circuit whose capture schematic is shown in Fig. B.60. SPICE will then be employed to determine the frequency response and the slew rate of the op amp. We will assume a 0.5-μm *n*-well CMOS technology for the MOSFETs and will use the SPICE level-1 model parameters listed in Table B.3. Observe that to eliminate the body effect and improve the matching between M_1 and M_2, the source terminals of the input PMOS transistors M_1 and M_2 are connected to their *n* well.

The op-amp circuit in Fig. B.60 is designed using a reference current $I_{REF} = 90\,\mu A$, a supply voltage $V_{DD} = 3.3$ V, and a load capacitor $C_L = 1$ pF. Unit-size transistors with $W/L = 1.25\,\mu m/0.6\,\mu m$ are used for both the NMOS and PMOS devices. The transistors are sized for an overdrive voltage $V_{OV} = 0.3$V. The corresponding multiplicative factors are given in Fig. B.60.

In SPICE, the common-mode input voltage V_{CM} of the op-amp circuit is set to $V_{DD}/2 = 1.65$ V. A bias-point simulation is performed to determine the dc operating point. Using the values found in the simulation output file for the small-signal parameters of the MOSFETs, we obtain[14]

$$G_{m1} = 0.333\,\text{mA/V}$$
$$G_{m2} = 0.650\,\text{mA/V}$$
$$C_1 = 26.5\,\text{fF}$$
$$C_2 = 1.04\,\text{pF}$$

using Eqs. (13.7), (13.12), (13.20), and (13.21), respectively. Then, using Eq. (13.30), the frequency of the second, nondominant, pole can be found as

$$f_{P2} \simeq \frac{G_{m2}}{2\pi C_2} = 97.2\,\text{MHz}$$

In order to place the transmission zero, given by Eq. (13.39), at infinite frequency, we select

$$R = \frac{1}{G_{m2}} = 1.53\,\text{k}\Omega$$

Now, using Eq. (13.37), the phase margin of the op amp can be expressed as

$$PM = 90° - \tan^{-1}\left(\frac{f_t}{f_{P2}}\right) \tag{B.44}$$

where f_t is the unity-gain frequency, given in Eq. (13.31),

$$f_t = \frac{G_{m1}}{2\pi C_C} \tag{B.45}$$

Using Eqs. (B.44) and (B.45), we determine that compensation capacitors of $C_C = 0.78$ pF and $C_C = 2$ pF are required to achieve phase margins of $PM = 55°$ and $PM = 75°$, respectively.

[14] Recall that G_{m1} and G_{m2} are the transconductances of, respectively, the first and second stages of the op amp. Capacitors C_1 and C_2 represent the total capacitance to ground at the output nodes of, respectively, the first and second stages of the op amp.

Example S.13.1 *continued*

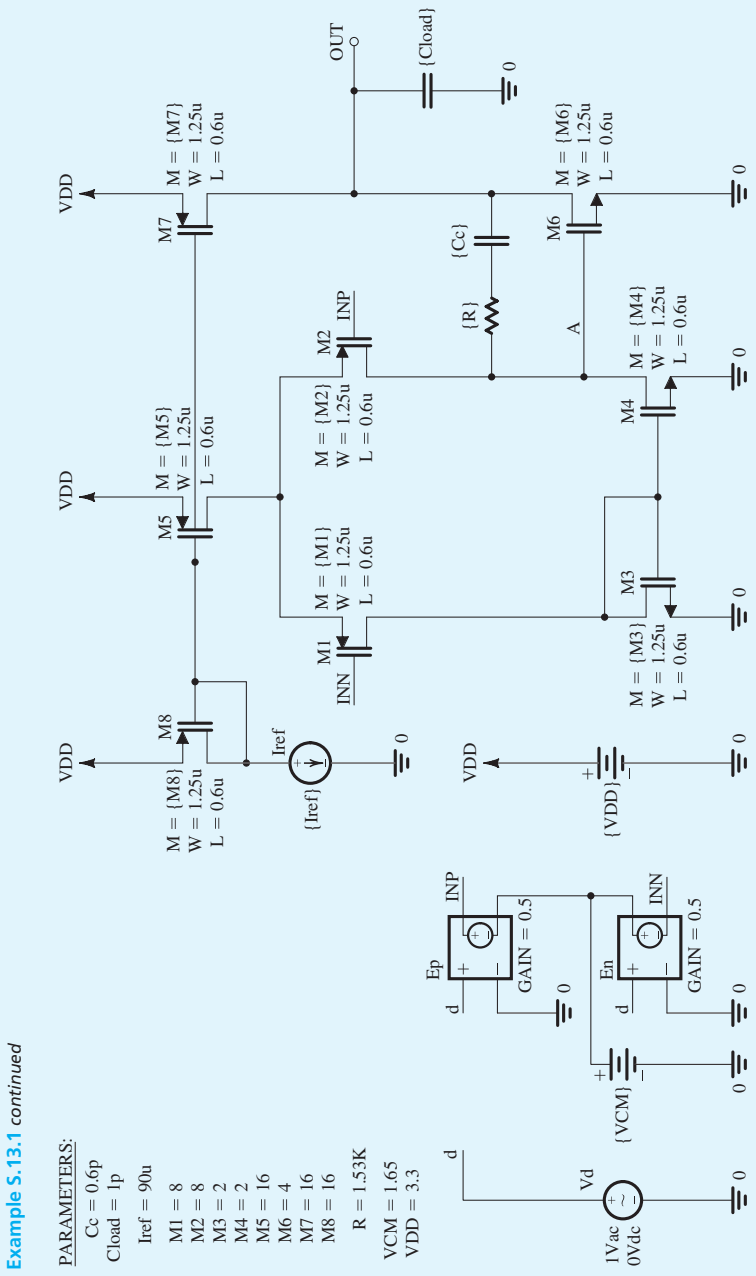


Figure B.60 Schematic capture of the two-stage CMOS op amp in Example S.13.1

Next, an ac-analysis simulation is performed in SPICE to compute the frequency response of the op amp and to verify the foregoing design values. It was found that with $R = 1.53\,\text{k}\Omega$, we needed $C_C = 0.6$ pF and $C_C = 1.8$ pF to set $PM = 55°$ and $PM = 75°$, respectively. We note that these values are reasonably close to those predicted by hand analysis. The corresponding frequency responses for the compensated op amp are plotted in Figs. B.61 and B.62. For comparison, we also show the frequency response of the uncompensated op amp ($C_C = 0$). Observe that the unity gain frequency f_t drops from 70.2 MHz to 26.4 MHz as C_C is increased to improve PM (as anticipated from Eq. B.45).

Rather than increasing the compensation capacitor C_C, the value of the series resistor R can be increased to improve the phase margin PM: For a given C_C, increasing R above $1/G_{m2}$ places the transmission zero at a negative real-axis location (Eq. 13.39), where the phase it introduces *adds* to the phase margin. Thus, PM can be improved without affecting f_t. To verify this point, we set C_C to 0.6 pF and simulate the op-amp circuit in SPICE for the cases of $R = 1.53\,\text{k}\Omega$ and $R = 3.2\,\text{k}\Omega$. The corresponding frequency response is plotted in Fig. B.63. Observe how f_t is approximately independent of R. However, by increasing R, PM is improved from 55° to 75°.

Increasing the PM is desirable because it reduces the overshoot in the step response of the op amp. To verify this point, we simulate in SPICE the step response of the op amp for $PM = 55°$ and $PM = 75°$. To do that, we connect the op amp in a unity-gain configuration, apply a small (10-mV) pulse signal at the input with very short (1-ps) rise and fall times to emulate a step input, perform a transient-analysis simulation, and plot the output voltage as shown in Fig. B.64. Observe that the overshoot in the step response drops from 15% to 1.4% when the phase margin is increased from 55° to 75°.

We conclude this example by computing SR, the slew rate of the op amp. From Eq. (13.41),

$$SR = 2\pi f_t V_{OV} = \frac{G_{m1}}{C_C} V_{OV} = 166.5\,\text{V}/\mu\text{s}$$

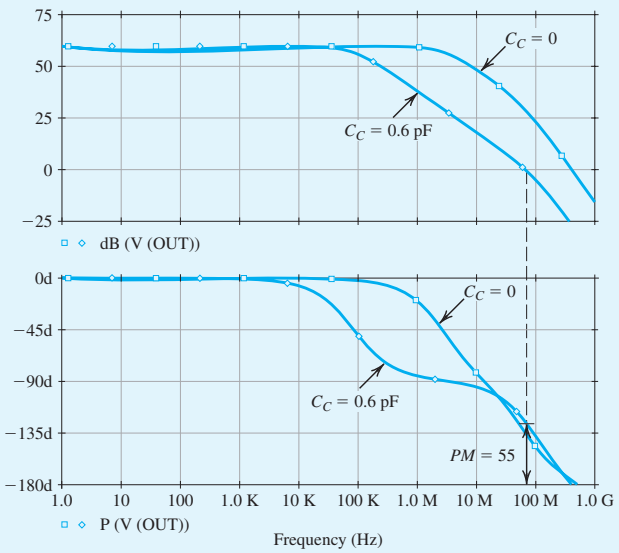


Figure B.61 Magnitude and phase response of the op-amp circuit in Fig. B.43: $R = 1.53\,\text{k}\Omega$, $C_C = 0$ (no frequency compensation), and $C_C = 0.6$ pF ($PM = 55°$).

Example S.13.1 *continued*

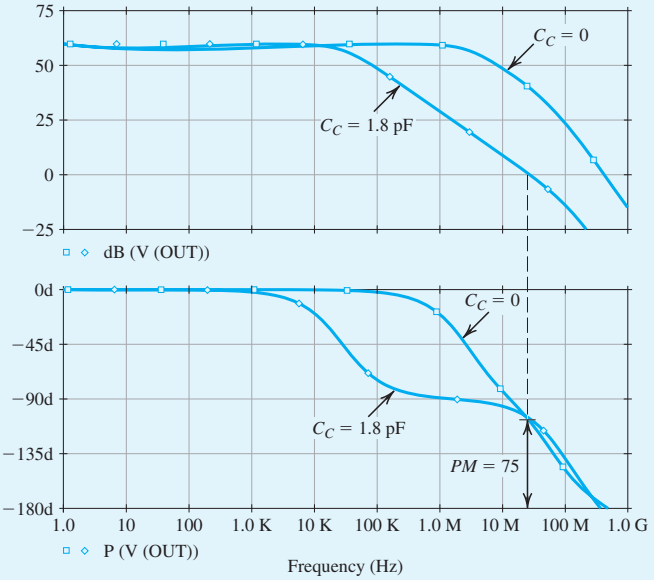


Figure B.62 Magnitude and phase response of the op-amp circuit in Fig. B.60: $R = 1.53$ kΩ, $C_C = 0$ (no frequency compensation), and $C_C = 1.8$ pF ($PM = 75°$).

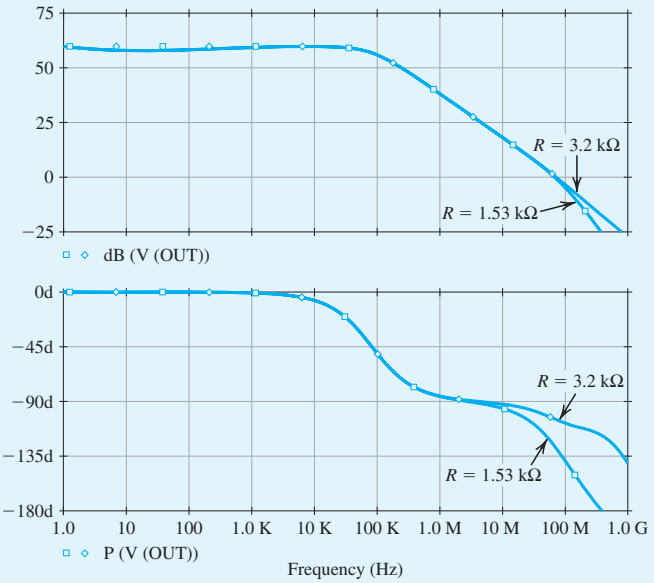


Figure B.63 Magnitude and phase response of the op-amp circuit in Fig. B.60: $C_C = 0.6$ pF, $R = 1.53$ kΩ ($PM = 55°$), and $R = 3.2$ kΩ ($PM = 75°$).

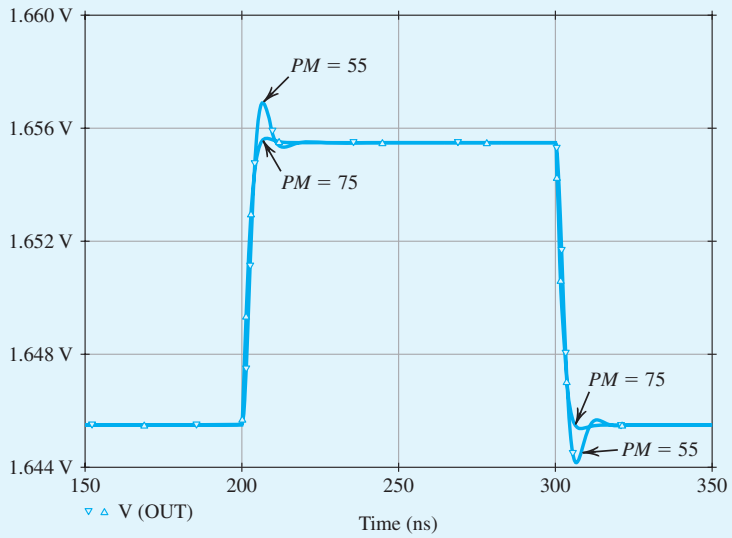


Figure B.64 Small-signal step response (for a 10-mV step input) of the op-amp circuit in Fig. B.60 connected in a unity-gain configuration: $PM = 55^\circ$ ($C_C = 0.6$ pF, $R = 1.53$ kΩ) and $PM = 75^\circ$ ($C_C = 0.6$ pF, $R = 3.2$ kΩ).

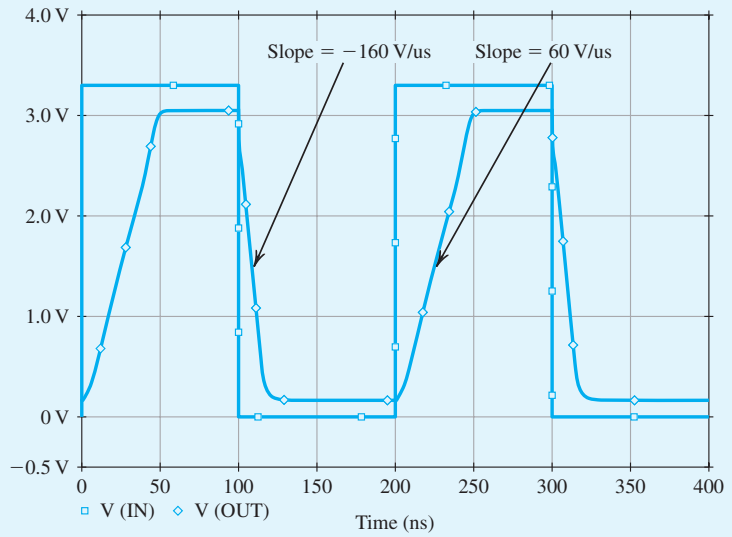


Figure B.65 Large-signal step response (for a 3.3-V step-input) of the op-amp circuit in Fig. B.60 connected in a unity-gain configuration. The slope of the rising and falling edges of the output waveform correspond to the slew rate of the op amp.

Example S.13.1 *continued*

when $C_C = 0.6$ pF. Next, to determine SR using SPICE (see Example S.2.2), we again connect the op amp in a unity-gain configuration and perform a transient-analysis simulation. However, we now apply a large pulse signal (3.3 V) at the input to cause slew-rate limiting at the output. The corresponding output-voltage waveform is plotted in Fig. B.65. The slope of the slew-rate-limited output waveform corresponds to the slew rate of the op amp and is found to be $SR = 160$ V/ms and 60 V/ms for the negative- and positive-going output, respectively. These results, with the unequal values of SR in the two directions, differ from those predicted by the simple model for the slew-rate limiting of the two-stage op-amp circuit (Section 13.1.6). The difference can perhaps be said to be a result of transistor M_4 entering the triode region and its output current (which is sourced through C_C) being correspondingly reduced. Of course, the availability of SPICE should enable the reader to explore this point further.

Example S.14.1

Verification of the Design of a Fifth-Order Chebyshev Filter

In this example we show how SPICE can be utilized to verify the design of a fifth-order Chebyshev filter. Specifically, we simulate the operation of the circuit whose component values are selected to provide the Chebyshev response in Exercise 14.4. The complete circuit is shown in Fig. B.66(a). It consists of a cascade of two second-order simulated-LCR resonators using the Antoniou circuit and a first-order op amp–RC circuit. Using SPICE, we would like to compare the magnitude of the filter response with that computed directly from its transfer function. Here, we note that SPICE can also be used to perform the latter task by using the Laplace transfer-function block in the analog-behavioral-modeling (ABM) library.

Since the purpose of the simulation is simply to verify the design, we assume ideal components. For the op amps, we utilize a near-ideal model, namely, a voltage-controlled voltage source (VCVS) with a gain of 10^6 V/V, as shown in Fig. B.66(b).[16]

In SPICE, we apply a 1-V ac signal at the filter input, perform an ac-analysis simulation over the range 1 Hz to 20 kHz, and plot the output voltage magnitude versus frequency, as shown in Fig. B.67. Both an expanded view of the passband and a view of the entire magnitude response are shown. These results are almost identical to those computed directly from the ideal transfer function, thereby verifying the correctness of the design.

[16] SPICE models for the op amp are described in Section B.1.1.

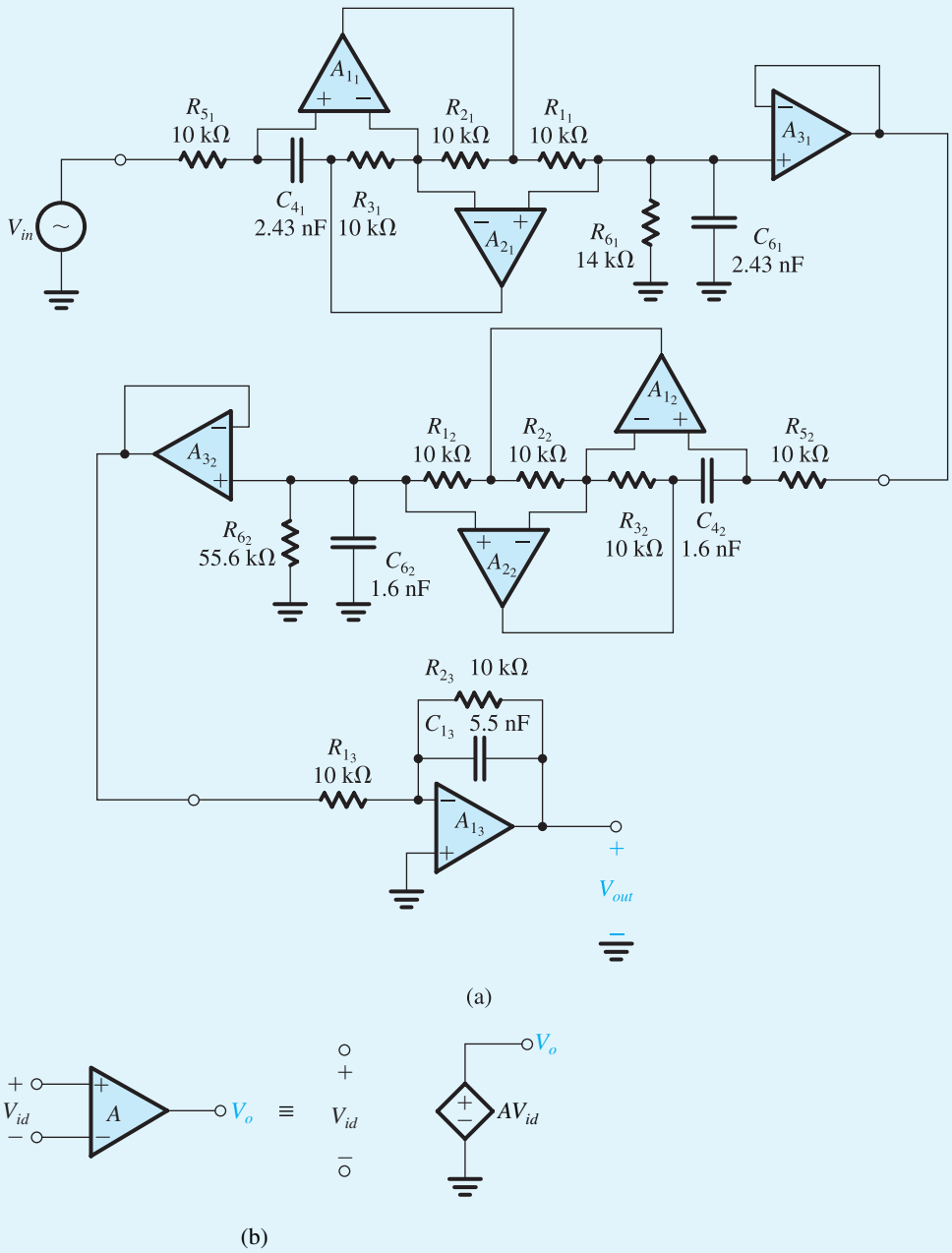
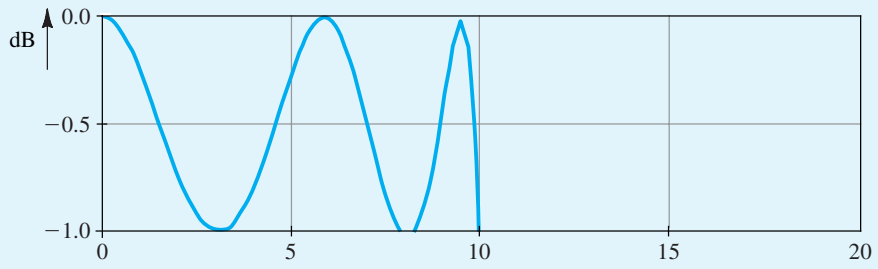
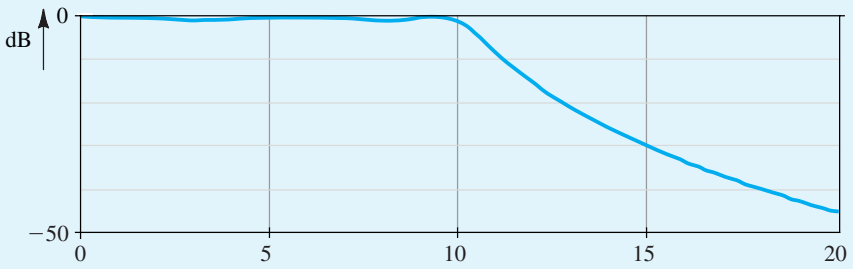


Figure B.66 Circuits for Example S.14.1. **(a)** Fifth-order Chebyshev filter circuit implemented as a cascade of two second-order simulated LCR resonator circuits and a single first-order op amp–RC circuit. **(b)** VCVS representation of an ideal op amp with gain A.

Example S.14.1 *continued*



(a)



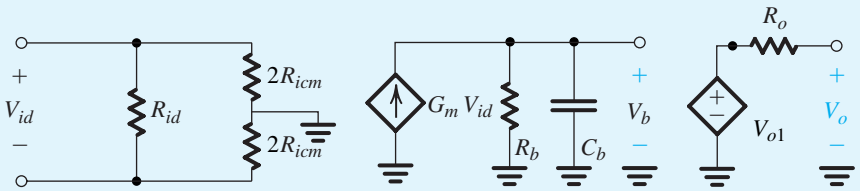
(b)

Figure B.67 Magnitude response of the fifth-order lowpass filter circuit shown in Fig. B.66: **(a)** an expanded view of the passband region; **(b)** a view of both the passband and stopband regions.

Example S.14.2

Effect of Finite Op-Amp Bandwidth on the Operation of the Two-Integrator-Loop Filter

In this example, we investigate the effect of the finite bandwidth of practical op amps on the response of a two-integrator-loop bandpass filter utilizing the Tow–Thomas biquad circuit of Fig. 14.25(b). The circuit is designed to provide a bandpass response with $f_0 = 10\,\text{kHz}$, $Q = 20$, and a unity center-frequency gain. The op amps are assumed to be of the 741 type. Specifically, we model the terminal behavior of the op amp with the single-time-constant linear network shown in Fig. B.68. Since the analysis performed here is a small-signal (ac) analysis that ignores nonlinearities, no nonlinearities are included in this op-amp



$$A_0 = G_m R_1 \qquad \omega_b = 1/R_b C_b$$

Figure B.68 One-pole equivalent-circuit macromodel of an op amp operated within its linear region.

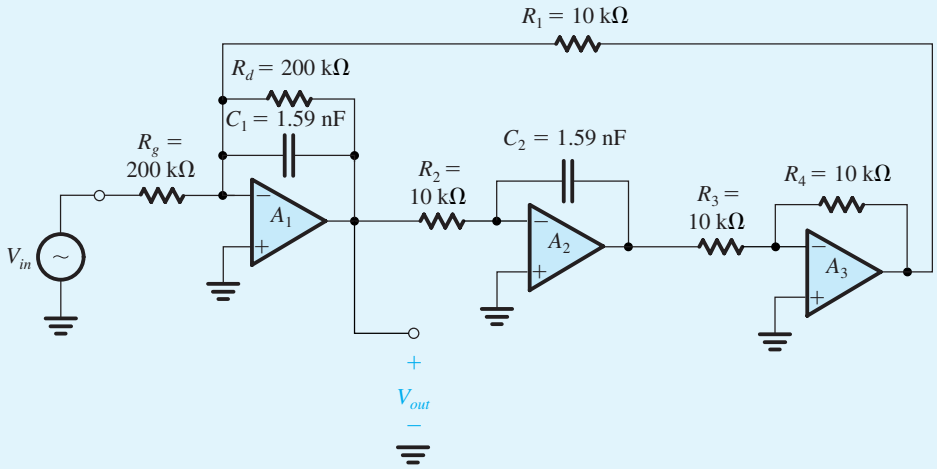


Figure B.69 Circuit for Example S.14.2 Second-order bandpass filter implemented with a Tow–Thomas biquad circuit having $f_0 = 10\,\text{kHz}$, $Q = 20$, and unity center-frequency gain.

macromodel. (If the effects of op-amp nonlinearities are to be investigated, a transient analysis should be performed.) The following values are used for the parameters of the op-amp macromodel in Fig. B.68:

$$R_{id} = 2\ \text{M}\Omega \qquad R_{icm} = 500\ \text{M}\Omega \qquad R_1 = 75\ \Omega$$
$$G_m = 0.19\ \text{mA/V} \qquad R_b = 1.323 \times 10^9\ \Omega \qquad C_b = 30\ \text{pF}$$

These values result in the specified input and output resistances of the 741-type op amp. Further, they provide a dc gain $A_0 = 2.52 \times 10^5$ V/V and a 3-dB frequency f_b of 4 Hz, again equal to the values specified for the 741. Note that the selection of the individual values of G_m, R_b, and C_b is immaterial as long as $G_m R_b = A_0$ and $C_b R_b = 1/2\pi f_b$.

The Tow-Thomas circuit simulated is shown in Fig. B.69. The circuit is simulated in SPICE for two cases: (1) assuming 741-type op amps and using the linear macromodel in Fig. B.68; and (2) assuming ideal op amps with dc gain of $A_0 = 10^6$ V/V and using the near-ideal model in Fig. B.66(b). In both cases,

Example S.14.2 *continued*

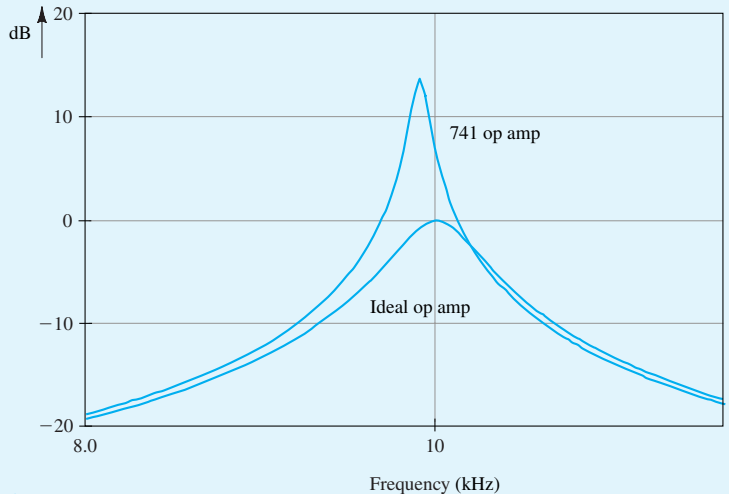
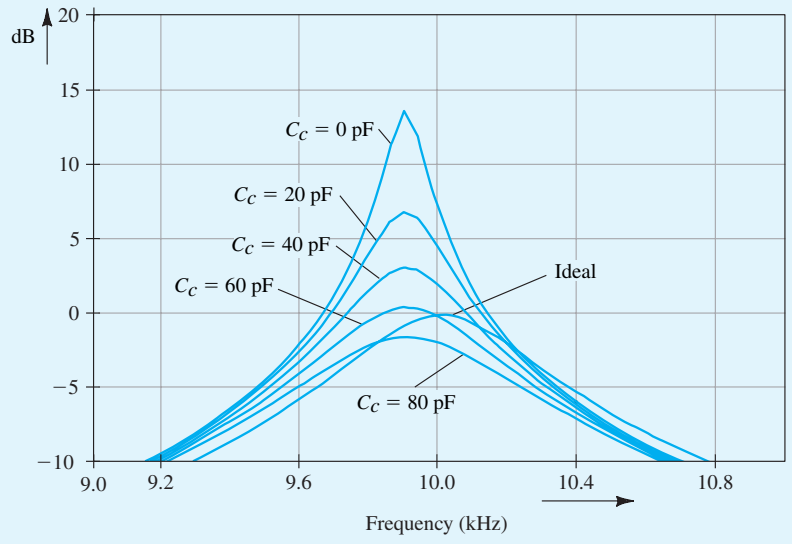


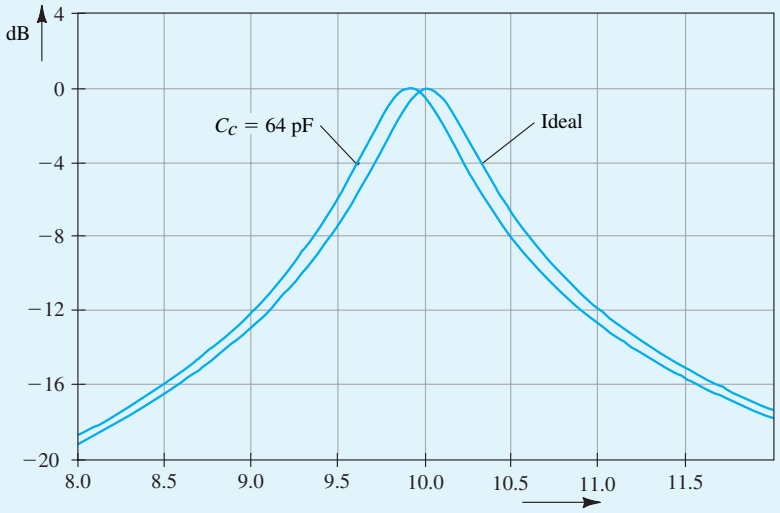
Figure B.70 Comparing the magnitude response of the Tow–Thomas biquad circuit (shown in Fig. B.69) constructed with 741-type op amps, with the ideal magnitude response. These results illustrate the effect of the finite dc gain and bandwidth of the 741 op amp on the frequency response of the Tow–Thomas biquad circuit.

we apply a 1-V ac signal at the filter input, perform an ac-analysis simulation over the range 8 kHz to 12 kHz, and plot the output-voltage magnitude versus frequency.

The simulation results are shown in Fig. B.70, from which we observe the significant deviation between the response of the filter using the 741 op amp and that using the near-ideal op-amp model. Specifically, the response with practical op amps shows a deviation in the center frequency of about –100 Hz, and a reduction in the 3-dB bandwidth from 500 Hz to about 110 Hz. Thus, in effect, the filter Q factor has increased from the ideal value of 20 to about 90. This phenomenon, known as *Q-enhancement,* is predictable from an analysis of the two-integrator-loop biquad with the finite op-amp bandwidth taken into account [see Sedra and Brackett (1978)]. Such an analysis shows that Q-enhancement occurs as a result of the excess phase lag introduced by the finite op-amp bandwidth. The theory also shows that the Q-enhancement effect can be compensated for by introducing phase lead around the feedback loop. This can be accomplished by connecting a small capacitor, C_C, across resistor R_2. To investigate the potential of such a compensation technique, we repeat the SPICE simulation with various capacitance values. The results are displayed in Fig. B.71(a). We observe that as the compensation capacitance is increased from 0 pF, both the filter Q and the resonance peak of the filter response move closer to the desired values. It is evident, however, that a compensation capacitance of 80 pF causes the response to deviate further from the ideal. Thus, optimum compensation is obtained with a capacitance value between 60 pF and 80 pF. Further experimentation using SPICE enabled us to determine that such an optimum is obtained with a compensation capacitance of 64 pF. The corresponding response is shown, together with the ideal response, in Fig. B.71(b). We note that although the filter Q has been restored to its ideal value, there remains a deviation in the center frequency. We shall not pursue this matter any further here; our objective is not to present a detailed study of the design of two-integrator-loop biquads; rather, it is to illustrate the application of SPICE in investigating the nonideal performance of active-filter circuits, generally.



(a)



(b)

Figure B.71 **(a)** Magnitude response of the Tow–Thomas biquad circuit with different values of compensation capacitance. For comparison, the ideal response is also shown. **(b)** Comparing the magnitude response of the Tow–Thomas biquad circuit using a 64-pF compensation capacitor and the ideal response.

Example S.15.1

Wien-Bridge Oscillator

For our first example on oscillators, we shall simulate the operation of the Wien-bridge oscillator whose schematic capture is shown in Fig. B.72. The component values are selected to yield oscillations at 1 kHz. We would like to investigate the operation of the circuit for different settings of R_{1a} and R_{1b}, with $R_{1a} + R_{1b} = 50\,\text{k}\Omega$. Since oscillation just starts when $(R_2 + R_{1b})/R_{1a} = 2$ (see Exercise 17.4), that is, when $R_{1a} = 20\,\text{k}\Omega$ and $R_{1b} = 30\,\text{k}\Omega$, we consider three possible settings: (a) $R_{1a} = 15\,\text{k}\Omega$, $R_{1b} = 35\,\text{k}\Omega$; (b) $R_{1a} = 18\,\text{k}\Omega$, $R_{1b} = 32\,\text{k}\Omega$; and (c) $R_{1a} = 25\,\text{k}\Omega$, $R_{1b} = 25\,\text{k}\Omega$. These settings correspond to loop gains of 1.33, 1.1, and 0.8, respectively.

In SPICE, a 741-type op amp and 1N4148-type diodes are used to simulate the circuit similar to the one in Fig. 15.8. A transient-analysis simulation is performed with the capacitor voltages initially set to zero. This demonstrates that the op-amp offset voltage is sufficient to cause the oscillations to start without the need for special start-up circuitry. Figure B.73 shows the simulation results. The graph in Fig. B.73(a)

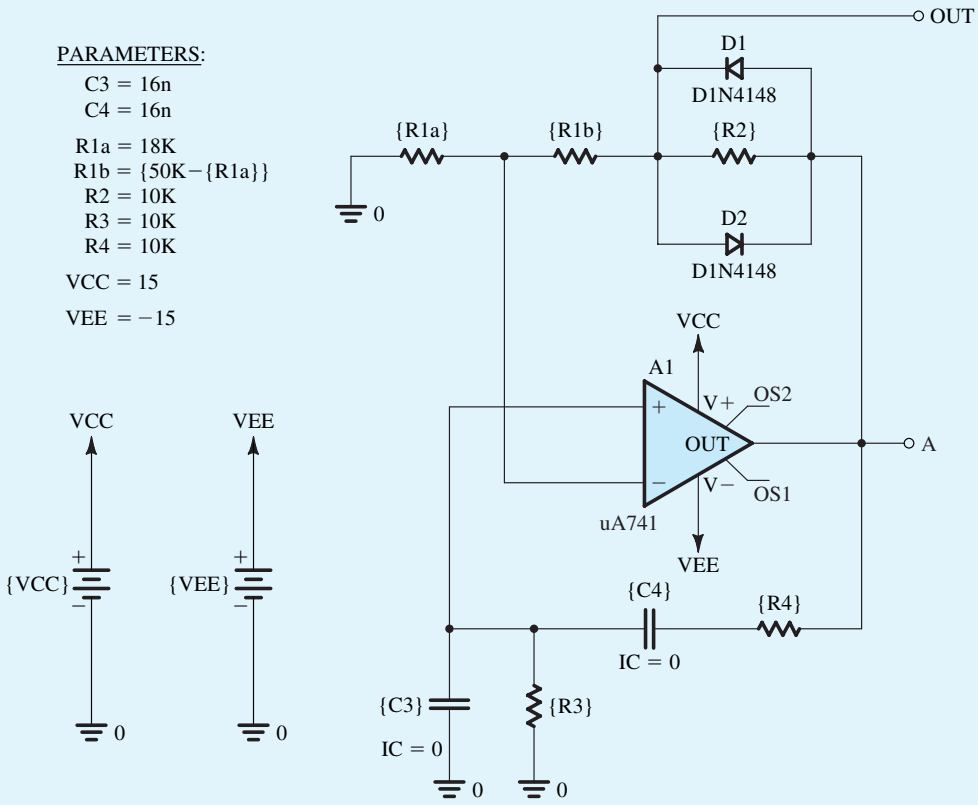
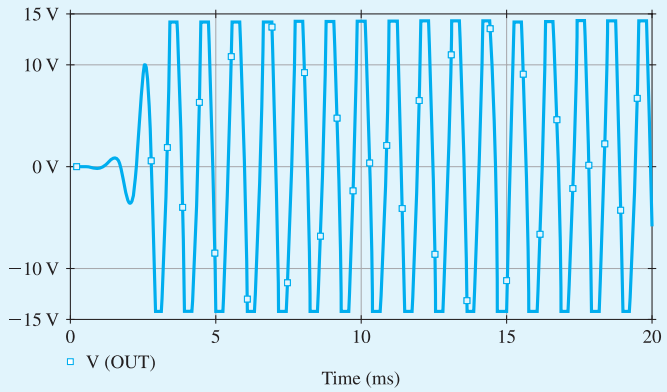
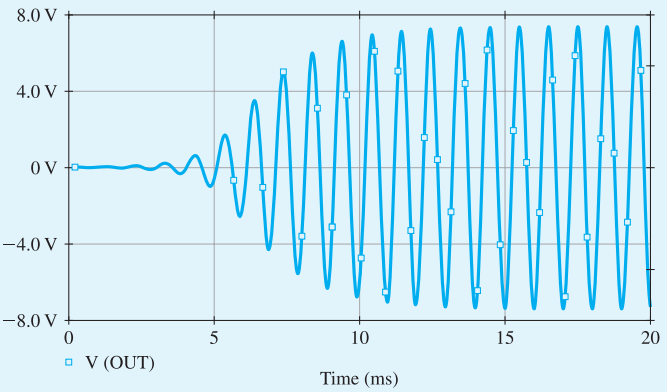


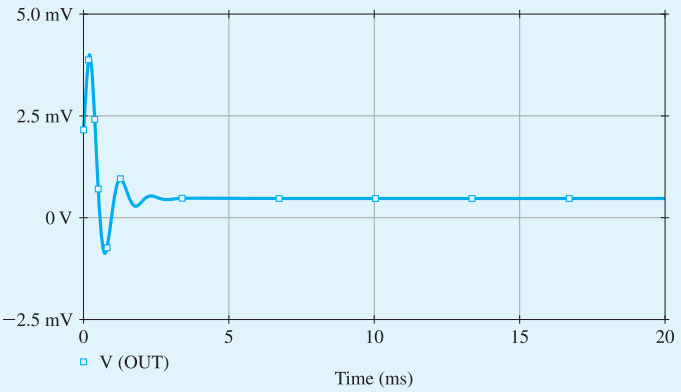
Figure B.72 Example S.15.1: Schematic capture of a Wien-bridge oscillator.



(a) R_{1a} = 15 kΩ, Loop Gain = 1.33



(b) R_{1a} = 18 kΩ, Loop Gain = 1.1



(c) R_{1a} = 25 kΩ, Loop Gain = 0.8

Figure B.73 Start-up transient behavior of the Wien-bridge oscillator shown in Fig. B.72 for various values of loop gain.

Example S.15.1 *continued*

shows the output waveform obtained for a loop gain of 1.33 V/V. Observe that although the oscillations grow and stabilize rapidly, the distortion is considerable. The output obtained for a loop gain of 1.1, shown in Fig. B.73(b), is much less distorted. However, as expected, as the loop gain is reduced toward unity, it takes longer for the oscillations to build up and for the amplitude to stabilize. For this case, the frequency is 986.6 Hz, which is reasonably close to the design value of 1 kHz, and the amplitude is 7.37 V. Finally, for a loop gain of 0.8, the output shown in Fig. B.73(c) confirms our expectation that sustained oscillations cannot be obtained when the loop gain is less than unity.

SPICE can be used to investigate the spectral purity of the output sine wave. This is achieved using the Fourier analysis facility. It is found that in the steady state, the output for the case of a loop gain of 1.1 has a THD figure of 1.88%. When the oscillator output is taken at the op-amp output (voltage v_A), a THD of 2.57% is obtained, which, as expected, is higher than that for the voltage v_{OUT}, but not by very much. The output terminal of the op amp is of course a much more convenient place to take the output.

Example S.15.2

Active-Filter-Tuned Oscillator

In this example, we use SPICE to verify our contention that a superior op amp–oscillator can be realized using the active-filter-tuned circuit of Fig. 15.13. We also investigate the effect of changing the value of the filter Q factor on the spectral purity of the output sine wave.

Consider the circuit whose schematic capture is shown in Fig. B.74. For this circuit, the center frequency is 1 kHz, and the filter Q is 5 when $R_1 = 50$ kΩ and 20 when $R_1 = 200$ kΩ. As in the case of the Wien-bridge circuit in Example S.15.1, 741-type op amps and 1N4148-type diodes are utilized. In SPICE, a transient-analysis simulation is performed with the capacitor voltages initially set to zero. To be able to compute the Fourier components of the output, the analysis interval chosen must be long enough to allow the oscillator to reach a steady state. The time to reach a steady state is in turn determined by the value of the filter Q; the higher the Q, the longer it takes the output to settle. For $Q = 5$, it was determined, through a combination of approximate calculations and experimentation using SPICE, that 50 ms is a reasonable estimate for the analysis interval. For plotting purposes, we use 200 points per period of oscillation.

The results of the transient analysis are plotted in Fig. B.75. The upper graph shows the sinusoidal waveform at the output of op amp A_1 (voltage v_1). The lower graph shows the waveform across the diode limiter (voltage v_2). The frequency of oscillation is found to be very close to the design value of 1 kHz. The amplitude of the sine wave is determined to be 1.15 V (or 2.3 V p-p). Note that this is lower than the 3.6 V estimated in Exercise 15.9. The latter value, however, was based on an estimate of 0.7-V drop across each conducting diode in the limiter. The lower waveform in Fig. B.75 indicates that the diode drop is closer to 0.5 V for a 1 V peak-to-peak amplitude of the pseudo-square wave. We should therefore expect the peak-to-peak amplitude of the output sinusoid to be lower than 3.6 V by the same factor, and indeed it is approximately the case.

In SPICE, the Fourier analysis of the output sine wave indicates that THD = 1.61%. Repeating the simulation with Q increased to 20 (by increasing R_1 to 200 kΩ), we find that the value of THD is reduced to 1.01%. Thus, our expectations that the value of the filter Q can be used as an effective means for constrolling the THD of the output waveform are confirmed.

Example S.15.2 *continued*

PARAMETERS:
C1 = 16n
C2 = 16n
R1 = 200K
R2 = 10K
R3 = 10K
R4 = 10K
R5 = 10K
R6 = 10K
VCC = 15
VEE = −15

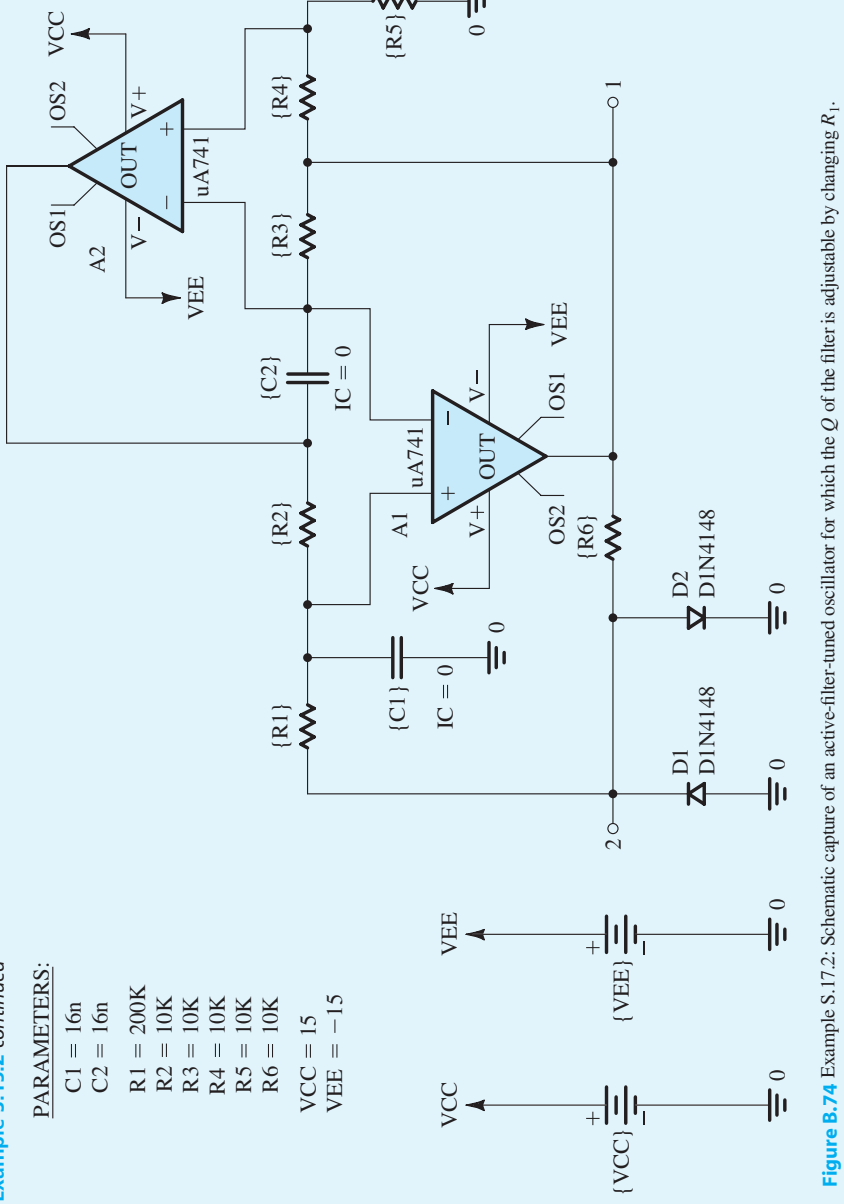


Figure B.74 Example S.17.2: Schematic capture of an active-filter-tuned oscillator for which the Q of the filter is adjustable by changing R_1.

Example S.15.2 *continued*

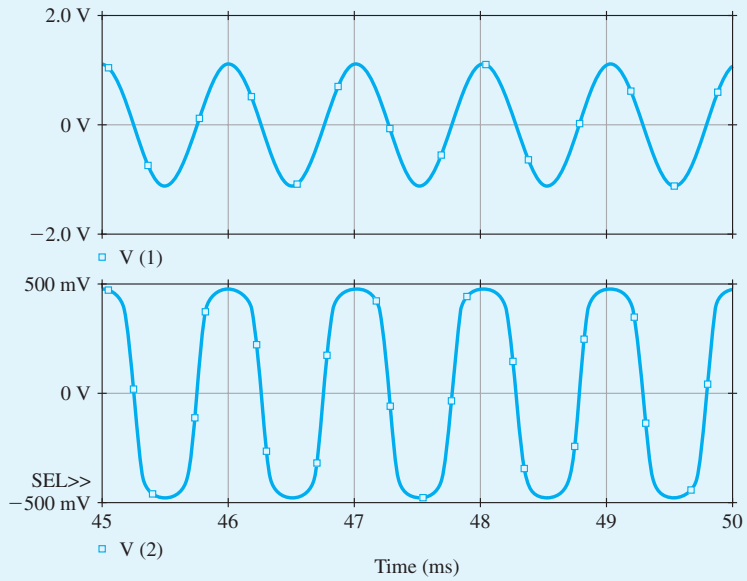


Figure B.75 Output waveforms of the active-filter-tuned oscillator shown in Fig. B.74 for $Q = 5$ ($R_1 = 50$ kΩ).

Example S.16.1

Operation of the CMOS Inverter

In this example, we will use SPICE to simulate the CMOS inverter whose schematic capture is shown in Fig, B.76. We will assume a 0.5-μm CMOS technology for the MOSFETs and use parts NMOS0P5 and PMOS0P5 whose level-1 model parameters are listed in Table B.3. In addition to the channel length L and the channel width W, we have used the multiplicative factor m to specify the dimensions of the MOSFETs. The MOSFET parameter m, whose default value is 1, is used in SPICE to specify the number of unit-size MOSFETs connected in parallel (see Fig. B.28). In our simulations, we will use unit-size transistors with $L = 0.5\,\mu$m and $W = 1.25\,\mu$m. We will simulate the inverter for two cases: (a) setting $m_p/m_n = 1$ so that the NMOS and PMOS transistors have equal widths, and (b) setting $m_p/m_n = \mu_n/\mu_p = 4$ so that the PMOS transistor is four times wider than the NMOS transistor (to compensate for the lower mobility in p-channel devices as compared with n-channel ones). Here, m_n and m_p are the multiplicative factors of, respectively, the NMOS and PMOS transistors of the inverter.

To compute both the voltage transfer characteristic (VTC) of the inverter and its supply current at various values of the input voltage V_{in}, we apply a dc voltage source at the input and perform a dc analysis

Example S.16.1 *continued*

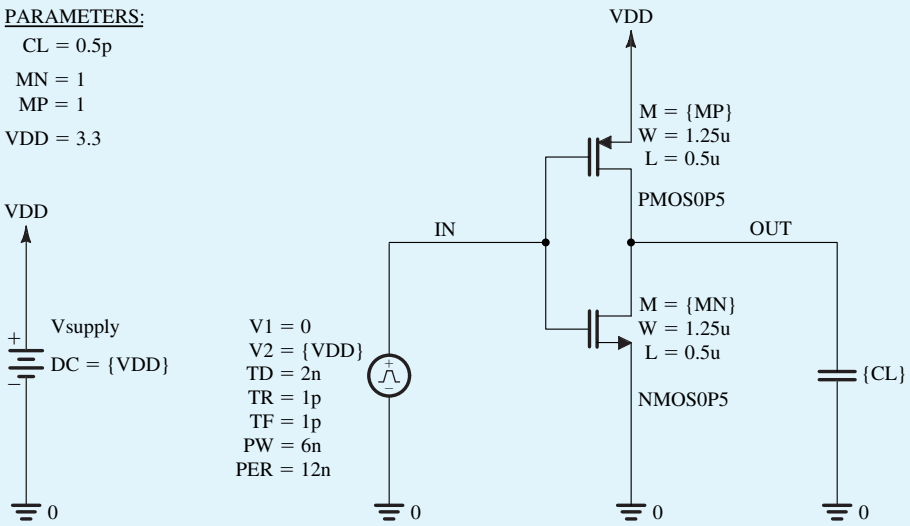


Figure B.76 Schematic capture of the CMOS inverter in Example S.16.1.

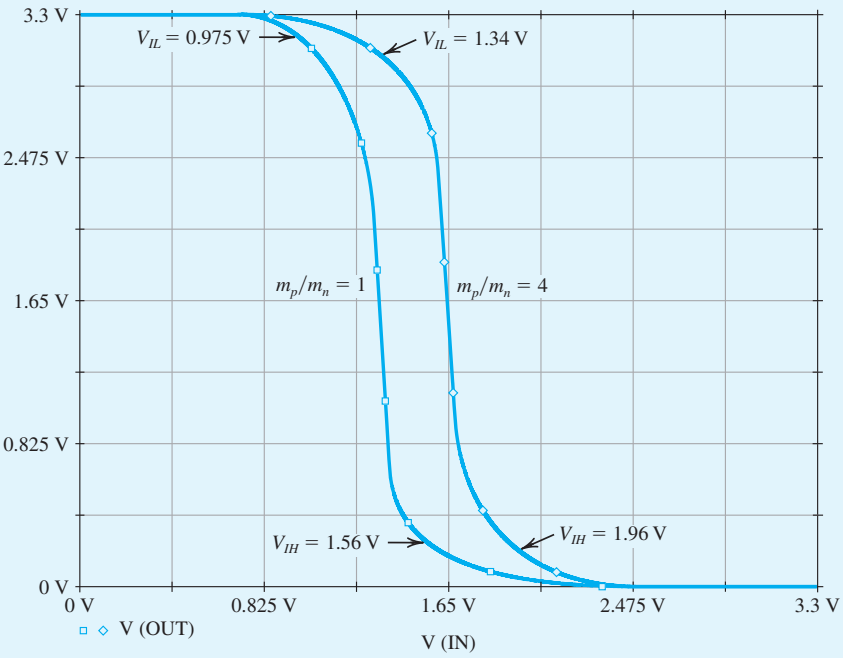


Figure B.77 Input–output voltage transfer characteristic (VTC) of the CMOS inverter in Example S.16.1 with $m_p/m_n = 1$ and $m_p/m_n = 4$.

Example S.16.1 *continued*

with V_{in} swept over the range of 0 to V_{DD}. The resulting VTC is plotted in Fig. B.77. Note that the slope of the VTC in the switching region (where both the NMOS and PMOS devices are in saturation) is not infinite as predicted from the simple theory presented in Chapter 16 (Section 16.3.2, Fig. 16.25). Rather, the nonzero value of λ causes the inverter gain to be finite. Using the derivative feature of our plotting tool, we can find the two points on the VTC at which the inverter gain is unity (i.e., the VTC slope is –1 V/V) and, hence, determine V_{IL} and V_{IH}. Using the results given in Fig. B.77, the corresponding noise margins are $NM_L = NM_H = 1.34$ V for the inverter with $m_p/m_n = 4$, while $NM_L = 0.975$ V and $NM_H = 1.74$ V for the inverter with $m_p/m_n = 1$. Observe that these results correlate reasonably well with the values obtained using the approximate formula in Eq. (16.38). Furthermore, note that with $m_p/m_n = \mu_n/\mu_p = 4$, the NMOS and PMOS devices are closely matched and, hence, the two noise margins are equal.

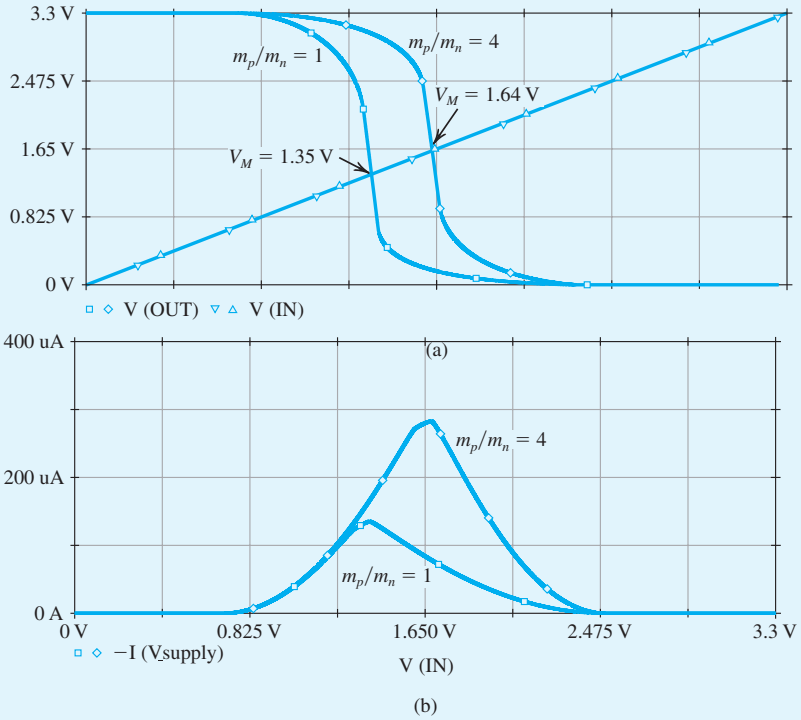


Figure B.78 **(a)** Output voltage and **(b)** supply current versus input voltage for the CMOS inverter in Example S.16.1 with $m_p/m_n = 1$ and $m_p/m_n = 4$.

The threshold voltage V_M of the CMOS inverter is defined as the input voltage v_{IN} that results in an identical output voltage v_{OUT}, that is,

$$V_M = v_{IN}\big|_{v_{OUT}=v_{IN}} \tag{B.46}$$

Thus, as shown in Fig. B.78, V_M is the intersection of the VTC, with the straight line corresponding to $v_{OUT} = v_{IN}$ (this line can be simply generated by plotting v_{IN} versus v_{OUT}, as shown in Fig. B.78). Note that

$V_M \simeq (V_{DD}/2)$ for the inverter with $m_p/m_n = 4$. Furthermore, decreasing m_p/m_n decreases V_M. Figure B.78 also shows the inverter supply current versus v_{IN}. Observe that the location of the supply-current peak shifts with the threshold voltage.

To investigate the dynamic operation of the inverter with SPICE, we apply a pulse signal at the input (Fig. B.76), perform a transient analysis, and plot the input and output waveforms as shown in Fig. B.79. The rise and fall times of the pulse source are chosen to be very short. Note that increasing m_p/m_n from 1 to 4 decreases t_{PLH} (from 1.13 ns to 0.29 ns) because of the increased current available to charge C_L, with only a minor increase in t_{PHL} (from 0.33 ns to 0.34 ns). The two propagation delays, t_{PLH} and t_{PHL}, are not exactly equal when $m_p/m_n = 4$, because the NMOS and PMOS transistors are still not perfectly matched (e.g., $V_{tn} \neq |V_{tp}|$).

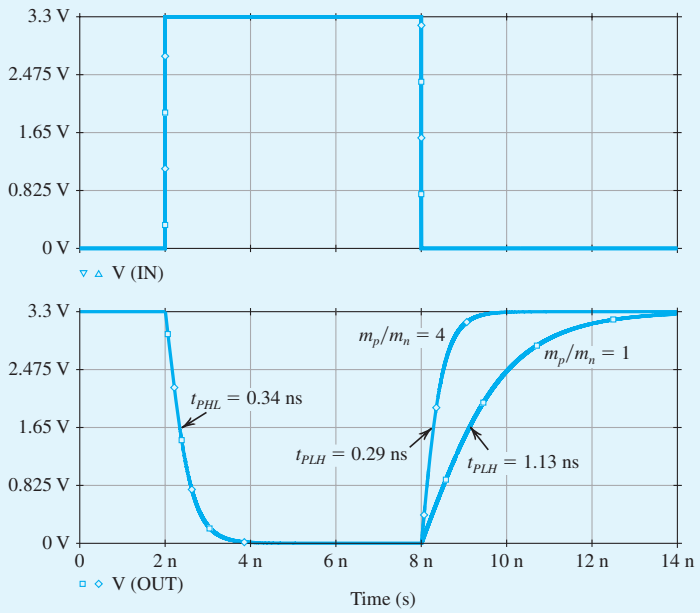


Figure B.79 Transient response of the CMOS inverter in Example S.16.1 with $m_p/m_n = 1$ and $m_p/m_n = 4$.