Evaluation Programs for SPICE Op Amp Models

Introduction

There is no consistent method for evaluating SPICE models in the industry, so it is hard to reproduce a specific manufacturer's results or to compare models between manufacturers. Furthermore, many of the SPICE models available from vendors do not correlate the vendor's data sheet for the corresponding op amp; hence, there is confusion about the validity of the model, the evaluation program, and/or the data sheet. This paper includes a collection of SPICE programs that have been used to evaluate some of the latest Intersil Corporation current feedback op amps. The programs illustrated here will be used to evaluate new Intersil op amp designs, both current and voltage feedback, so they will serve as a standard until modified by common agreement. These programs have several advantages: they are written, they have been debugged, macros exist to eliminate the plotting effort, they cover the pertinent parameters, and they contain equations to normalize the output for "Bode" type plots.

Six programs which cover 13 parameters including: inverting gain, non-inverting gain, positive power supply current, negative power supply current, positive input bias current, negative input bias current, offset current, positive offset voltage, negative offset voltage, differential offset voltage, non-inverting common-mode voltage, transient response, and enable/disable response are used to evaluate the op amp. These thirteen parameters are displayed in eight plots which have seventeen curves.

Printed copies of these programs are given here, and electronic copies are available on the "Intersil Corporation-Analog SPICE Macromodels" disk dated January 1996 or later.

AC Transfer Function For An Inverting Op Amp

The first program (see Figures 1, 2, and 3) is named cfaig.cir, and it simulates the AC transfer function for an inverting op amp. This program uses three op amps so it computes the transfer function for three different gains in one pass. The program requires the user to supply the feedback resistance values for each gain, the gain settings, the load resistance, the load capacitance, and the power supply voltage in volts. The program assumes that the op amp is run off two power supplies of equal and opposite polarity each of which is referenced to ground, so it applies the entered voltage to the op amp as a positive and negative supply with equal magnitudes. If a single supply op amp is to be evaluated with these programs just enter half the power supply voltage, and the analysis will be equivalent.

Unless the output is normalized the vertical scale will have to be large enough to accommodate the difference in gains, so small effects such as peaking may be hard to discern or measure. The program configures the load resistor as a voltage divider, and the output is taken at the voltage divider output. If the op amp gain is 10 the load resistor gain will be 0.1. If a load resistor is not required by the data sheet enter a large value such as $1000 G\Omega$; the large resistor will not affect the circuit operation while the normalization feature is retained. Now the three curves will plot on top of each other similar to the GBW curves shown in most data books.
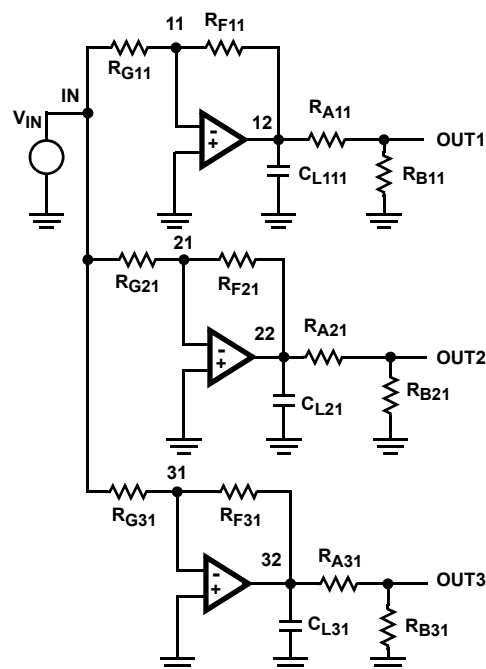

FIGURE 1. SCHEMATIC OF INVERTING OP AMP

*This program simulates the AC transfer function for an inverting op amp.

*It has three op amps; each with a gain that is specified by the

*user with a .param statement. The user must specify the load R_L, C_L, the

*feedback resistors R_{F1}, R_{F2}, R_{F3} and the corresponding gains G_1, G_2, G_3.

*The power supply voltage is set by the parameter "vsupply". The load resistors

*are automatically split into voltage dividers to normalize the gain plot,

*and the gains can be plotted in dB by calling the macros G_1, G_2, and G_3.

*The inputs are tied together, and the outputs are called OUT1, OUT2, and

*OUT3 corresponding to the respective gains. The op amp model is entered with

*a .lib statement. The model in the subcircuit call (x statement) must correspond to the model

*called in the .lib statement (3 times).

.param C_L=10pf

.param R_L=400

.param R_{F1}=750

.param R_{F2}=750

.param R_{F3}=750

.param G_1=1

.param G_2=2

.param G_3=3

.param vsupply=5

.lib b:ha502x.cir

x1 0 11 3 4 12 3 ha502x

x2 0 21 3 4 22 3 ha502x

x3 0 31 3 4 32 3 ha502x

V_{IN} in 0 ac 1

R_{F11} 11 12 {R_{F1}}

R_{F21} 21 22 {R_{F2}}

R_{F31} 31 32 {R_{F3}}

R_{G11} in 11 {R_{F1}/abs(G_1) }

R_{G21} in 21 {R_{F2}/abs(G_2) }

R_{G31} in 31 {R_{F3}/abs(G_3) }

R_{A11} 12 OUT1 {R_L* (abs(G_1)-.99999) /abs(G_1) }

R_{A21} 22 OUT2 {R_L* (abs(G_2)-.99999) /abs(G_2) }

R_{A31} 32 OUT3 {R_L* (abs(G_3)-.99999) /abs(G_3) }

R_{B11} 0 OUT1 {R_L/abs(G_1) }

R_{B21} 0 OUT2 {R_L/abs(G_2) }

R_{B31} 0 OUT3 {R_L/abs(G_3) }

C_{L11} 0 12 {C_L}

C_{L21} 0 22 {C_L}

C_{L31} 0 32 {C_L}

V_{CC} 3 0 {vsupply}

V_{EE} 4 0 {-1*vsupply}

.ac dec 50 1meg 3000meg

.probe

.end

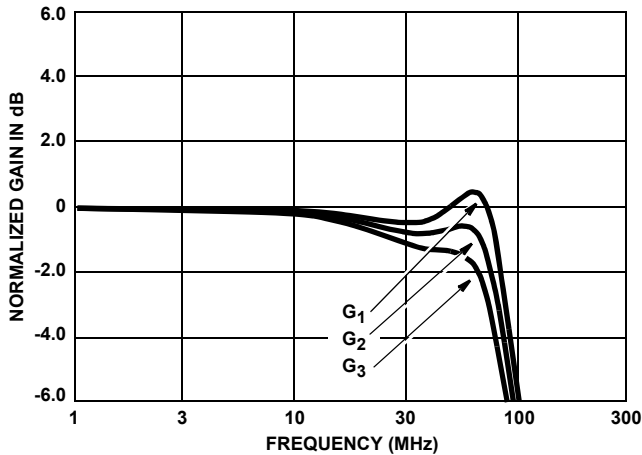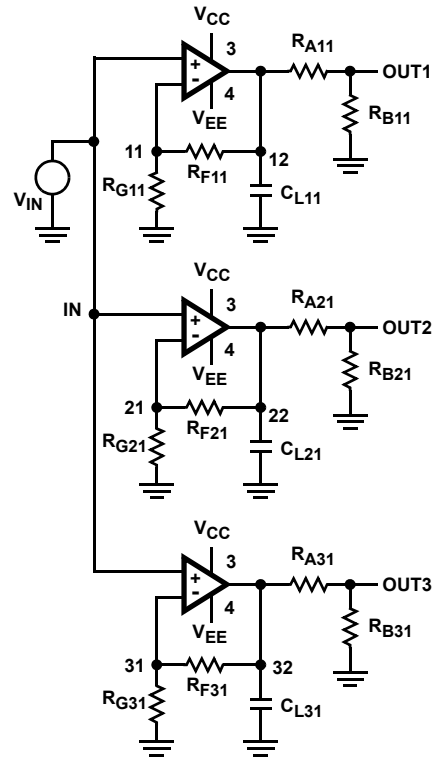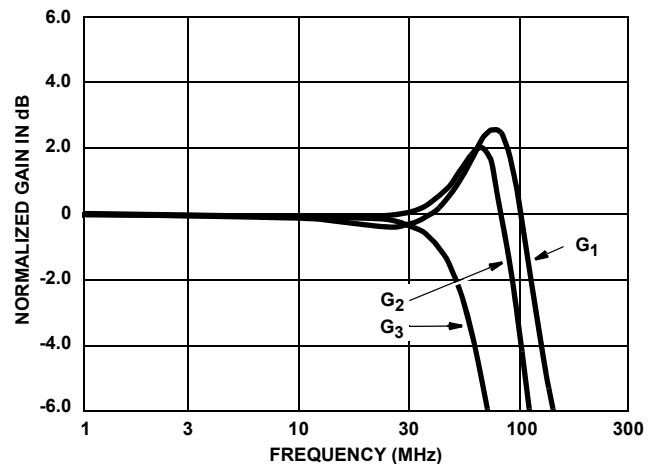**FIGURE 2. INVERTING OP AMP AC TRANSFER FUNCTION PROGRAM**

FIGURE 3.   INVERTING OP AMP AC TRANSFER FUNCTION PLOT

AC Transfer Function for a Non-Inverting Op Amp

The second program (see Figures 4, 5, and 6) is named cfanig.cir, and it simulates the AC transfer function for a non-inverting op amp. This program uses three op amps so it can compute the transfer function for three different gains in one pass. The program requires the user to supply the feedback resistance values for each gain, the gain settings, the load resistance, the load capacitance, and the power supply voltage in volts. The program assumes that the op amp is run off two power supplies of equal and opposite polarity each of which is referenced to ground, so it applies the entered voltage to the op amp as a positive and negative supply with equal magnitudes. If a single supply op amp needs to be evaluated just enter half the power supply voltage, and the analysis will be equivalent.

Unless the output is normalized the vertical scale will have to be large enough to accommodate the difference in gains, so small effects such as peaking may be hard to discern or measure. The program configures the load resistor as a voltage divider, and the output is taken at the voltage divider output. If the op amp gain is 10, the load resistor gain will be 0.1. If a load resistor is not required by the data sheet enter a large value such as $1000G\Omega$; the large resistor will not affect the circuit operation while the normalization feature is retained. Now the three curves will plot on top of each other similar to the GBW curves shown in most data books.

FIGURE 4. NON-INVERTING OP AMP SCHEMATIC

FIGURE 5. NON-INVERTING OP AMP AC TRANSFER FUNCTION PLOT

*This program simulates the transfer function for a non-inverting op amp.

*It has three op amps; each with a gain that is specified by the user

*with a .param statement. The user must specify the load R_L, C_L, the

*feedback resistors R_{F1}, R_{F2}, R_{F3} and the corresponding gains G_1, G_2, G_3.

*The power supply voltage is set by the parameter "vsupply". The load resistors

*are automatically split into voltage dividers to normalize the gain plot,

*and they can be plotted in dB by calling the macros G_1, G_2 and G_3.

*The inputs are tied together, and the outputs are called OUT1, OUT2, and

*OUT3 corresponding to the respective gains. The op amp model is entered with

*a .lib statement. The model in the subcircuit call (x statement) must correspond to the model

*called in the .lib statement (3 times).

.param C_L=10pf
.param R_L=400
.param R_{F1}=1000
.param R_{F2}=681
.param R_{F3}=383
.param G_1=1
.param G_2=2
.param G_3=10
.param vsupply=5
.lib b:ha502x.cir
x1 in 11 3 4 12 3 ha502x
x2 in 21 3 4 22 3 ha502x
x3 in 31 3 4 32 3 has02x
vin in 0 ac 1
R_{F1}1 11 12 {R_{F1}}
R_{F2}1 21 22 {R_{F2}}
R_{F3}1 31 32 {R_{F3}}
R_{G1}1 0 11 {R_{F1}/(G_1-.99999) }
R_{G2}1 0 21 {R_{F2}/(G_2-.99999) }
R_{G3}1 0 31 {R_{F3}/(G_3-.99999) }
R_{A1}1 12 OUT1 {R_L*(G_1-.99999)/G_1}
R_{A2}1 22 OUT2 {R_L*(G_2-.99999)/G_2}
R_{A3}1 32 OUT3 {R_L*(G_3-.99999)/G_3}
R_{B1}1 0 OUT1 {R_L-(R_L*(G_1-.99999)/G_1) }
R_{B2}1 0 OUT2 {R_L-(R_L*(G_2-.99999)/G_2) }
R_{B3}1 0 OUT3 {R_L-(R_L*(G_3-.99999)/G_3) }
C_{L1}1 0 12 {C_L}
C_{L2}1 0 22 {C_L}
C_{L3}1 0 32 {C_L}
V_{CC} 3 0 {vsupply}
V_{EE} 4 0 {-1*vsupply}
.ac dec 50 1meg 3000meg
.probe
.end

FIGURE 6. NON-INVERTING OP AMP AC TRANSFER FUNCTION PROGRAM

DC Parameters For a Non-Inverting Op Amp

The third program (see Figures 7 through 11) is named cfadc.cir, and it simulates the salient DC parameters for a non-inverting op amp. The program requires the user to supply the feedback resistance values, the load resistance, and the power supply voltage in volts. The program assumes that the op amp is run off two power supplies of equal and opposite polarity each of which is referenced to ground, so it applies the entered voltage to the op amp as a positive and negative supply with equal magnitudes. If a single supply op amp needs to be evaluated just enter half the power supply voltage, and the analysis will be equivalent.

The input signal to the op amp is a DC sweep. The sweep input enables a data analysis at $V_{IN} = 0V$ which is often a data book point, and the parameters can be evaluated at various other points of interest. The input currents can be examined by plotting the currents through the feedback resistor, R_F, and the input resistor, R_I. The difference between these currents is the input offset current. When the voltage is swept through zero the offset voltage for zero input voltage can be calculated. Either input offset voltage can be plotted by selecting the correct node voltage, or the differential input voltage can be plotted be selecting V(11)-V(13). The supply currents are plotted by selecting $I(V_{EE})$ or $I(V_{CC})$ for the negative and positive power supplies respectively.
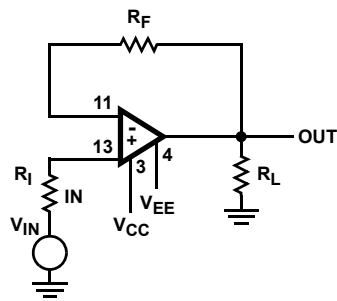
FIGURE 7. NON-INVERTING OP AMP SCHEMATIC (DC)
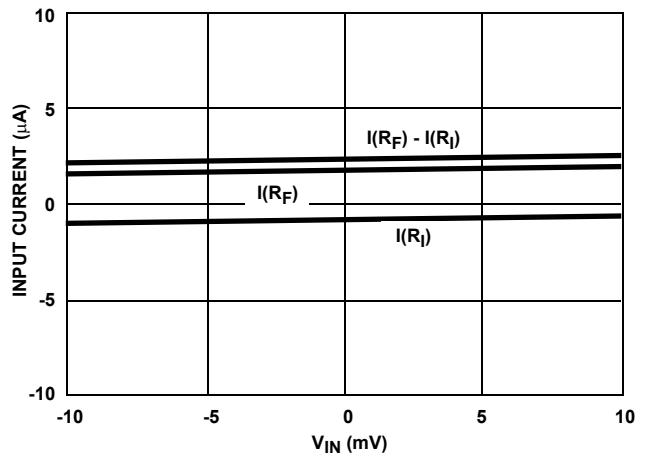
FIGURE 8. NON-INVERTING OP AMP INPUT CURRENT PLOT
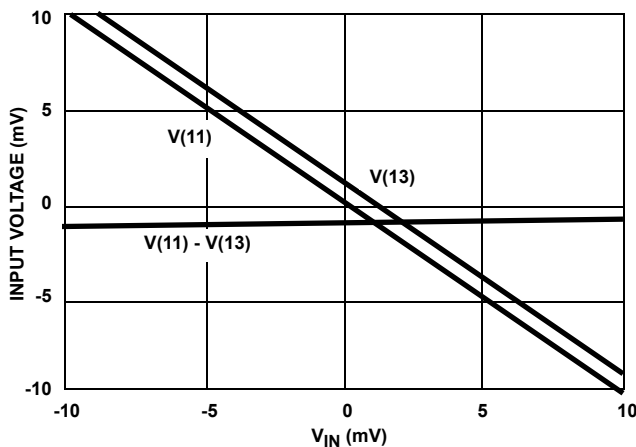
FIGURE 9. NON-INVERTING OP AMP INPUT OFFSET VOLTAGE PLOT
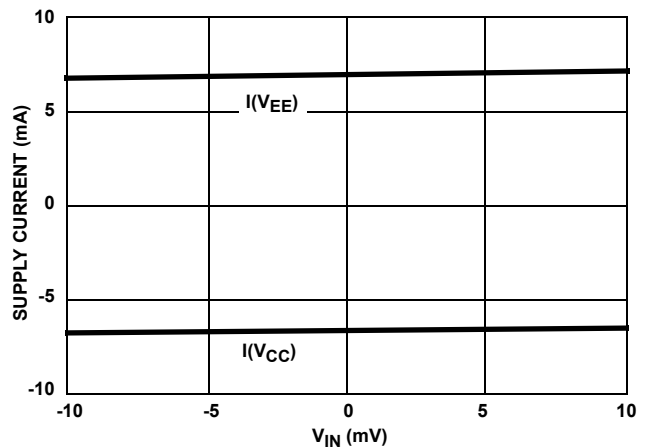
FIGURE 10. NON-INVERTING OP AMP POWER SUPPLY CURRENT PLOT

*This program simulates the salient DC parameters for a non-inverting op amp.

*The user must specify the feedback and load resistance with a .param

*statement. The power supply voltage is set by the parameter "vsupply". Input

*currents are measured as I_{RI}, the non-inverting input current, I_{RF}, the

*inverting input current, and (I_{RF}-I_{RI}) the offset current. The offset

*voltage is calculated with the equation Vos=v(13)-v(11). The power supply

*currents can be determined by looking at the parameter I_{CC} and I_{EE}. The

*model is entered with a .lib statement. The model in the subcircuit call (x statement)

*must correspond to the model called in the .lib statement.

.param R_F = 1000

.param R_L = 400

.param vsupply = 5

.lib b:ha502x.cir

x1 13 11 3 4 out 3 ha502x

V_{IN} in 0

R_{F1} 11 out {R_F}

R_I in 13 {R_F}

R_{L1} 0 out {R_L}

V_{CC} 3 0 {vsupply}

V_{EE} 4 0 {-1*vsupply}

.dc V_{IN} -.1 .1 .001

.probe

.end

FIGURE 11. NON-INVERTING OP AMP DC TRANSFER FUNCTION PROGRAM

CMRR For A Non-Inverting Op Amp

The fourth program (see Figures 12, 13, and 14) is called cfacmrr.cir, and it simulates the common mode rejection ratio for a non-inverting op amp. The program uses two identical non-inverting op amps to implement the equation CMRR = change in input offset voltage divided by the common mode input voltage change. The CMRR equation is shown in circuit parameters in Figure 12. Referring to Figure 13, it is seen that

the common mode difference voltage is 2V. The input is a square wave so the measurement should be made in an area which has settled out. Notice that the worse common-mode input voltage is negative so that value of -1.3mV was used in the calculation. Using a square wave rather than a DC signal enables the inspection of both quadrants prior to calculating the CMRR. The program requires the user to supply the feedback resistance value, the load resistance, and the power supply voltage in volts.

$$CMRR = DB\frac{(V(IN1) - V(11)) - (V(IN2) - V(21))}{(V(IN2) - V(IN1))} = DB\frac{\Delta V_{IO}}{\Delta V_{CM}}$$
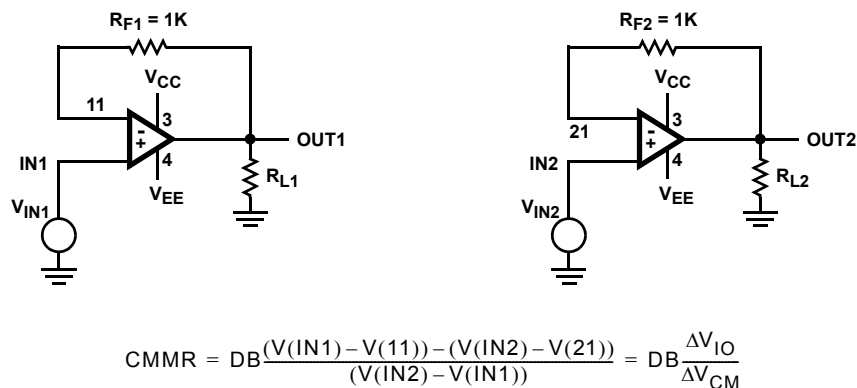
FIGURE 12. SCHEMATIC AND EQUATION FOR COMMON-MODE REJECTION CIRCUIT

*This program simulates the common-mode rejection ratio for a non-inverting op
*amp. The equation recommended for the calculation is
*CMRR=dB((V(IN1)-V(11)) -(V(IN2)-V(21)))/(V(IN2)-(VIN1)) and this program uses
*two identical op amps to obtain the calculation data. The user
*must specify the feedback resistance, R_F, and the load resistance, R_L.
*The power supply voltage is set by the parameter "vsupply". The op amp model
*is entered with a .lib statement. The model in the subcircuit call (x statement) must
*correspond to the model in the .lib statement (2 times).
.param R_F=1K
.param R_L=400
.param vsupply=5
.lib b:ha502x.cir
x1 IN1 11 3 4 OUT1 3 ha502x
x2 IN2 21 3 4 OUT2 3 ha502x
V_{IN1} IN1 0 pulse (0 1m .1ns .1ns .1ns 100ns 200ns)
V_{IN2} IN2 0 pulse (0 2.001 .1ns .1ns.1ns 100ns 200ns)
R_{F1} 11 OUT1 {R_F}
R_{F2} 21 OUT2 {R_F}
R_{I1} OUT1 0 {R_L}
R_{L2} OUT2 0 {R_L}
V_{CC} 3 0 {vsupply}
V_{EE} 4 0 {-1*vsupply
.tran 20ns 420ns
.probe
.end

FIGURE 13. COMMON-MODE REJECTION PROGRAM
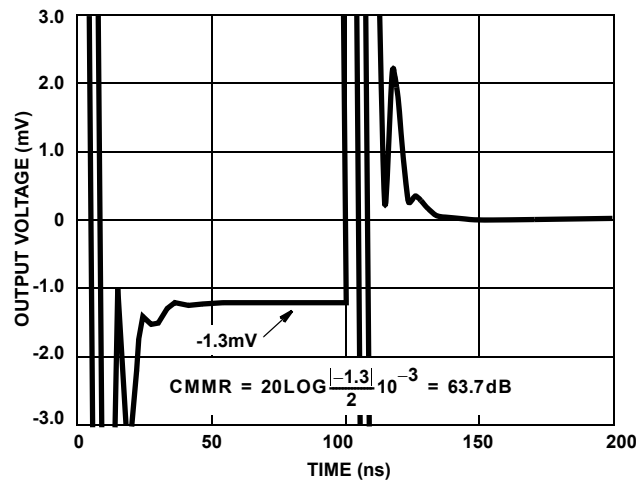
FIGURE 14. PLOT OF COMMON-MODE REJECTION PROGRAM OUTPUT

Transient Response For A Non-Inverting Op Amp

The fifth program (see Figures 15, 16, and 17) is called cfatran.cir, and it simulates the transient response for a non-inverting op amp. If the input signal is small, about 100mv as shown in Figure 16, the analysis will be small signal. Larger input signals will yield a large signal analysis. The program requires the user to supply the feedback resistance, the load

resistance, the input resistance, and the power supply voltage in volts. The program assumes that the op amp is run off two power supplies of equal and opposite polarity each of which is referenced to ground, so it applies the entered voltage to the op amp as a positive and negative supply with equal magnitudes. If a single supply op amp needs to be evaluated just enter half the power supply voltage, and the analysis will be equivalent. The rise time, fall time, slew rate, and delay time can be read off the plot shown in Figure 16.
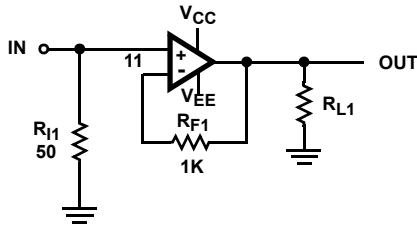
FIGURE 15. TRANSIENT RESPONSE CIRCUIT SCHEMATIC
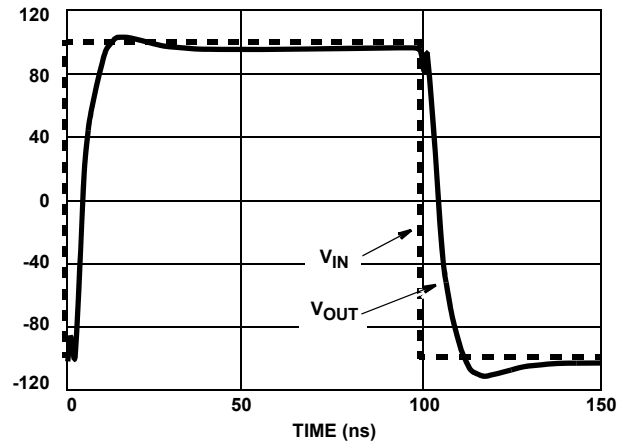
FIGURE 16. TRANSIENT RESPONSE PROGRAM OUTPUT PLOT

*This program simulates the time domain response for a non-inverting op amp.

*The response will be small signal or large signal depending on the amplitude

*of the input signal. The user must specify the load resistance, R_L, the

*feedback resistance, R_F, and the input resistance, R_I. The power supply

*voltage is set by the parameter "vsupply". The op amp model is entered with

*a .lib statement. The model in the x statement must correspond to the

*model called in the .lib statement.

.param R_I = 50

.param R_F = 1K

.param R_L = 100

.param vsupply=5

.lib b:ha502x.cir

x1 in 11 3 4 out 3 ha502x

V_{IN1} in 0 PULSE (-.1 .1 .1ns .1ns .1ns 100ns 200ns)

R_{F1} 11 out {R_F}

R_{L1} out 0 {R_L}

R_{I1} 0 in {R_I}

V_{CC} 3 0 {vsupply}

V_{EE} 4 0 [-1*vsupply]

.tran 20ns 420ns

.probe

.end

FIGURE 17. TRANSIENT RESPONSE PROGRAM

Enable Response For A Non-Inverting Op Amp

The Sixth program (see Figures 18, 19, and 20) is called cfaenabl.cir, and it computes the response of a non-inverting op amp to an enable control signal. A $2V_{DC}$ excitation is applied to the positive op amp input, and a square wave is applied to the enable input. The enable signal swings from ground to the positive power supply rail, thus simulating an open collector driver. This signal can be modified as required, but it must be disconnected from the supply voltage by removing the {vsupply} term

prior to modification. The program requires the user to supply the feedback resistance, the load resistance, the input resistance, and the power supply voltage in volts. The program assumes that the op amp is run off two power supplies of equal and opposite polarity each of which is referenced to ground, so it applies the entered voltage to the op amp as a positive and negative supply with equal magnitudes. If a single supply op amp needs to be evaluated just enter half the power supply voltage, and the analysis will be equivalent. The enable response times can be read off of Figure 19.
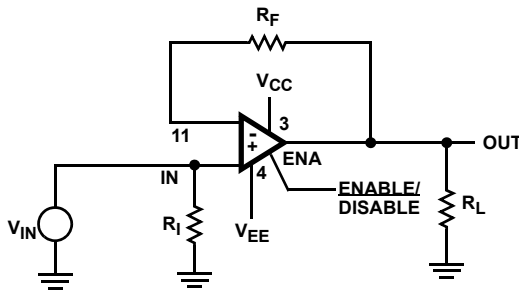
FIGURE 18. ENABLE RESPONSE CIRCUIT SCHEMATIC
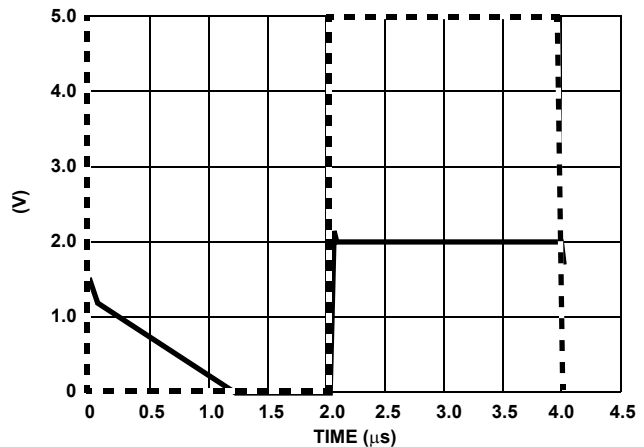
FIGURE 19. ENABLE RESPONSE PROGRAM OUTPUT PLOT

*This program simulates the time response of the enable/disable function of
*the non-inverting op amp. The response is obtained with a $2V_{DC}$
*input signal, while the voltage on the enable pin swings from V_{CC} to ground.
*The user must specify the load resistance, R_L, the feedback resistance, R_F,
*and the input resistance, R_I. The power supply voltage is set by a parameter
*"vsupply". The op amp model is entered with a .lib statement. The model in the subcircuit call
*(x statement) must correspond to the model called in the .lib statement. If
*the rise and fall times of the enable signal are too fast the program may
*not converge (10ns is optimal).
.param R_I = 50
.param R_F = 1K
.param R_L = 100
.param vsupply=5
.lib b:ha502x.cir
x1 in 11 3 4 out ena ha502x
V_{IN} in 0 2
vena ena 0 PULSE ({vsupply} 0 0ns 10ns 10ns 2000ns 4000ns)
R_{F1} 11 out {R_F}
R_{L1} out 0 {R_L}
R_{I1} 0 in {R_I}
V_{CC} 3 0 {vsupply}
V_{EE} 4 0 {-1*vsupply}
.tran 20ns 4020ns
.probe
.end

FIGURE 20. ENABLE RESPONSE PROGRAM

Summary

Six programs which compute and plot the response of an op amp are described here. These programs make it easy to complete the SPICE analysis, and by using them the results may be compared to new Intersil Corporation data sheets. This approach gives the minimum data required to evaluate the model, but the actual evaluation must be made by the design engineer. If the model indicates 2dB peaking at the -3dB point while the data sheet shows 3dB peaking at the same point, which does the engineer believe? Since the model is an approximation, the data sheet should be more correct than the model, but because the data sheet is based on "typicals" their curves are sometimes hard to reproduce. This vague area must be resolved through the experience of the design engineer coupled with laboratory data. There is no substitute for accurate measurements! When the data sheet curves, the model curves and the lab curves all fit within a reasonable tolerance the design engineer can begin to trust the models. Keep testing though, because models have been known to be very unpredictable.

Any comments, deletions or additions should be directed to one of the authors for inclusion in revisions on this document.

RENESAS

Renesas Electronics Corporation

http://www.renesas.com

SALES OFFICES

Refer to "http://www.renesas.com/" for the latest and detailed information.

Renesas Electronics America Inc.
1001 Murphy Ranch Road, Milpitas, CA 95035, U.S.A.
Tel: +1-408-432-8888, Fax: +1-408-434-5351

Renesas Electronics Canada Limited
9251 Yonge Street, Suite 8309 Richmond Hill, Ontario Canada L4C 9T3
Tel: +1-905-237-2004

Renesas Electronics Europe Limited
Dukes Meadow, Millboard Road, Bourne End, Buckinghamshire, SL8 5FH, U.K
Tel: +44-1628-651-700, Fax: +44-1628-651-804

Renesas Electronics Europe GmbH
Arcadiastrasse 10, 40472 Düsseldorf, Germany
Tel: +49-211-6503-0, Fax: +49-211-6503-1327

Renesas Electronics (China) Co., Ltd.
Room 1709 Quantum Plaza, No.27 ZhichunLu, Haidian District, Beijing, 100191 P. R. China
Tel: +86-10-8235-1155, Fax: +86-10-8235-7679

Renesas Electronics (Shanghai) Co., Ltd.
Unit 301, Tower A, Central Towers, 555 Langao Road, Putuo District, Shanghai, 200333 P. R. China
Tel: +86-21-2226-0888, Fax: +86-21-2226-0999

Renesas Electronics Hong Kong Limited
Unit 1601-1611, 16/F., Tower 2, Grand Century Place, 193 Prince Edward Road West, Mongkok, Kowloon, Hong Kong
Tel: +852-2265-6688, Fax: +852 2886-9022

Renesas Electronics Taiwan Co., Ltd.
13F, No. 363, Fu Shing North Road, Taipei 10543, Taiwan
Tel: +886-2-8175-9600, Fax: +886 2-8175-9670

Renesas Electronics Singapore Pte. Ltd.
80 Bendemeer Road, Unit #06-02 Hyflux Innovation Centre, Singapore 339949
Tel: +65-6213-0200, Fax: +65-6213-0300

Renesas Electronics Malaysia Sdn.Bhd.
Unit 1207, Block B, Menara Amcorp, Amcorp Trade Centre, No. 18, Jln Persiaran Barat, 46050 Petaling Jaya, Selangor Darul Ehsan, Malaysia
Tel: +60-3-7955-9390, Fax: +60-3-7955-9510

Renesas Electronics India Pvt. Ltd.
No.777C, 100 Feet Road, HAL 2nd Stage, Indiranagar, Bangalore 560 038, India
Tel: +91-80-67208700, Fax: +91-80-67208777

Renesas Electronics Korea Co., Ltd.
17F, KAMCO Yangjae Tower, 262, Gangnam-daero, Gangnam-gu, Seoul, 06265 Korea
Tel: +82-2-558-3737, Fax: +82-2-558-5338