

The analysis of negative feedback amplifiers by using the Pspice simulator

1. General overview

In this paper, the Pspice Student version 9.1 (the free version provided by the Cadence company) software is used to simulate a negative feedback amplifier (NFA).

The "Schematics" editor is run to draw / simulate the desired circuit. The **Draw-Get New Part ... (Ctrl + G)** menu is accessed to place components on the page. For example, the desired component (in our case a resistor **R1**) is placed conveniently by using the mouse on the schematic. If you want to rotate the component by a multiple of 90^0 , select the component (its symbol becomes red) and apply the **Edit-Rotate** command (**Ctrl + R**). The file may be saved by using the **File-Save As** command with a suggestive name (for example **NFA-SP**). Any other circuit changes may be saved with the **File-Save** command (**Ctrl + S**).

The components used inside the schematics are given in table 1.

Table 1

Name	Component	Value	Library
V_s	VAC	10mV	Source.slb
$R_x (x=1\dots n)$	R	- only the numerical value for ohms range - for kilo ohms range a "k" should be placed after the numerical value (no space between characters)	Analog.slb
$P_x (x=1\dots n)$	POT	5k The cursor position may be adjusted by changing the SET parameter between 0 and 1.00	Breakout.slb
$C_x (x=1\dots n)$	C	- For picofarads range: a p should be placed immediately after the numerical value - For nanofarads range: a n should be placed immediately after the numerical value - For microfarads range: a u should be placed immediately after the numerical value	Analog.slb
V_2, V_{CC}	VDC	15V	Source.slb
I_s	IAC	1uA	Source.slb
Q_1, Q_2	Q2N2222	NPN low power bipolar junction transistor (BJT)	Eval.slb
		putere	
Ground	GND_ANALOG	0	Port.slb

Four circuits that correspond for two types of negative feedback amplifier are drawn in Figs. 1-4 :

- Voltage amplifier with voltage-voltage negative feedback or series- shunt topology (the input of the amplifier is considered the base of Q_1 , the output is the collector of Q_2) (Figs.1-2);
- Transimpedance amplifier with voltage-current feedback or shunt-shunt (the input of the amplifier is considered the base of Q_1 , the output is the emitter of Q_2) (Figs.3-4).

For each type of the presented negative feedback amplifiers, two cases were treated:

- The circuit has a closed loop (Figs. 1, 3);
- The circuit has an open loop (Figs. 2, 4).

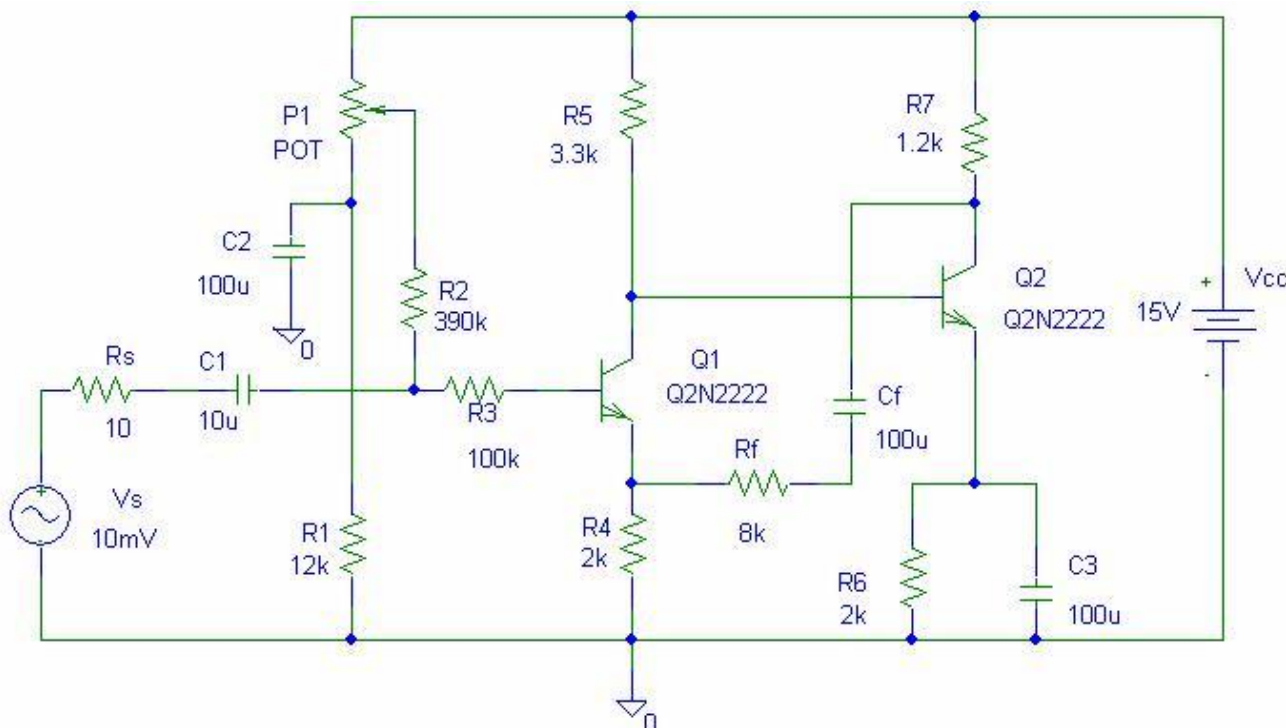


Fig 1 Voltage amplifier circuit with a series-shunt negative feedback

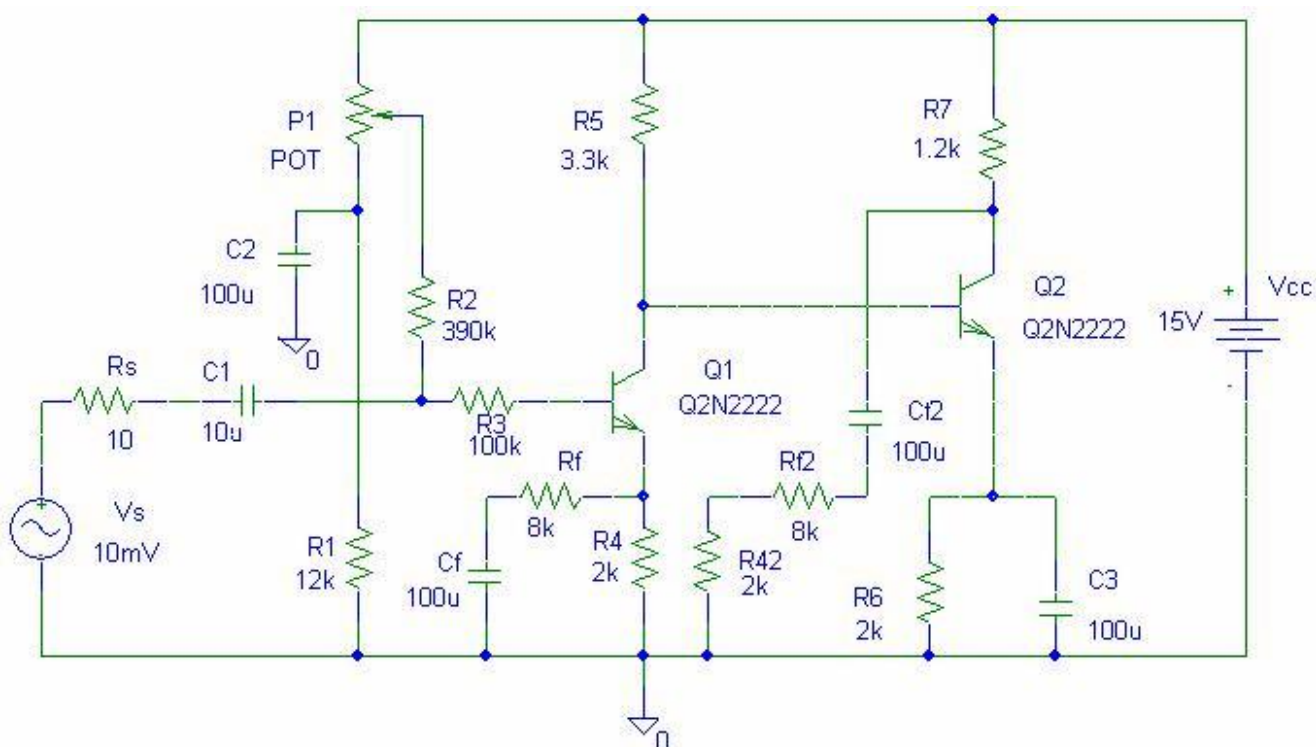


Fig. 2 Voltage amplifier circuit with open-loop series-shunt negative feedback

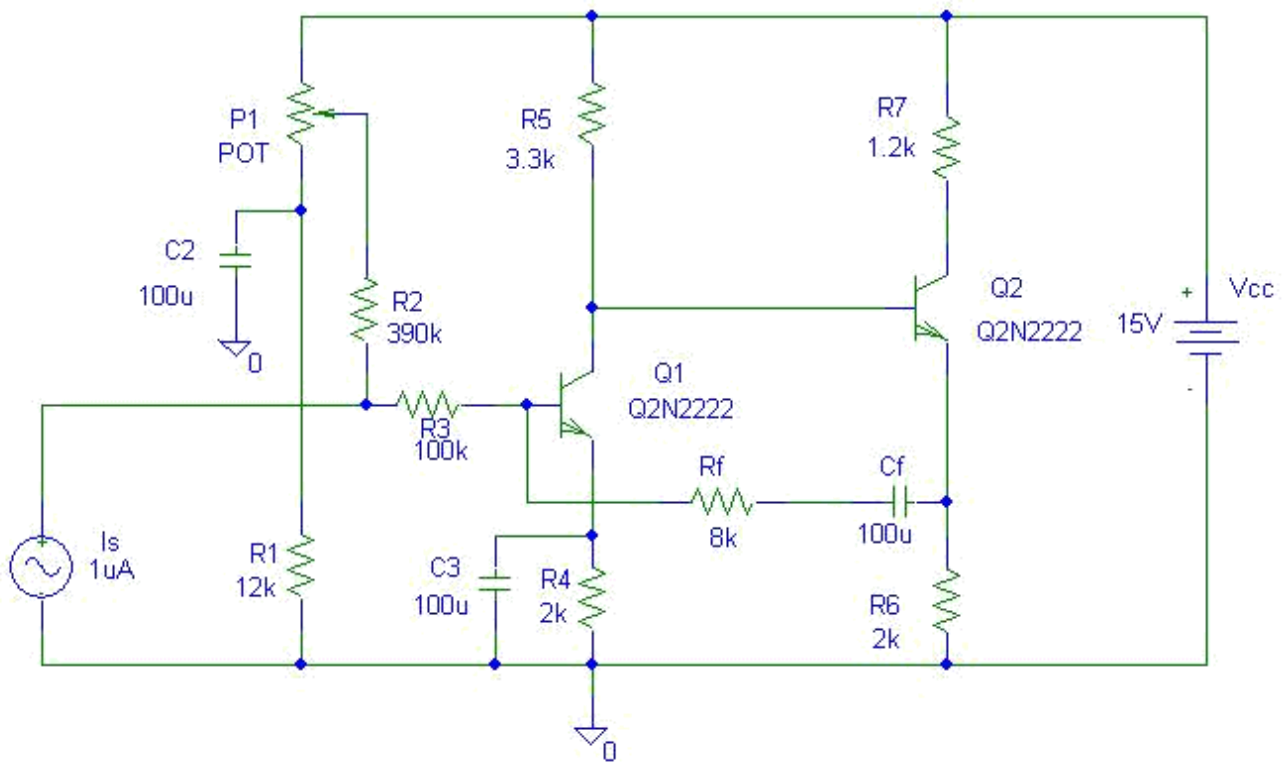


Fig 3. The transimpedance amplifier circuit with a shunt-shunt negative feedback

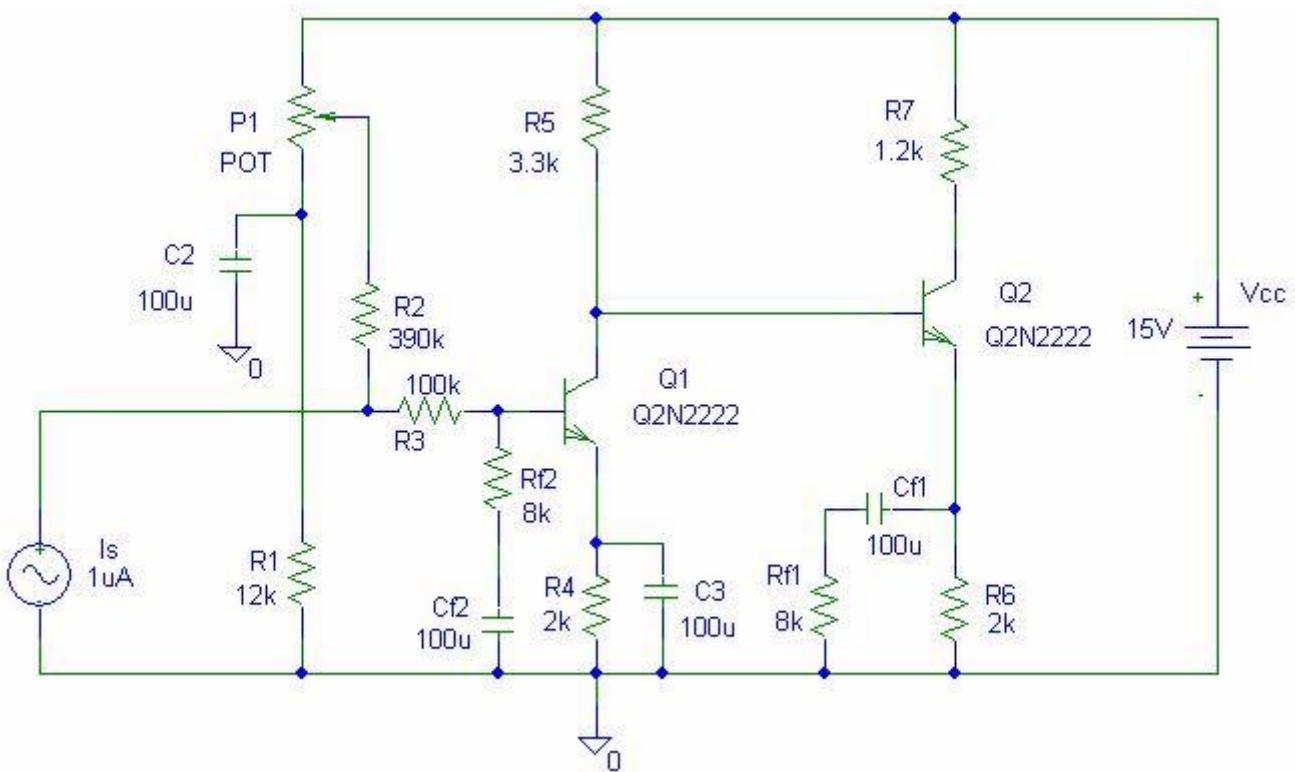


Fig. 4 Transimpedance amplifier circuit with a broken loop using shunt-shunt negative feedback

2. Editing, running, and graphical analysis of the results

The analysis in the frequency domain will be performed to determine the -3dB gain bandwidth of the circuit by using the "Analysis-Setup ... -AC Sweep (Enabled)" command. Choose "AC Sweep Type" = Decade for a logarithmic representation of the frequency on X axis. The Sweep Parameters are:

- Pts/decade =101;
- Start Freq =1
- End Freq = 100meg

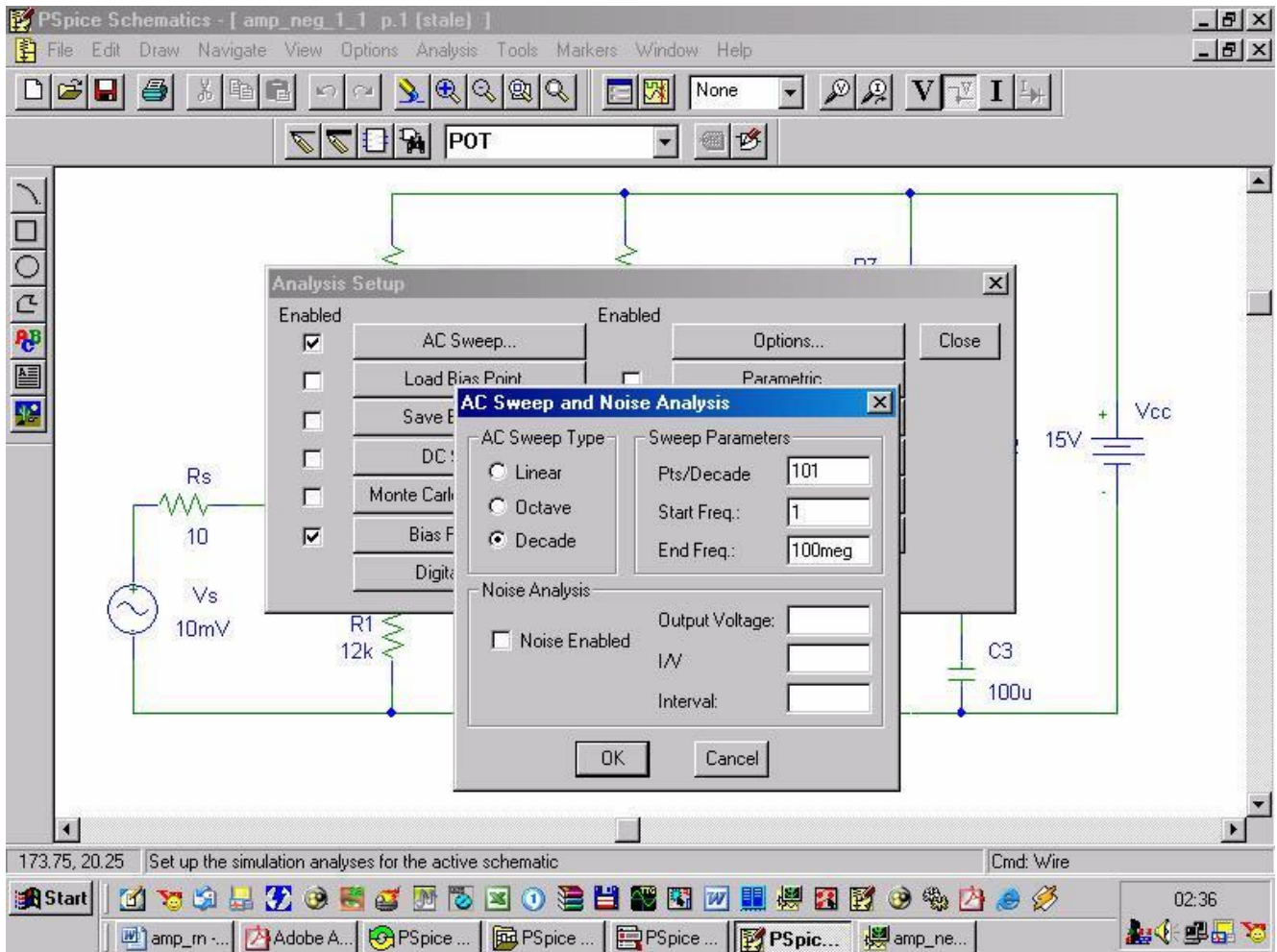


Fig. 5. Editing the parameters for the frequency domain analysis

In other words, the simulation will be done in the 1Hz-100MHz band, with a simulation step of 101 points per decade. After entering the simulation parameters, press the "OK" button and close the "Analysis Setup" menu by using the "Close" command. Run the simulation using the "Analysis - Simulate" command (or directly by pressing the F11 key).

After running the simulation, the "OrCAD PSpice A / D Demo" program is opening to graphically represent the gain versus frequency characteristic. To draw this graph, run the "Trace-Add Trace" command. If you want to view the voltage gain according to the frequency you should choose:

$$V(Q2:c) / V(Q1:b).$$

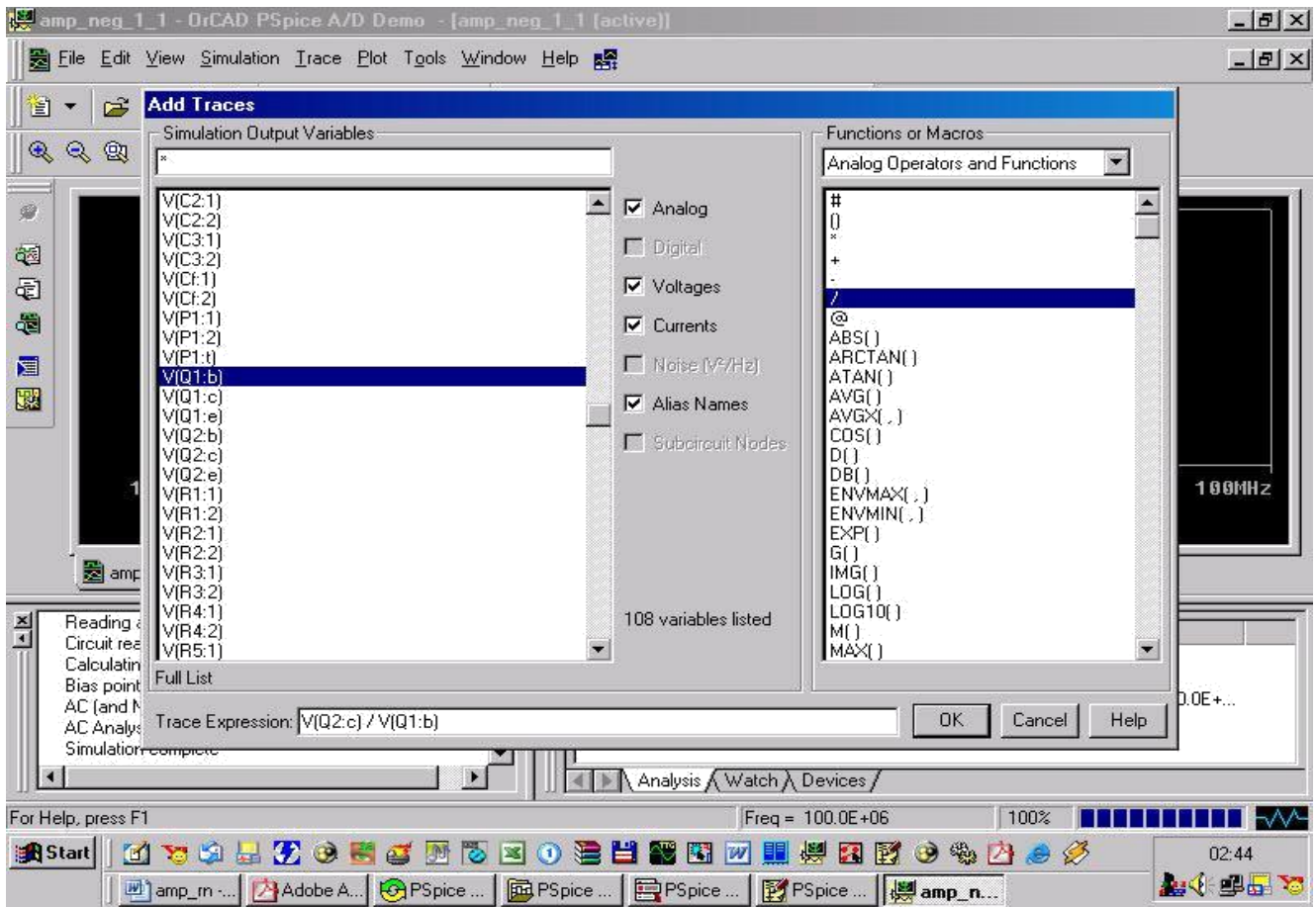


Fig. 6. Choosing the parameters for the gain-frequency characteristic

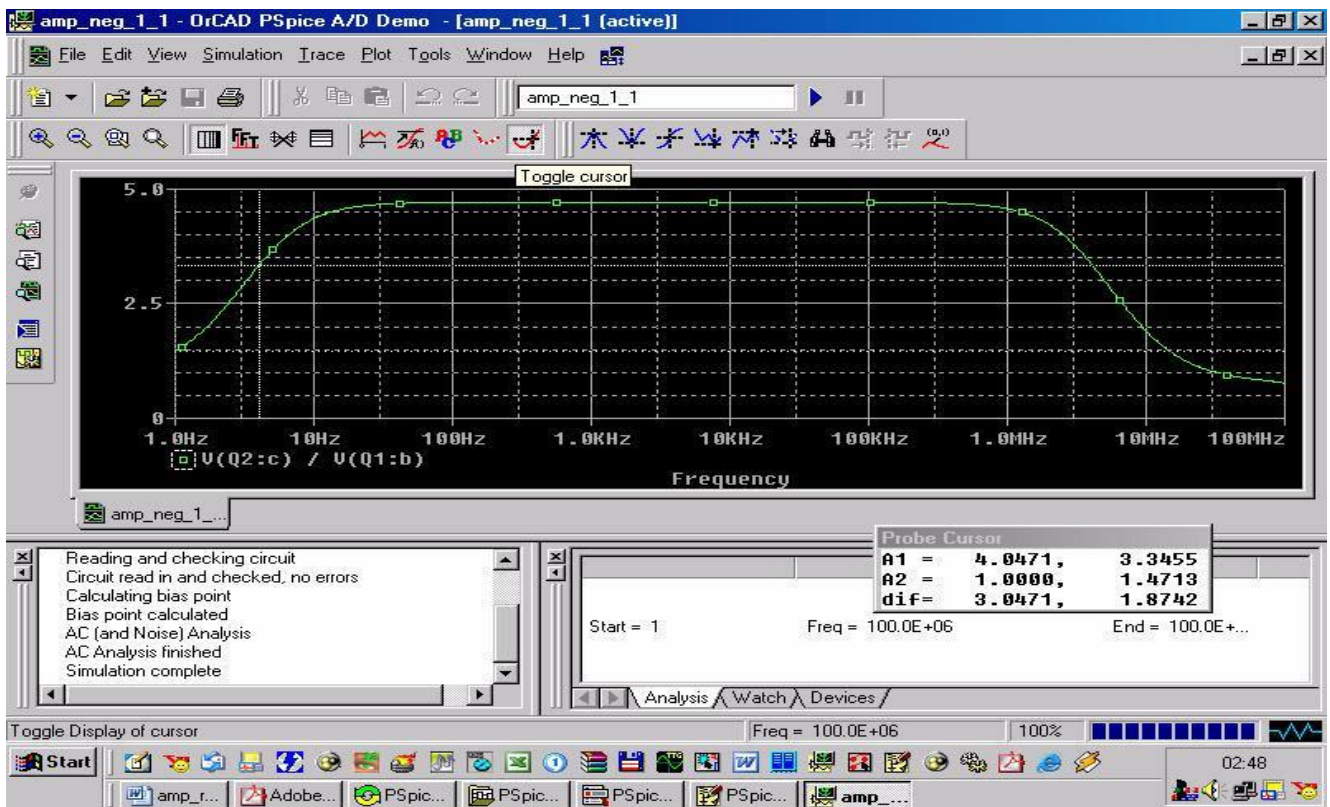


Fig. 7. The measurement of F_j and F_s frequencies by using the toggle cursor feature

Tick the Toggle Cursor and determine the lower limit F_j and the upper limit F_s frequencies values for the analyzed circuit (at $1/\sqrt{2}$ of the maxim gain).

3. Laboratory work

3.1. Draw the circuit corresponding to the negative feedback voltage amplifier shown in Figure 1 using the Schematics utility.

3.1.2 Modify the position of the P1 potentiometer cursor by choosing one of the following values for the SET parameter: 0.407, 0.7 si 0.99.

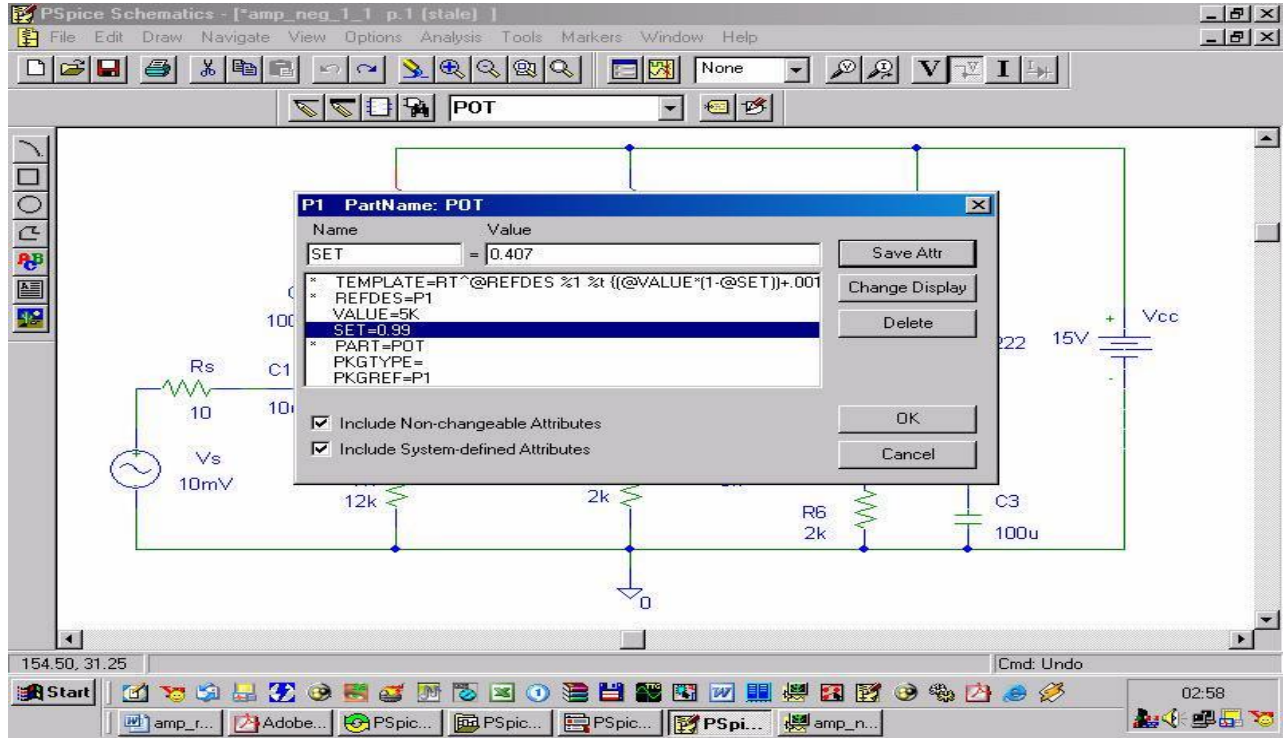


Fig. 8. Changing the cursor of the P potentiometer

3.1.3 Check the "Enable Bias Voltage Display" button marked with V for plotting the potentials in the nodes.

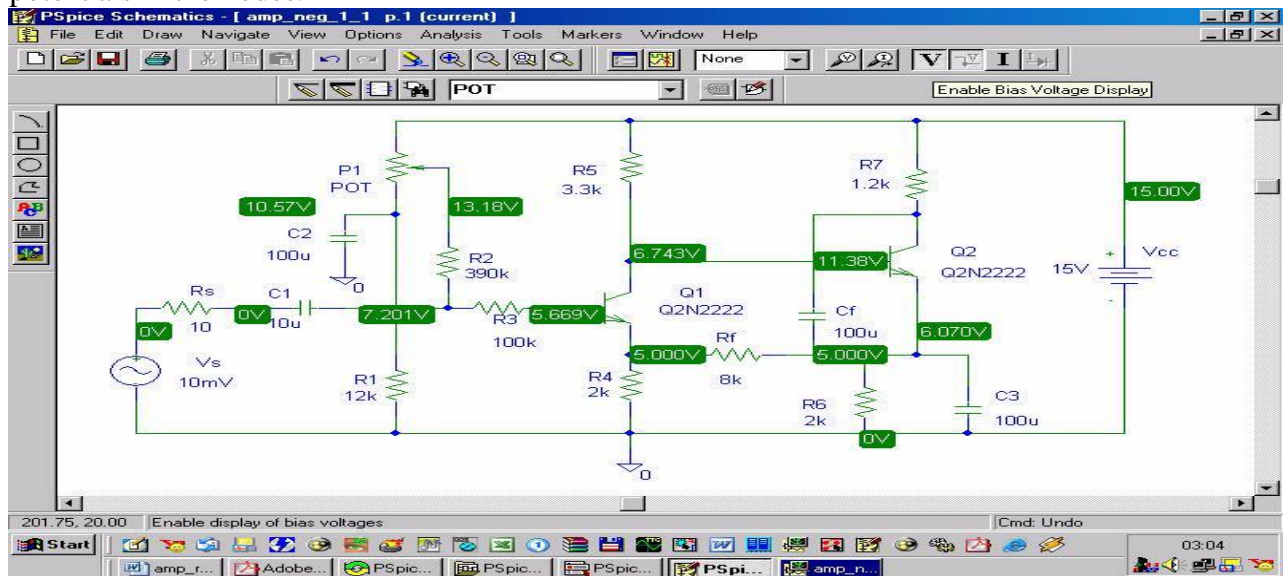


Fig 9. The representation of the potentials in the nodes of the diagram

3.1.4 Change the temperature of the components in the simulated circuit by running the following command: "Analysis –Setup -Temperature". The simulation will be done at different temperature values: 0⁰C , 27⁰C and 60⁰C.

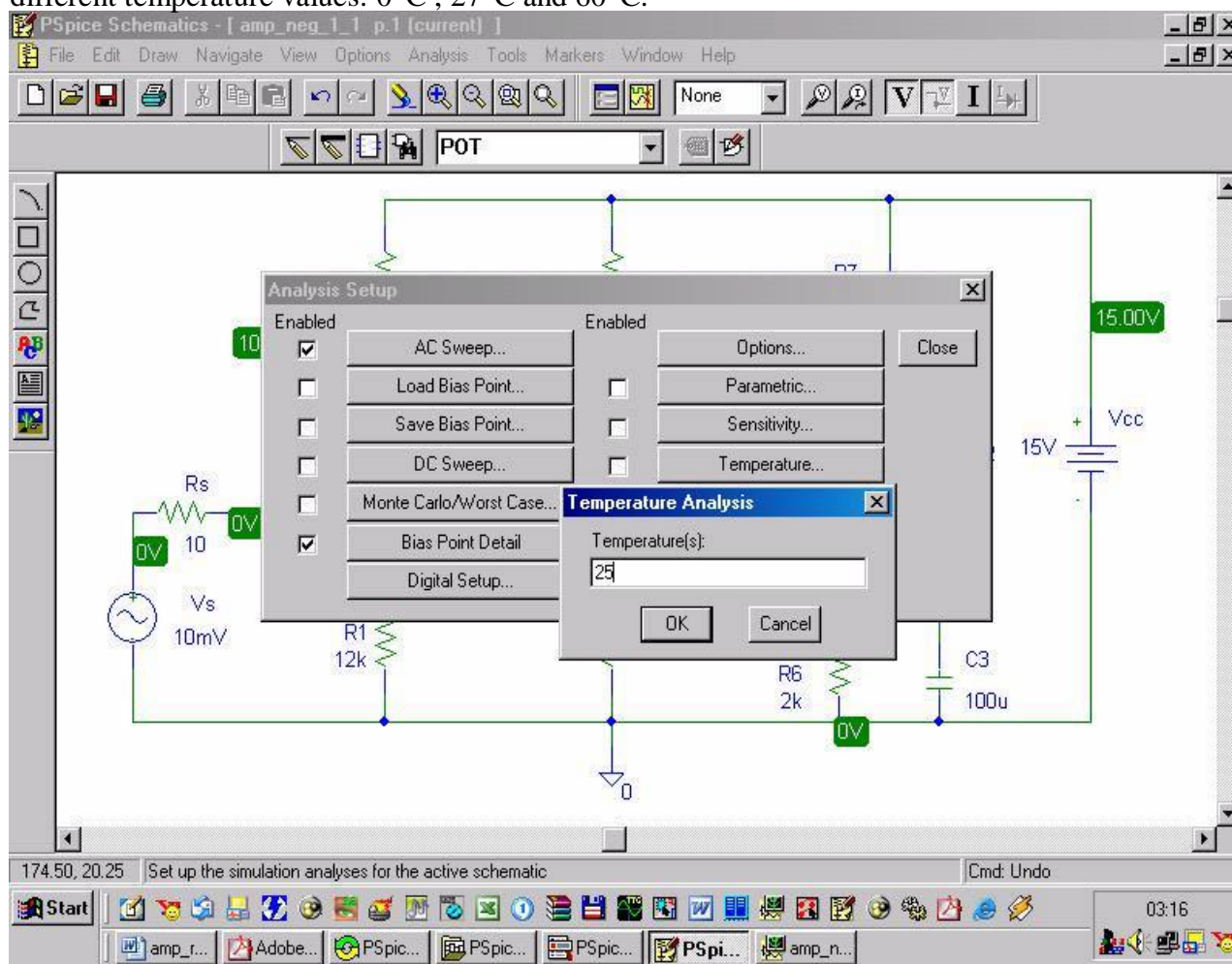


Fig.10. Changing the temperature of the components in the simulated circuit

When the simulation has been accomplished, the following parameters will be displayed:

3.1.5 The transimpedance gain (A_Z) : $V(Q2:c) / I(R3)$. Write the cutoff frequencies (the frequency values where the transimpedance gain is at $1/\sqrt{2}$ of the maximum value) F_J and F_S in *Table 2*.

Table 2

SET	A_Z	A_I	A_V	Z_I	F_J	F_S
0.407						
0.7						
0.99						

3.1.6 The current gain (A_I) will be determined: $I(R7) / I(R3)$. The voltage gain (A_V) will be noted: $V(Q2:c) / V(Q1:b)$

3.1.7 Determine the input impedance (Z_I): $V(Q1:b) / I(R3)$ and write its value in *Table 1*.

3.1.8 Select the SET parameter SET=0.407 and change the temperature values: 0⁰C , 27⁰C , 60⁰C. Resume the simulations from paragraphs 3.1.5 , 3.1.6 , 3.1.7 and put the results in *Table 3*.

Table 3

T (°C)	A _Z	A _I	A _V	Z _I	F _J	F _S
0						
27						
60						

3.2. By using the *Schematics* editor, draw the circuit corresponding to the voltage amplifier circuit with the with open-loop series-shunt negative feedback (Figure 2).

3.2.1 Determine the gain values (a_z , a_i , a_v), the open-loop input impedance (z_i) and the cutoff frequencies (f_j , f_s) by modifying the SET parameter. The results are displayed in *Table 4*.

Table 4

SET	a_z	a_i	a_v	z_i	f_j	f_s
0.407						
0.7						
0.99						

3.2.2 Determine the gain values (a_z , a_i , a_v), the open-loop input impedance (z_i) and the cutoff frequencies (f_j , f_s) by modifying the temperature (SET=0.407). The results are noted in *Table 5*.

Table 5

T (°C)	a_z	a_i	a_v	z_i	f_j	f_s
0						
27						
60						

3.3. Draw the circuit corresponding to the transimpedance amplifier (Figure 3) by using the *Schematics* utility.

3.3.1 Determine the gain values for the transadmittance amplifier (A_Y) $I_E(Q2) / V(Q1:b)$ for the different values of the SET parameter. List these values in *Table 6*.

3.3.2 Determine the gain values A_I ($I_E(Q2) / I(R3)$), A_V ($V(Q2:e) / V(Q1:b)$), the input impedance Z_I ($V(Q1:b) / I(R3)$) and the cutoff frequencies (F_J , F_S) when the SET parameter is changed (T=27°C). The results are written in *Table 6*.

Table 6

SET	A_Y	A_I	A_V	Z_I	F_J	F_S
0.407						
0.7						
0.99						

3.3.3 Determine the closed loop gain values (A_Y , A_I , A_V), the input impedance (Z_I) and the cutoff frequencies (F_J , F_S) by modifying the temperature (SET=0.407). The results are noted in *Table 7*.

Table 7

T (°C)	A_Y	A_I	A_V	Z_I	F_J	F_S
0						
27						
60						

3.4. Draw the circuit corresponding to the transimpedance amplifier circuit in the open loop configuration (Figure 4) using the Schematics program.

3.4.1 Determine the gain values (a_Y , a_I , a_V), the input impedance (z_I) and the cutoff frequencies (f_J , f_S) by modifying the SET parameter, keeping the temperature value constant ($T=27^{\circ}\text{C}$). The results are written in *Table 8*.

3.4.2. Determine the gain values (a_Y , a_I , a_V), the input impedance (z_I) and the cutoff frequencies (f_J , f_S) by modifying the temperature (SET=0.407). The results are noted in *Table 9*.

Table 8

SET	a_Y	a_I	a_V	z_I	f_J	f_S
0.407						
0.7						
0.99						

Table 9

T ($^{\circ}\text{C}$)	a_Y	a_I	a_V	z_I	f_J	f_S
0						
27						
60						

4. Questions

1. What is the effect of the shunt-shunt negative feedback on the voltage gain? And what is the effect of the shunt-shunt negative feedback on the current gain? Comment your answers.
2. What are the effects of the temperature changes on the studied circuits?
3. If the feedback loop in Fig. 1 is opened, what is the effect on the bandwidth of the amplifier?