Instytut Elektroniki Politechniki Łódzkiej
Wólczańska 223, 90-924 Łódź, Poland, tel. 042 636 00 65, fax 042 636 22 38

Radio Frequency Circuits Laboratory

Excercise 6 & 7
RF Circuits Designing with *Serenade* Software

Aim of the exercise

The aim of this exercise is to become acquainted with the methods of the radio frequency circuit design. Your task is to design the broadband amplifier with one bipolar transistor, and perform the optimization of the circuit using Serenade software.

The Serenade software allows for creating schematic drawings of circuits composed of active and passive elements, and components with distributed or focused parameters. The schematic can be converted into circuit file format and simulated with the Compact Software Simulation tools. The program offers a possibility of performing nonlinear analysis and optimization (analyzing work point of active elements, noise parameters, harmonics distortion, analysis of the voltage-controlled elements, etc.). Linear analysis and optimization are also possible (analysis of frequency characteristics, matching, etc.).

Your task is to create the schematic given in the instruction using Serenade software, choose the best transistor model for this purpose, define the range of the components for the linear optimization, define the parameters and goals of the optimization, and perform the analysis of the frequency characteristics of the designed circuit.

Amplifier parameters

The following amplifier parameters are assumed:

- input voltage	10V
- frequency band	250-500MHz
- input impedance	50Ω
- output impedance	50Ω
- gain ($\|S_{21}\|$)	12dB
- input matching ($\|S_{11}\|$)	<-20dB
- output matching ($\|S_{22}\|$)	<-20dB

You have to use the high frequency **npn** transistor of **BFQ 67** type. This is a small surface-mounted device. All the passive components employed in the circuit are manufactured using the same technology. As a result the circuit has small dimensions and the components have focused parameters for the frequencies up to 500MHz.

The broadband amplifier should amplify the signals of small amplitude. The work point of the chosen transistor is: $\mathbf{U_{CE}=4V}$ and $\mathbf{I_C=10mA}$. We are assuming that the circuit fulfills the requirements connected with nonlinear harmonics distortions and it does not introduce any additional noise. It means that the most important part of the design phase is the linear optimization (shape of the frequency characteristics). The nonlinear analysis (nonlinear distortions and noise) are less important.

Circuit design

In the design process the objectives are:

- achieve the required gain for given frequency range;
- achieve possibly flat gain characteristics;
- preserve the input and output matching values.

The first step of the design procedure is to choose the appropriate active element – the transistor model. It should have high gain value for specified frequency range and the work point. Keep in mind that the transistor's parameters, such as gain, nonlinear distortions and noise, depend mostly on the work point of the transistor. In this exercise the transistor type and its work point are already chosen. To achieve the desired gain characteristics, the common-emitter amplifier circuit should be employed.

The transistor's gain decreases as the frequency increases. To get the flat $|S_{22}|$ characteristics, the correcting components are necessary. You should employ the negative feedback components, such as resistor connected in series with the emitter, and the serial RL connection between the collector and the base.

The passive components in the feedback loop influence not only the gain characteristics but also the input and output impedance of the circuit. Changing the values of the passive components will give the possibility of achieving the desired matching characteristics ($|S_{11}|$ and $|S_{22}|$). To improve matching you should apply matching circuits.

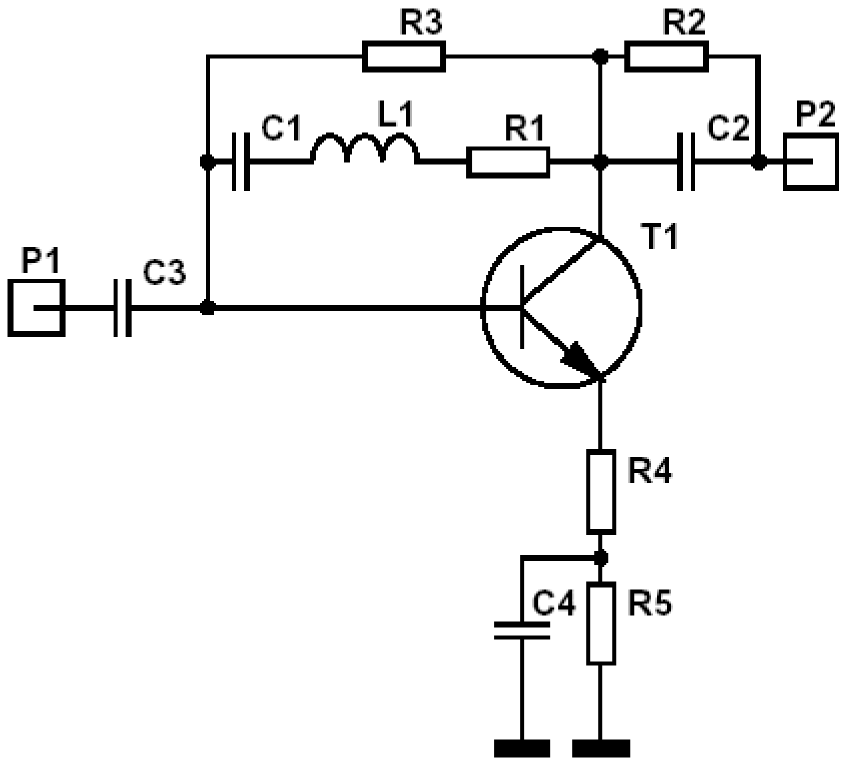

Figure 1Schematic circuit of the broadband amplifier

The common-emitter configuration of the amplifier is presented in figure 1. The resistors R2, R3 and R5 determine the work point of the transistor (R4 resistance is negligibly small comparing to R5). R1, C1 and L1, as well as R4 (negative feedback components) determine the frequency characteristics of the amplifier. C2 influences the input matching characteristics, while C4 is large blocking capacitance.

<u>Work point of the transistor</u>

The circuit is supplied through the input port P2. The current gain of the BFQ 67 transistor is equal to h_{21E}=100. The required work point (U_{CE}=4V, I_C=10mA) can be achieved by selecting proper values of R2, R3 and R5 resistances (R4 << R5). For simplicity we assume that resistances R2 and R5 are equal. This will make us sure to have high R5 resistance and the stable work point (thanks to the negative feedback for constant components). R3 resistance is used for proper polarization of the base.

Creating the project in Serenade software

The first step is creating the new project. Select *Project* option from the *Serenade Desktop* dialog box. The *New Project* dialog will pop up. Fill in the *Project Name* filed and click the *Create* button. If the *Serenade Desktop* dialog does not appear after starting the program, select *New→Project* from the program menu. Several windows will pop up, including the one containing error and warning messages. The schematic editor window has the same name as the project with the extension (**.sch*).

The schematic file (**.sch*). can be edited without creating the new project, however in such case you will not be able to perform the analysis of the circuit.

Editing the schematic

The basic parts for constructing the circuit can be found in the *Parts* menu. Some of the components are available also from the toolbar. The components placed in the design window must be properly connected using the *Wire* component from the *Parts* menu. Additionally the control blocks are necessary for performing the analysis and optimization of the circuit.

For example, to place the series RSL connection in the design window, click on the SRX toolbar button with the left mouse button. This component can be also accessed through *Parts→Lumped→Series/Parallel Combination→Series RLC*. Move the cursor to the design area, pick a position and click the left mouse button to fix the symbol in position. The *Properties* dialog box will pop up. Some field of this dialog are marked with **req** (required). Fill them in with the appropriate values as they are necessary to start the analysis of the circuit. The rest of the fields may remain empty.

In case of serial RLC connection, the required parameters are resistance, inductance and capacitance. The resistance is entered as a number without units (e.g. 47000 denotes 4.7kΩ). Capacitance and inductance values must be entered as a number followed by the prefix P, N, U or M and a unit (F for capacitance and H for inductance). For example, 3.3PF denotes 3.3pF capacitance, while *NH is the 8nH inductance. **Do not separate the numbers and the units.**

Other components are placed in the design area in the same way:

- resistor: *Parts→Lumped→Resistors;*
- capacitor: *Parts→Lumped→Capacitors→Ideal(Constatnt G);*
- inductor
- ground: *Parts→Ground;*

The main component of the circuit form fig. 1 is the transistor. It can be place in the schematic in the same way as the basic components. However, this method requires many parameters to be defined. Another method is to use active components models from the component library. The BFQ 67 transistor model can be accessed through *Parts→DeviceLibrary*. The *Device Library Section* dialog box will pop up. Define the manufacturer (for BFQ 67 it is Philips company) and the type of the component. For bipolar transistors linear and nonlinear models are available.

The *Schematic Connectors* menu allows you to get and place input and output connections to your schematic. Choose *Parts→Lumped→Schematic Connectors→Microwave Port*, place it in the design area, and enter the port name (p1, p2, etc.) in the *Instance Name* dialog box. In case of the circuit from fig. 1 the input port should be marked with p1, and output port – with p2. The ports have 50Ω resistance by default.

To delete the component from the schematic, place the cursor on the selected component, click the left mouse button and press Delete button on the keyboard. To enter new value for the component, double click on the component with the left mouse button. You can change the position of the component on the design plane by choosing *Edit→Move* or *Edit→Rotate* option after selecting the component.

Linear analysis

The first step is to define the frequency band. Select the *Linear Frequency* control block from the *Parts* menu. When the *Properties* dialog pops up, double click on the *req* in the *Value* filed for the *Freq.* Property. Now, type the instruction (e.g. *ESTP 20MHz 2GHz 50*). It means that an exponential sweep generates the frequencies in the range from 20MHz to 2GHz. Another possibility *STEP f1 f2 Δf* command, where a linear sweep from f_1 to f_2 in Δf steps is given.

When the control block is fixed, the linear analysis can be performed. Select *Settings→Linear* and then start the process selecting *Analysis→Analysis*. Yu can also do this using buttons from the toolbar.

To view the results of the analysis select *Report→Report Editor*. The *Liner Analysis Report* window will pop up. You have to define the *response* of the circuit and the *Function* of the response. For example, to view the $|S_{21}|$ plot in dB select *Response – S21* and *Function – dB()*. Next click the *Add Trace* button, then *Display*, and *Close*. You can plot several variables in one window using *Add Trace* command.

The plot may be modified. Place the mouse pointer on the element you want to modify and double click the left mouse button. The dialog box will pop up, where you can change thee parameters of the plot display. For example, to change the frequency scale from linear to logarithmic, double click the X-axis. The *Background Properties* dialog box pops up. Select *Log* from the *rescale*, confirm the changes by clicking *Apply* and close the dialog box.

You can switch between the windows using the list in the *Window* option form the menu.

Linear optimization

Before the optimization you have to define the components to be optimized and the goals of the optimization. To do this, you have to add two control blocks and change the components values to appropriate varying value names.

In case of the circuit in fig. 1, R1, L1, C2 and R2 components should be optimized. We assume that C1, C3 and C4 are all equal to 2.2nF, what simplifies the optimization process. The goal of the optimization is to achieve good input and output matching characteristics, and possibly flat gain characteristics in frequency range 250MHz-0.5GHz.

To change the value of a given component to the appropriate varying value name, double click its symbol, so that the dialog box pops up. Replace the component's value with the varying value name (the most convenient way is to use the same name for the component and the varying value assigned to it, e.g. R1, L1, C2, etc.). If several components in the circuit have the same value, you can assign the same varying value name to all of them.

The value and the range of each varying value are defined in *VAR* control block. Select *Parts→Control Block→Variables* and place the component in the design area. The dialog box will pop up. Enter the label name and the value using the first empty *Name* and *Value* fields. The value of the component is entered in the form: *?z1 w1 z2?*, where *w1* is the varying value in range *z1* to *z2*. For example, if there is an inductor L1 in the circuit with the property name L1 and varying initial value 15nH, you would write: *Name L1, Value ?1NH 15NH 50NH?* If the range is not defined, you would enter *?w1?*, where *w1* is the varying value. The component value will remain constant if you omit the question marks.

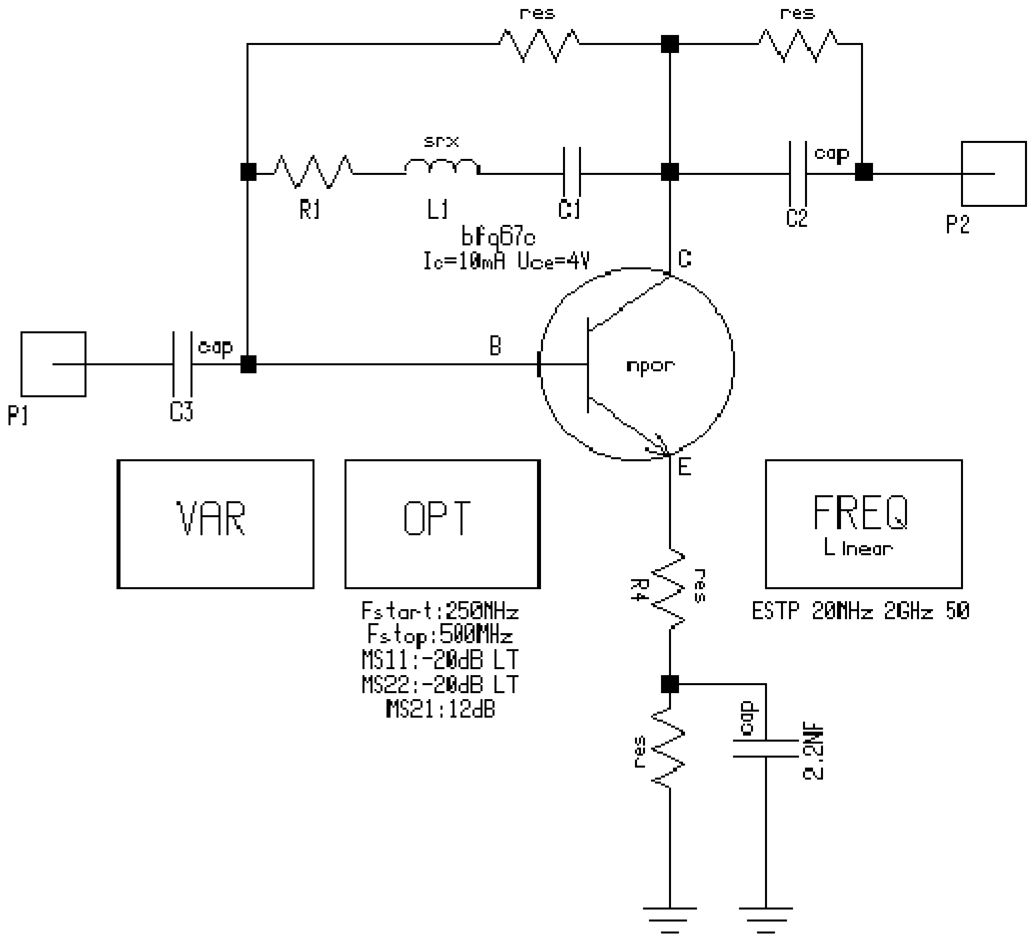

Figure 2 Schematic circuit of the amplifier for linear analysis and optimization

The optimization goals are given in *OPT* control block (*Parts→Control Blocks→Linear Optimization Info*). After adding OPT part, the *Linear Optimization* dialog box will pop up, where you have to enter the optimization frequency range and the desired circuit parameters. In out case the you have to type the following: *Fstart259MHz, Fstop 500MHz* and *MS11 –20dB LT, MS21 12dB, MS22 –20dB LT*. The *LT* keyword means that the value should be less or equal to the given value (possible is also using *GT* keyword meaning greater or equal).

When the control blocks are added to the schematic, you can start the optimization procedure. Select *Settings→Linear*, and then *Analysis→Optimization*. The Optimization dialog box will pop up. At first you have to perform the random optimization. Select *Optimization Type: Random*, specify the number of iterations, e.g. *Iterations: 60*, and press *Optimize* button. The second part is the gradient optimization with over a dozen iterations, e.g. *Optimization type: Gradient, Iterations: 15*. You can also start optimization using the toolbar buttons.

After the optimization process is finished, the varying values in the VAR control block will change. Perform once more the linear analysis of $|S_{11}|$, $|S_{21}|$ and $|S_{22}|$ characteristics in order to check the effect of the optimization. If you are not satisfied with the results, additional optimization iterations are necessary.

Nonlinear analysis

The nonlinear analysis requires the circuit modification. Replace the linear transistor model with the nonlinear one (*Parts→Device Library*) and choose the following options in the *Device Library Selection* dialog box: *nonlin, npn, Philips, bfq 67*. It is necessary to add the sinusoidal inducing signal at the input of the circuit, power supply components, and the *Nonlinear Analysis* control block (fig. 3).

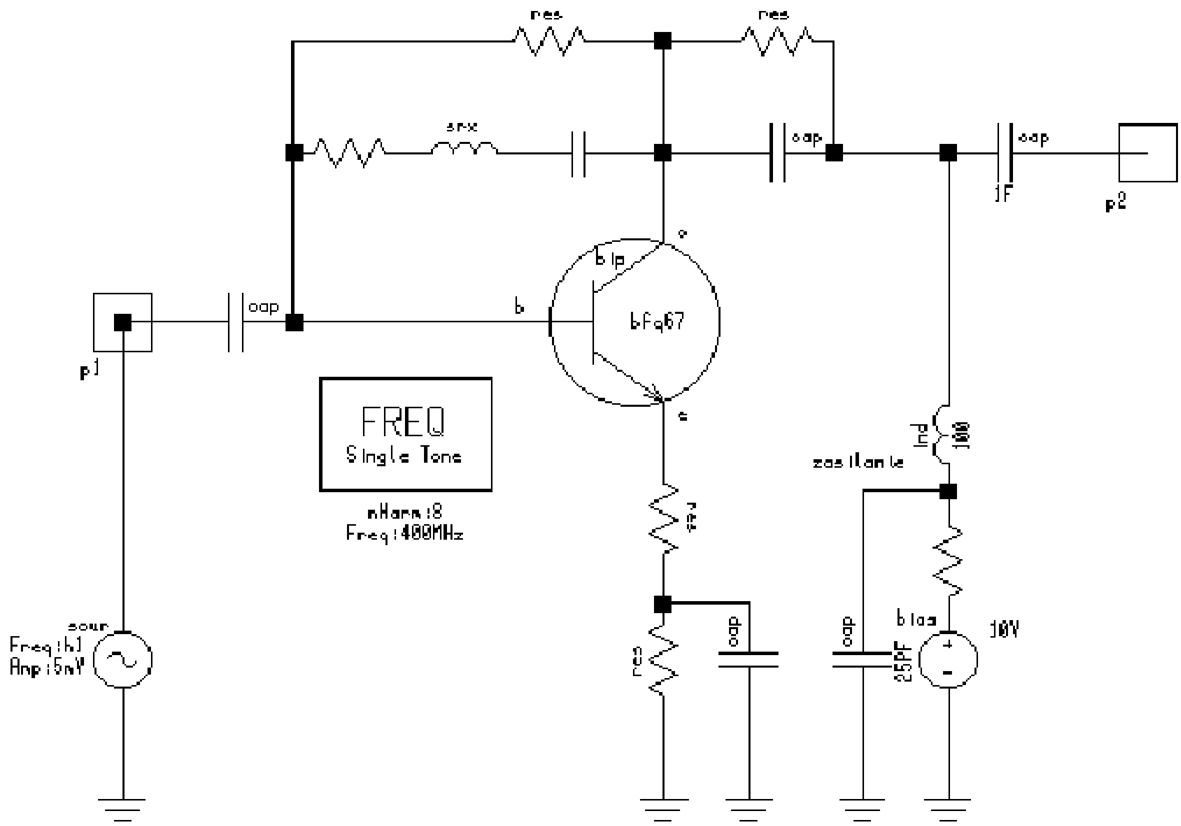

Figure 3 Schematic circuit of the amplifier for nonlinear analysis

Connect the sinusoidal inducing signal by selecting *Parts→Sources→Sinusoidal RF Source*. The RF source has the following parameters: *Freq h1, Amp. 500mV, Option E*. This is the voltage inducing signal (E) with the frequency of the first harmonics h1 and the amplitude equal to 500mV.

Connect the large capacitance (e.g. 1F) between the output port and the circuit. This will eliminate the constant component of the signal. The supply circuit for the transistor must be designed in such a way that it does not influence the frequency characteristics. Connect the voltage source (*Parts→Sources→DC Voltage Bias Source*) with parameters: *V 10V, Name Supply* (do not enter the R value). The voltage source must be connected to the circuit through the large inductance, e.g. 100H (*Parts→Lumped→Inductors→Ideal(Constant R)).* The small capacitance (e.g. 25pF) has to be connected in parallel between the voltage source and the ground.

The frequency for nonlinear analysis is defined in *FREQ Single Tone* control block (*Parts→Control Blocks→Nonlinear Frequency (Single Tone)*). In the *Nonlinear Frequency* dialog box specify the frequency value of the first harmonics and the number of the harmonics to be used in the simulation (e.g. *Freq 400MHz, nHarm 8*).

Select *Settings→Nonlinear* and start the analysis (*Analysis→Analysis*). In the *Nonlinear Circuit Analysis* dialog box select *Regular*, *Show Bias Point*, and press the *Analyze* button. Choosing the *Show Bias Point* option will cause the calculation of the work point of the transistor in the first phase of the analysis. Compare the calculated values of the collector current and the collector-emitter voltage with the assumed ones. In the second phase of the analysis, the parameters connected with the nonlinear distortions will be computed.

The analysis results can viewed on the plots. Select *Report→Report Editor*. In the *Nonlinear Analysis Report* dialog box select *Response – V1, Function – None, Domain – Time* and press the *Add Trace* button. This will display the input voltage characteristics. To view the output voltage characteristics, select *Response – V2*, press the *Add Trace* button, the *Display* button and close the dialog box.

You can use the same procedure to view the voltage harmonics at the output. Select *Report→Report Editor* and in the dialog box choose: *Response – V2, Function – None, Domain – Spectral*, add the trace by clicking the *Add Trace* button, display the plot and close the dialog box.

Perform the analysis several times for different amplitudes of the inducing signal and different frequencies of the inducing signal (the frequencies must be lower than the upper limit of the amplification frequency band). Observe if such signals may introduce distortions, and if they can, than in what conditions.

You can modify the plots just like in the case of the linear simulation. If you do not close the plot windows, they will be updated after each analysis of the circuit.

Experiment procedure

1. Draw the schematic circuit of the amplifier (fig. 1) for constant voltage and current components. Calculate the values of the resistance responsible for the work point of the transistor. Estimate the values of the components of the amplifier with the linear transistor model. **(Prepare before the classes.)**
2. Enter the circuit schematic with the properly defined parameters of the components.
3. Perform the analysis of $|S_{11}|$, $|S_{21}|$ and $|S_{22}|$ characteristics in the frequency range 20MHz – 2GHz.
4. Perform the circuit optimization in the frequency range 250MHz – 500MHz. The goal of the optimization is: $|S_{21}|$=12dB, $|S_{11}|$ < -20dB, $|S_{22}|$ < -20dB.
5. Replace the optimized values with the nearest standard decade values from table 1. Use E24 section.
6. Perform the analysis of $|S_{11}|$, $|S_{21}|$ and $|S_{22}|$ after the optimization. Assess whether results are satisfactory or not.
7. Enter the schematic circuit of the amplifier with the nonlinear transistor model and the optimized values of the components.
8. Perform the nonlinear analysis of the circuit for different frequencies and different amplitudes of the inducing signal. Determine the amplitudes and the frequencies of the signals that can be amplified by this circuit.

9*. Determine the $|S_{11}|$, $|S_{21}|$ and $|S_{22}|$ characteristics for the linear model of the transistor BFQ 67 working in the common-emitter configuration (work point U_{CE}=4V I_C=10mA, 50Ω impedance ports, frequency band 10MHz – 10GHz).

Table 1. Decade (mod10) multiples and submultiples.

E6	10	15	22	33	47	68						
E12	10	12	15	18	22	27	33	39	47	56	68	82
E24	10	11	12	13	15	16	18	20	22	24	27	30
	33	36	39	43	47	51	56	62	68	75	82	91
E48	100	105	110	115	121	127	133	140	147	154	162	169
	178	187	196	205	215	226	237	249	261	274	287	301
	316	332	348	365	383	402	422	442	464	487	511	536
	562	590	619	649	681	715	750	787	825	866	909	953
E96	100	102	105	107	110	113	115	118	121	124	127	130
	133	137	140	143	147	150	154	158	162	165	169	174
	178	182	187	191	196	200	205	210	215	221	226	232
	237	243	249	255	261	267	274	280	287	294	301	309
	316	324	332	340	348	357	365	374	383	392	402	412
	422	432	442	453	464	475	487	499	511	523	536	549
	562	576	590	604	619	634	649	665	681	698	715	732
	750	768	787	806	825	845	866	887	909	931	953	976