Simulation # 5

Noise analysis of Electronic circuits

Electrical Network Simulation Laboratory (EE2701)
Department of Electrical Engineering,
NIT Rourkela

Aim: **Noise analysis of Electronic circuits**

Objective:

a. Understanding the concept of noise in the electronic circuits
b. Noise analysis of op-amp based amplifier
c. Plotting spectral density curves for the circuit
d. Noise figure analysis for the BJT amplifier

Theory:

Noise in Electronic Circuits:

Noise is electrical or electromagnetic energy that reduces the quality of a signal. Noise affects digital, analog and all communications systems. Noise Analysis calculates the noise contribution from each resistor and semiconductor device at the specified output node. Multisim creates a noise model of the circuit using noise models of each resistor and semiconductor devices and then performs AC-like analysis. It calculates the noise contribution of each component and propagates it to the output of the circuit sweeping through the frequency range specified.

Multisim can model three different kinds of noise:

- **Thermal noise** (also known as Johnson or white noise) is temperature dependent and caused by the thermal interaction between free electrons and vibrating ions in a conductor. Its frequency content is spread equally throughout the spectrum.
- **Shot noise** is caused by the discrete-particle nature of the current carriers in all forms of semiconductors. It is the major cause of transistor noise.
- **Flicker noise** is usually generated by BJTs and FETs and occurs in frequencies below 1 KHz. This is type of noise is also known as excess noise or pink noise. It is inversely proportional to frequency and directly proportional to temperature and DC current levels.

Multisim performs **Noise Analysis** using the following approach:

- Each resistor and semiconductor device is considered as noise generator.
- Each noise generator's contribution is calculated and propagated by the appropriate transfer function to the output of the circuit.
- The total output noise at the output node is the RMS (Root Mean Square) sum of the individual noise contribution.
- The result is then divided by the gain from input source to the output source to get the equivalent input noise. This is the amount of noise which, if injected at the input source into a noiseless circuit, would cause the previously calculated amount of noise at the output.

Noise figure:

Noise figure is used to specify exactly how noisy a device is. For a transistor, noise figure is simply a measure of how much noise the transistor adds to the signal during the amplification process. In a circuit network, the noise figure is used as a figure of merit to compare the noise in a network with the noise in an ideal or noiseless network. It is a measure of the degradation in signal-to-noise ratio (SNR) between the input and output ports of a network.

Noise figure for a particular component (for instance, a transistor) is provided on the part datasheet. In a SPICE model, the noise figure is specified by the parameter NF. Below is an extract of a SPICE model for a transistor:

```
.MODEL BF517 NPN (IS=0.480F NF=1.008 BF=99.655 VAF=90.000 IKF=0.190
+ ISE=7.490F NE=1.762 NR=1.010 BR=38.400 VAR=7.000 IKR=93.200M
+ ISC=0.200F NC=1.042
```

Although noise figure in a model is achieved using a combination of several model parameters, the parameter NF is the most significant value (when simulating noise figure in a transistor model).

Multisim calculates the noise figure using the equation:

$$F = NO / GNS$$

Where, Multisim prints the noise figure in dB, that is: 10 log10 (F).

Table 1 Parameters used in Noise Analysis

Parameter	Meaning
Input noise reference source	Specifies the name of the independent voltage or current source that is to be the input reference source to which equivalent input noise is referred.
Output Node	Specifies the node at which all noise contributions will be summed.
Reference Node	Specifies the reference node for the output noise voltage.
Change Filter	Displays nodes contained within subcircuits or hierarchical blocks. There are three options: **Display internal nodes**. Displays nodes within hierarchical blocks and subcircuits. **Display submodules**. Displays components within semiconductor devices determined by the SPICE model of the semiconductor device. **Display open pins**. Displays all unconnected nodes of the circuit.
Calculate power spectral density curves	Generates a graph of the power spectral density.
Points per summary	Specifies how often the noise contributions of each noise generating device are reported. The recommended value is 1.
Calculate total noise values	Generates a table with total noise data.

Table 2 Parameters used in Noise Figure Analysis.

Parameter	Meaning
Input noise reference source	Specifies the name of the independent voltage or current source that is to be the input reference source to which equivalent input noise is referred.
Output Node	Specifies the node at which all noise contributions will be summed.
Reference Node	Specifies the reference node for the output noise voltage.
Change Filter	Displays nodes contained within sub circuits or hierarchical blocks. There are three options: • **Display internal nodes**. Displays nodes within hierarchical blocks and sub circuits. • **Display sub modules**. Displays components within semiconductor devices determined by the SPICE model of the semiconductor device. • **Display open pins**. Displays all unconnected nodes of the circuit.
Frequency	Specifies the frequency for the analysis.
Temperature	Specifies the temperature for the analysis (typically 27°C).

Procedure:

A. Noise analysis

1. Figure 1 shows a basic operational amplifier with a gain of 5. You will use **Noise Analysis** to obtain results for noise voltage for **R1** and **R2** and display a graph of the noise spectrum across a frequency range between 1 Hz and 10 GHz.
2. Select **Simulate » Analyses » Noise Analysis**. The **Noise Analysis** window opens. Table 1 describes the **Analysis Parameters** tab in detail.
3. Configure **the Noise Analysis Parameter**
4. Select the **Output** tab
5. Select the **Variables in circuit** list, select **All variables** from the drop-down list, and then highlight **inoise_total_rr1** from the list. Click the **Add** button to move the variable to the right side under **Selected variables for analysis.**
6. Repeat this process for the **inoise_total_rr2** variable. The **Output** tab will look as shown in Figure 2.
7. Click **Simulate**. The **Grapher View** opens and displays the noise contribution for each resistor

B. Spectral Density Plot

1. Use circuit in Figure 1 for plotting spectral density.
2. Select **Simulate » Analyses » Noise Analysis**. In the **Analysis Parameters** tab enable **Calculate power spectral density curves**.
3. Enter 1 in the **Points per summary** field.
4. Select the **Output** tab.
5. Add the variables **onoise_rr1** and **onoise_ rr2** to the **Selected variables for analysis** list
6. Click **Simulate**. The **Grapher View** shows the noise spectral density curves.

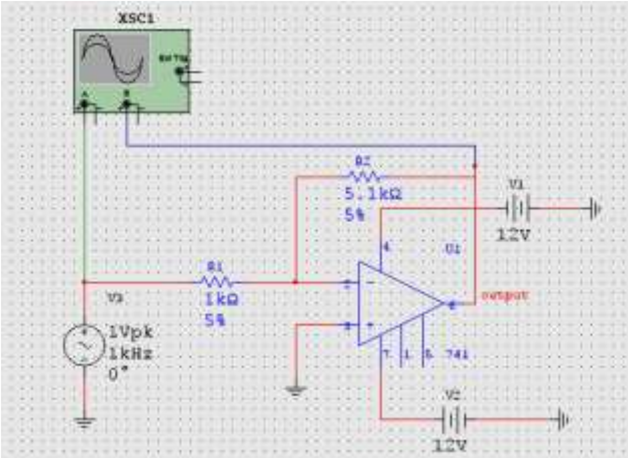

Figure 1 Circuit for noise analysis

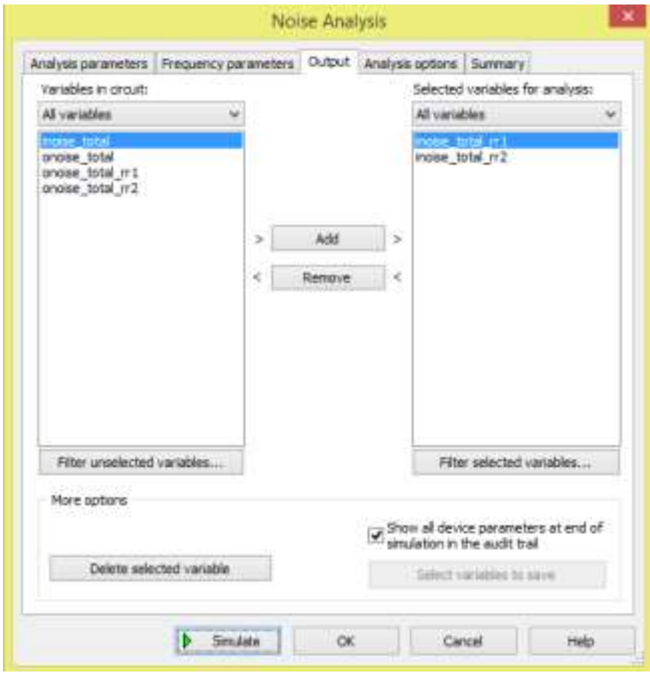

Figure 2Outoput tab settings for Noise analysis

C. Noise Figure analysis

1. Consider the RF Amplifier circuit shown in Figure 3. The amplifier is configured for maximum output power at 1.5MHz. You will use **Noise Figure Analysis** to calculate the noise figure at 10 kHz.
2. Use the Bode Plotter to verify the frequency response of the circuit. Using the cursor you can verify that the maximum gain is at approximately at 1.5 MHz.
3. Close bode plot and stop simulation.
4. Select **Simulate » Analyses » Noise Figure Analysis**. The **Noise Figure Analysis** window opens.
5. Configure the **Analysis Parameters**.Click **Simulate**. The **Grapher View** opens and displays the result.

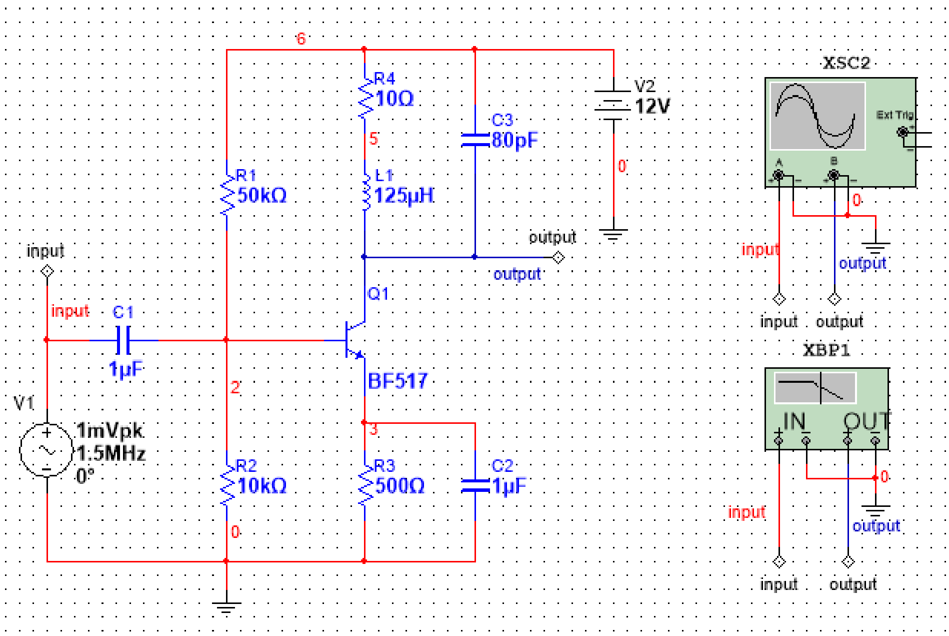

Figure 3 Circuit for Noise figure analysis

Questions:

1. Why signal strength is often expressed in logarithmic decibels.
2. List the various types of noise in electronic circuits along with the sources.
3. What is the effect of changing gain of amplifier on the integrated noise of resistor?
4. Comment on the relation of noise voltage with the frequencies from the spectral density curves.
5. Discuss the effect of frequency and temperature on noise in the amplifier circuit simulated.
6. What is the difference between noise analysis and noise figure analysis?