OptiSPICE

Tutorials - Basic

Opto-Electronic Circuit Design Software

Version 5.2

Optiwave

OptiSPICE

Tutorials - Basic

Opto-Electronic Circuit Design Software

Table of Contents

Basic Tutorials

To quickly gain an understanding of the core features provided with OptiSPICE it is recommended to perform the tutorials included in this document. It includes the following examples:

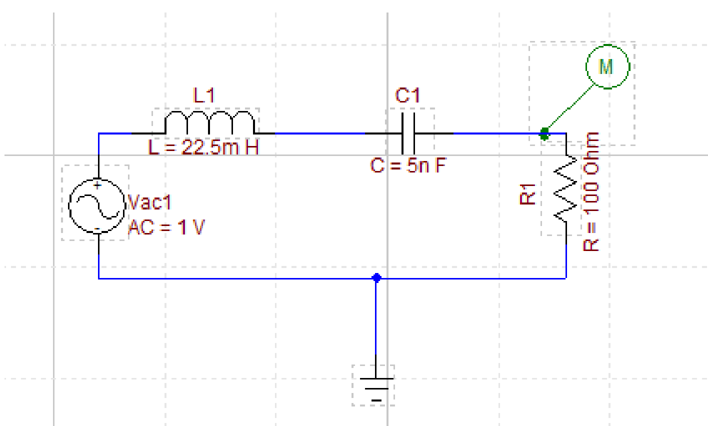
- ***AC Analysis*** which shows how to setup and run an AC simulation
- ***DC Analysis*** which shows how to setup and run a DC simulation
- ***Parameter sweep*** which shows the steps involved in performing the parameter sweep analysis of an electrical circuit.
- ***Wavelength sweep*** which shows how to determine the frequency response of an optical circuit by sweeping the wavelength of a laser source.
- ***Transient and noise simulation*** which shows how to run a transient simulation of an electrical circuit that includes electrical noise.

AC Analysis

This example demonstrates the steps involved in performing AC analysis for the design shown in Figure 1.

> ***Note:*** The OptiSPICE Schematic associated with this example can also be found within the folder: *OptiSPICE 5.2 Samples\Tutorials\Basic\AC Analysis*

Figure 1   Example band-pass filter for AC analysis

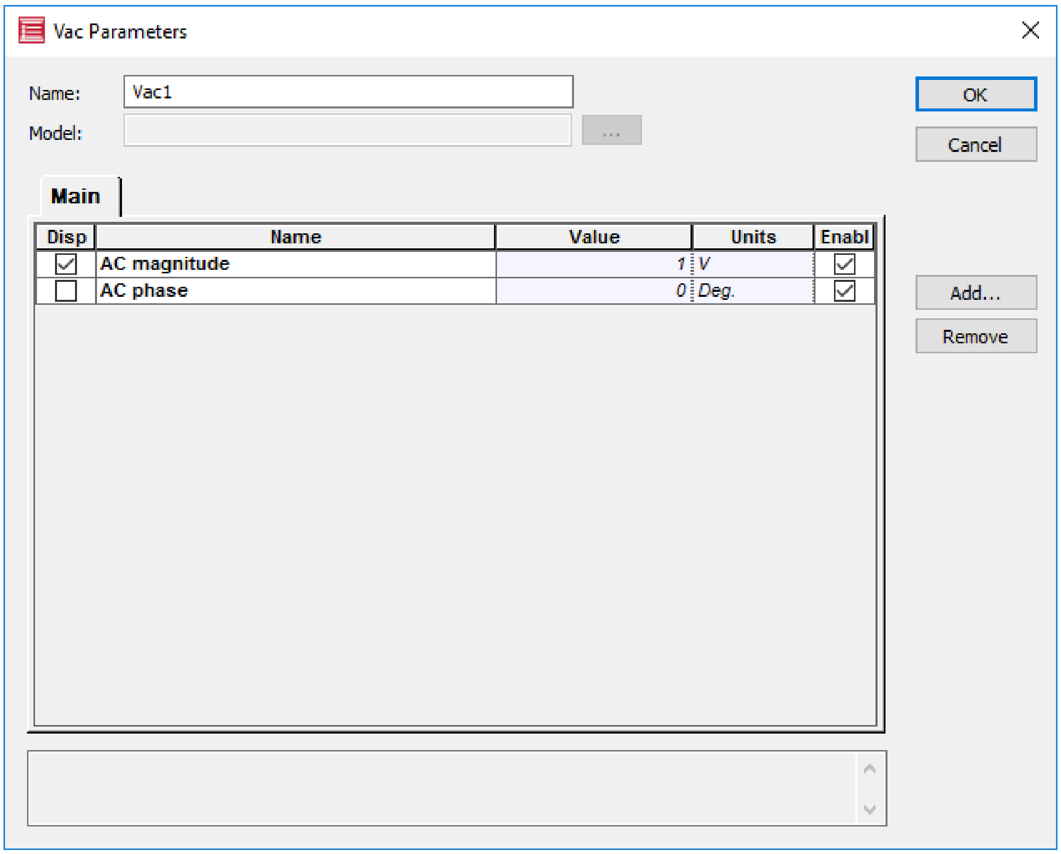


Circuit design

For the AC analysis, you need to provide an AC voltage or a current source. In this design, an AC voltage source is used. To place an AC source into the schematic, select the device **Vac** from the **Electrical** library. For details about placing and connecting devices, see the OptiSPICE Schematics book. Complete the circuit by placing and connecting necessary devices (inductor, capacitor, resistor and probe) as shown in Figure 1 and change the device parameter values as well. To change AC source values, perform the following steps:

Step	Action
1	Double click **Vac1** device.
2	In the dialog box, enter **1** for the **AC magnitude** as shown in Figure 2.
3	Click **OK**.

Figure 2 AC source values

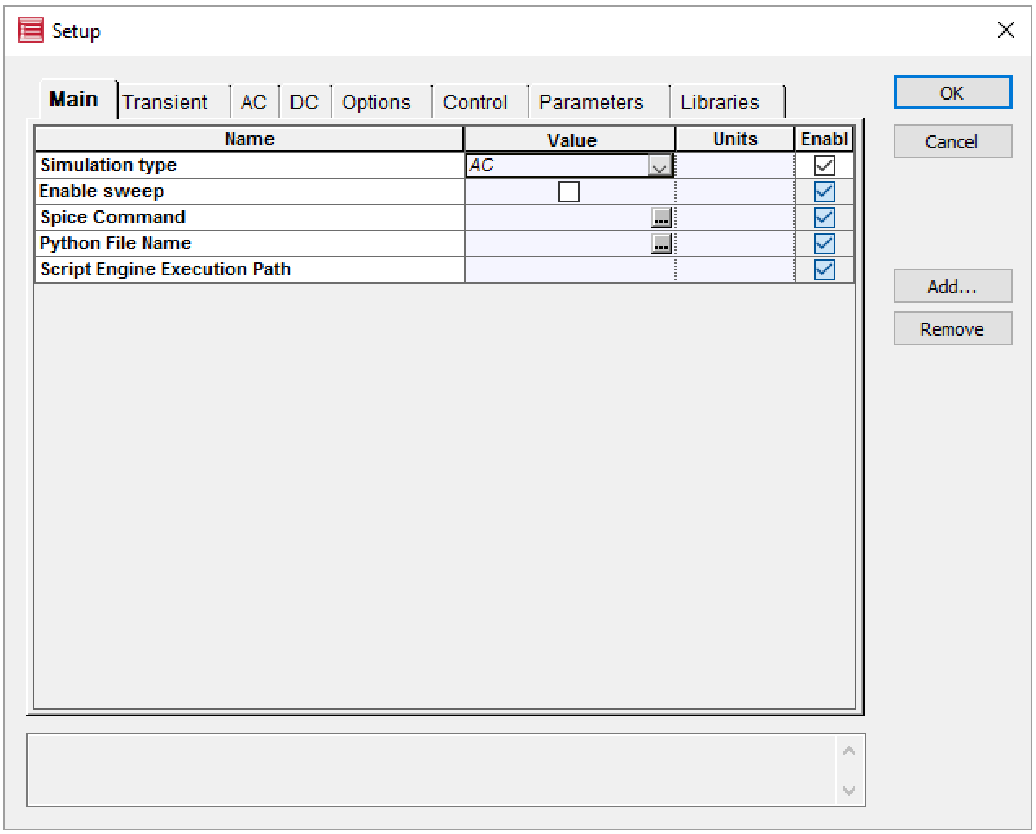


Setup AC analysis

Perform the following steps to setup the AC analysis and frequency sweep.

Step	Action
1	Select **Analysis > Setup**.
2	Select AC as the simulation type from the drop down menu

Figure 3 Enabling AC analysis

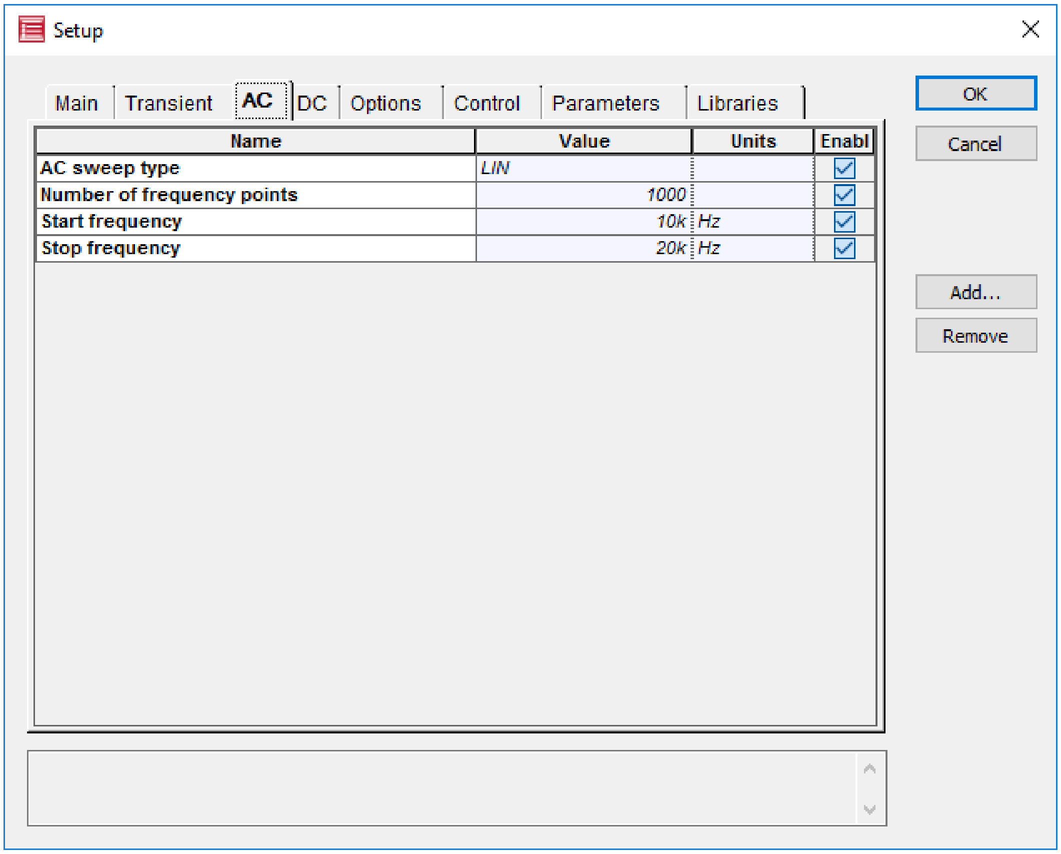


3	Click on the **AC** tab
4	In the **AC** tab, type the following values (also see Figure 4):

- **Number of frequency points**: 1000
- **Start frequency**: 10k
- **End frequency**: 20k

5 Click **OK**.

Figure 4 Entering AC sweep values



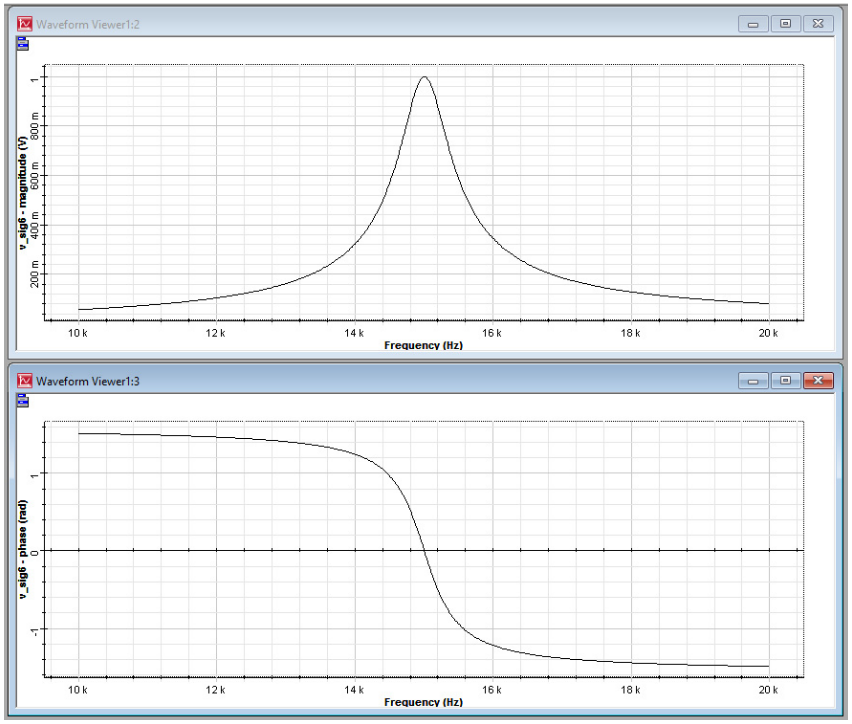
Running the simulation

Running the simulation is same as for transient analysis. Save the design and select **Analysis > Run**. Click on ***Launch Waveform Viewer*** once the simulation ends

Viewing results

After running the simulation, you can directly plot the results from the waveform viewer (see Figure 5)

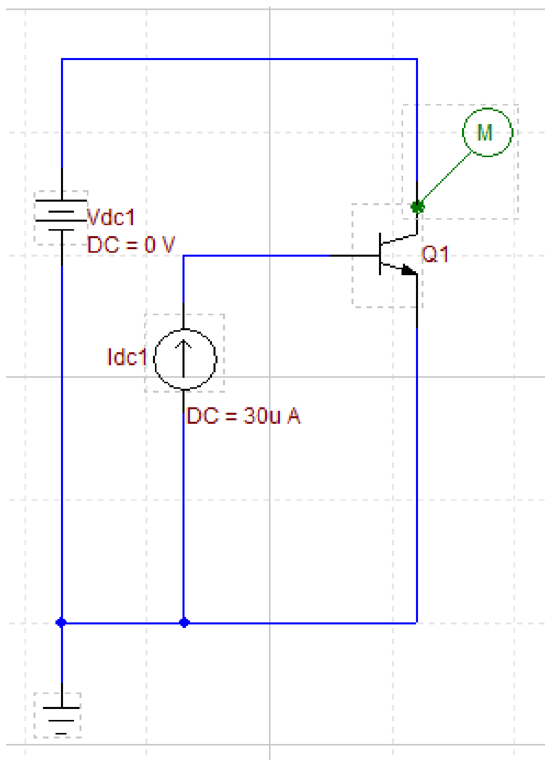
Figure 5 Results from Waveform Viewer



DC Analysis

This example demonstrates the steps involved in performing DC analysis for the MOSFET example shown in Figure 6. In this example, by performing the DC sweep of ***Vd***, you can plot the ***Vd*** vs. ***Id*** (drain current) graph.

Figure 6 Analysis for a HBT (Mextram 504 model)



Circuit design

Perform the following steps to design the circuit shown in Figure 6.

> ***Note:*** The OptiSPICE Schematic associated with this example can also be found under the folder: *OptiSPICE 5.2 Samples\Tutorials\Basic\DC Analysis*

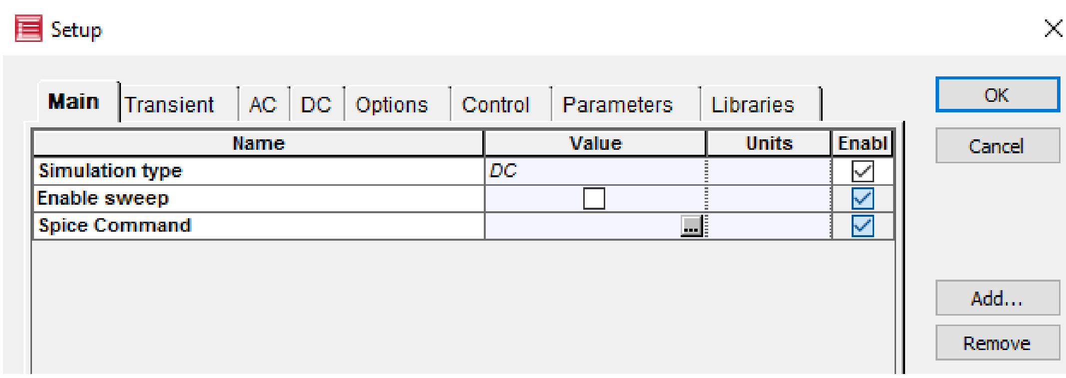
Step	Action
1	Place an NPN Mextram504 transistor, a DC current and a DC voltage source from the electrical library and connect them as shown.
2	Change the DC current value to 30 uA.
3	Place a probe on the pin of the Q1 as shown in Figure 6. Double click on the probe to make sure it measures the current

Setup DC analysis

Perform the following steps to setup the DC sweep.

Step	Action
1	Select **Analysis > Setup**.
2	Select **DC** as the simulation type from the drop down menu

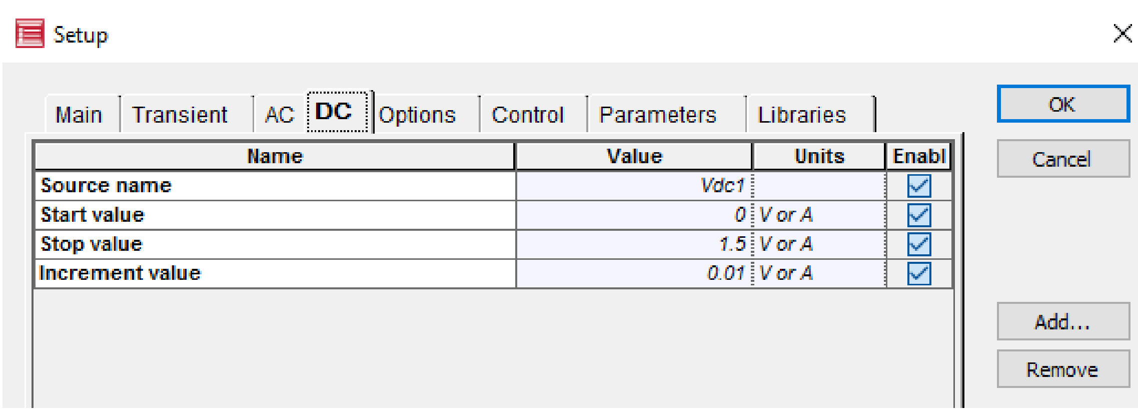
Figure 7   Enabling DC analysis



Step	Action
3	Click **DC** tab
4	In the **DC** tab, type the following values (also see Figure 8):

- **Source name**: Vd1
- **Start value**: 0
- **Stop value**: 1.5
- **Increment**: 0.01

Figure 8   Entering DC sweep values



Step	Action
5	Click **OK**.

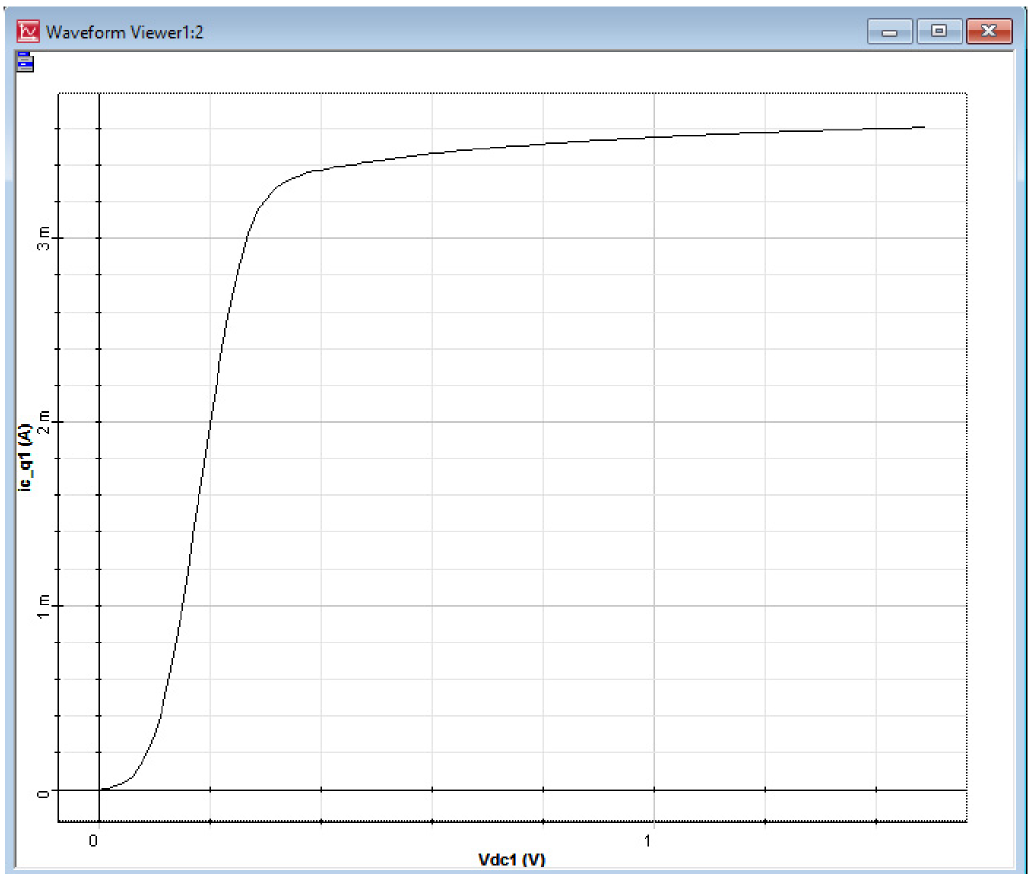
Running the simulation

Running simulation is same as for transient analysis. Save the design and select **Analysis > Run**. Click on ***Launch Waveform Viewer*** once the simulation ends.

Post processing

After running the simulation, you can directly plot the results from the waveform viewer (see Figure 9)

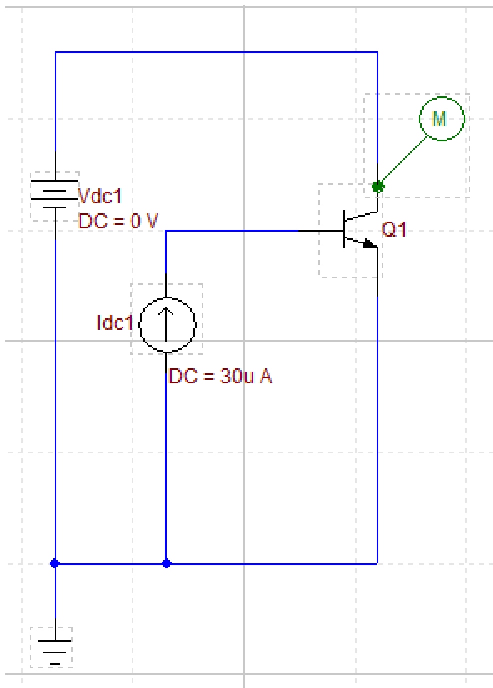
Figure 9 Simulation results from the Waveform Viewer



Parameter Sweep Analysis

This example demonstrates the steps involved in performing parameter sweep analysis for the circuit example shown in Figure 10. In this example, the DC current value is specified as a parameter and the simulation is performed over a range of DC current values. Each DC current value yields a separate simulation result.

Figure 10 Example circuit for parameter sweep



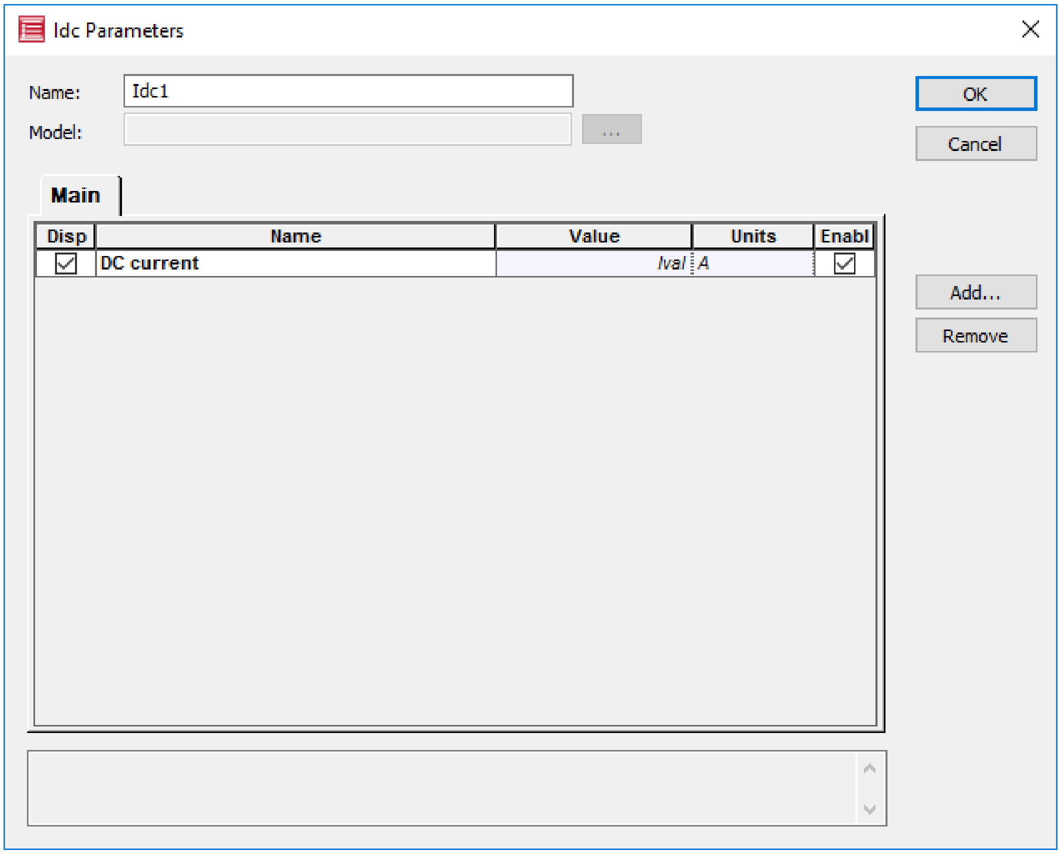
Circuit design

The OptiSPICE Schematic associated with this example can also be found within the folder: *OptiSPICE 5.2 Samples\Tutorials\Basic\Parameter Sweep*

Perform the following steps to parameterize the DC current value.

Step	Action
1	Double click on Idc1.
2	Enter ***Ival*** as the DC current value (see Figure 11)

Figure 11 Enter parameter for Idc

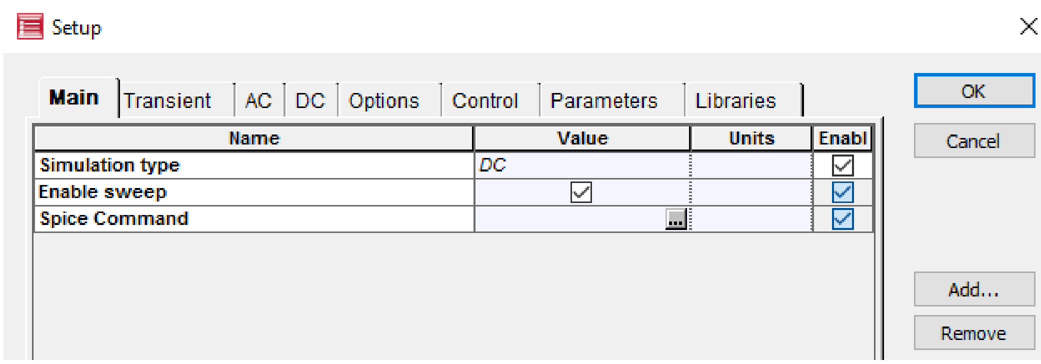


Setup parameter sweep

Perform the following steps to setup Parameter sweep.

Step	Action
1	Select **Analysis > Setup**.
2	Select the **Value** check box beside **Enable sweep** (see Figure 12)

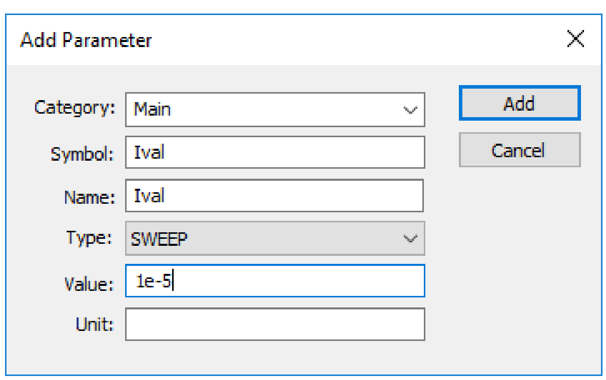
Figure 12 Enabling parameter sweep



3	Click on the **Parameters** tab
4	In the Parameters tab, click the **Add** button.
5	In **Add Parameter** dialog box, enter following as shown in Figure 13

- **Symbol**: Ival
- **Name**: Ival
- Select **Type**: **SWEEP**
- **Value**: 1e-5 (this value is the default value and will be used if sweep analysis is disabled)

Figure 13 Adding sweep parameter



6 Click **Add**.

7 Click the sweep icon in the value cell (see Figure 14). Clicking launches the **Parameter Sweep** dialog box (see Figure 15).

Figure 14 Sweep icon

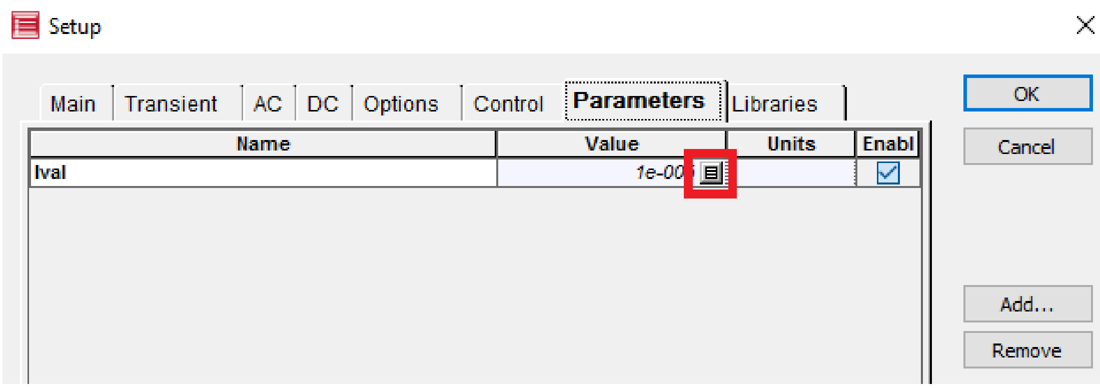
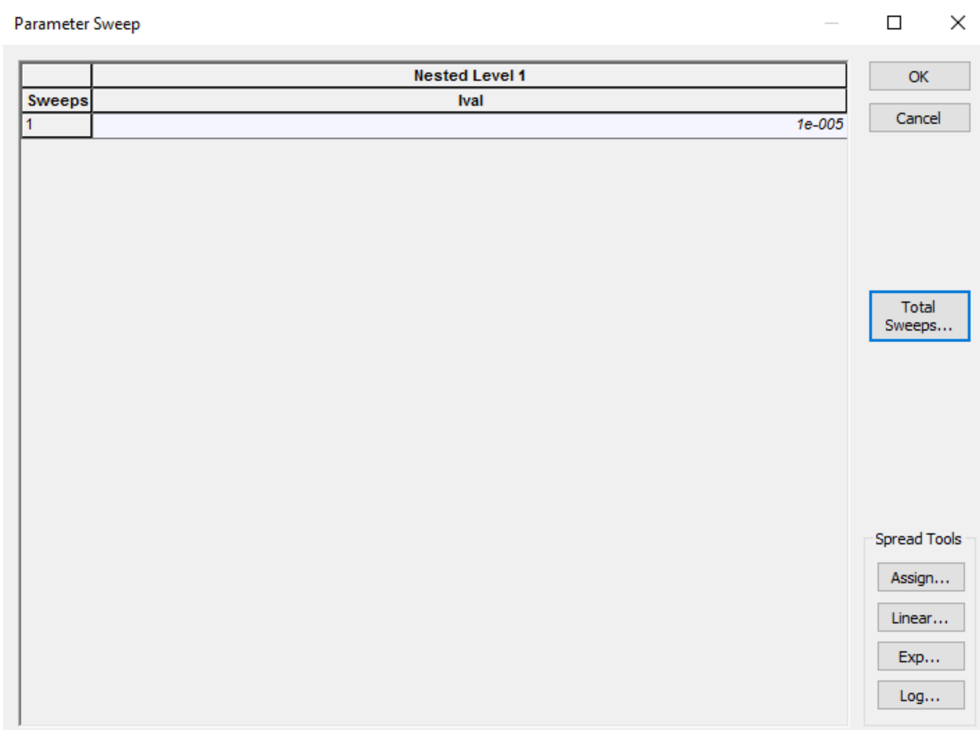


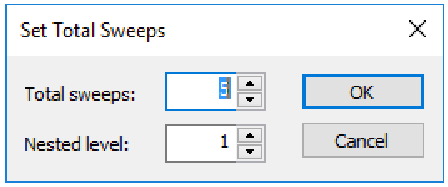
Figure 15 Parameter Sweep dialog box



8 In the **Parameter Sweep** dialog box, click **Total Sweeps.**

9 Enter **5** for **Total Sweeps** (see Figure 16)

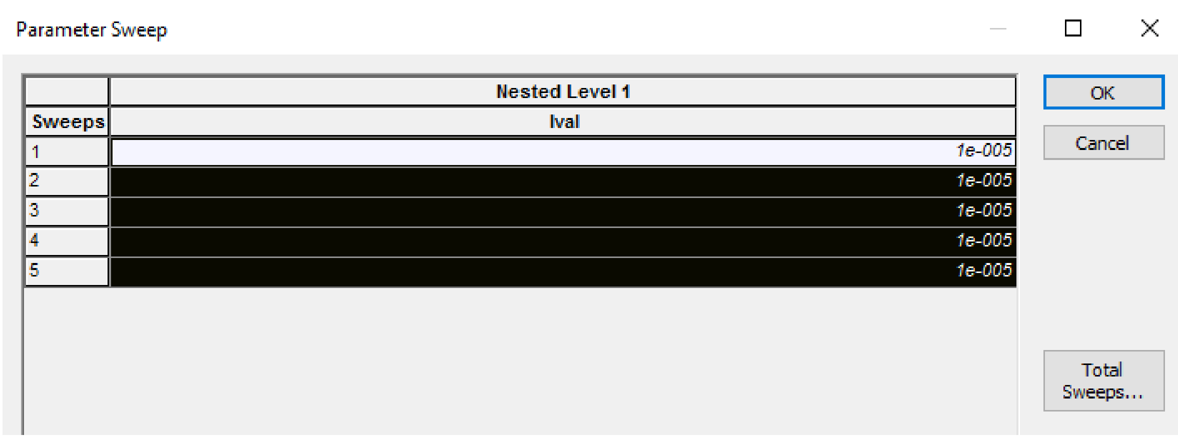
Figure 16 Setting total number of sweeps



10 Click **OK**.
This sets the five sweeps with the value 1e-5.

11 Select all five sweeps as shown in Figure 17

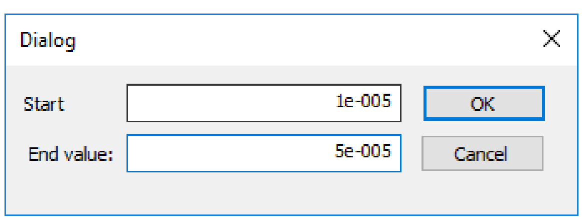
Figure 17 Sweep values set with initial values



12 Click **Linear**

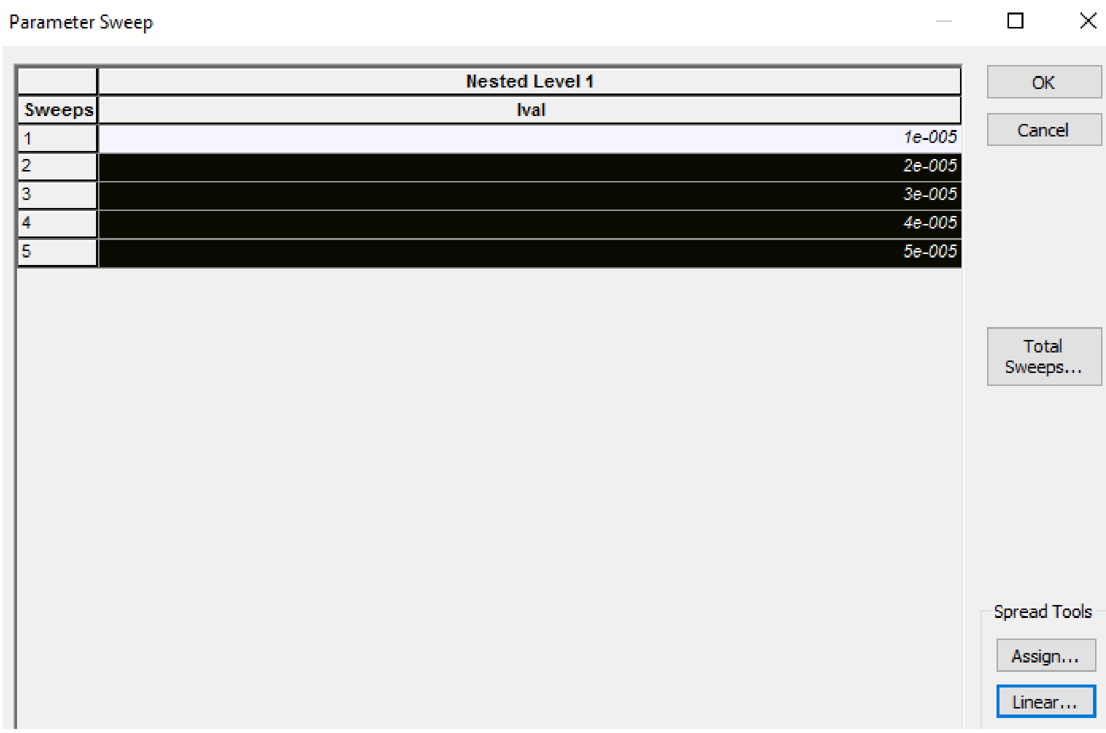
13 Enter **5e-5** as the **End value** (see Figure 18)

Figure 18 Entering linear variation



14 Click **OK**. Now the sweep values show the linear variation (see Figure 19)

Figure 19 Sweep values after linear variation is applied



15 Click **OK** on the Parameter Sweep dialog box.

16 Click **OK** on the Setup dialog box.

Running the simulation

Running simulation is same as for transient analysis. Save the design and select **Analysis > Run**. Click on ***Launch Waveform Viewer*** once the simulation ends.

Visualizing results

Perform the following steps to visualize multiple graphs in the same plot window.

- Double click on “ic_q1” on the left panel in the waveform viewer (see Figure 20)
- Drag on drop each “ic_q1” 2D icon onto the same graph (see Figure 21).

Figure 20 Simulation results

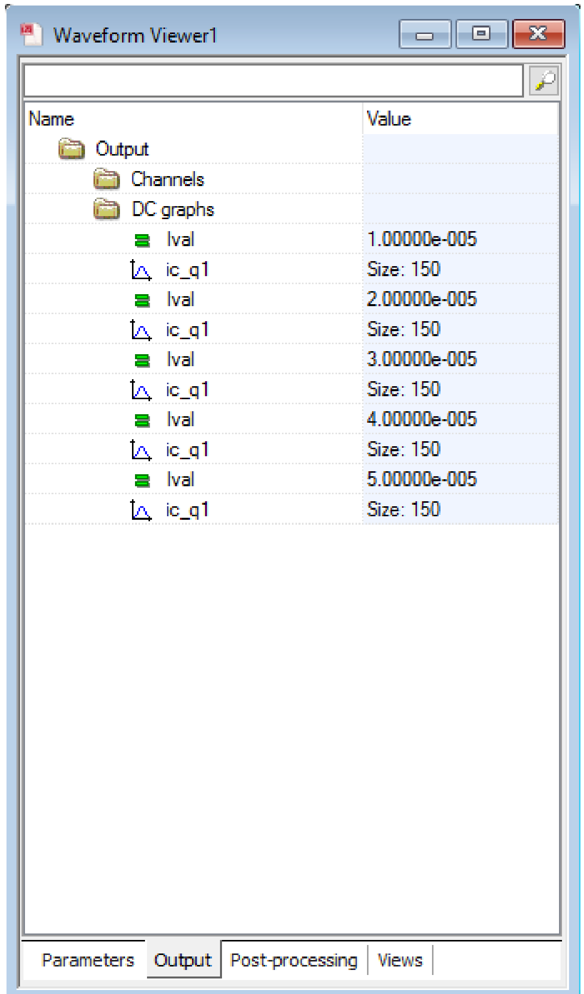
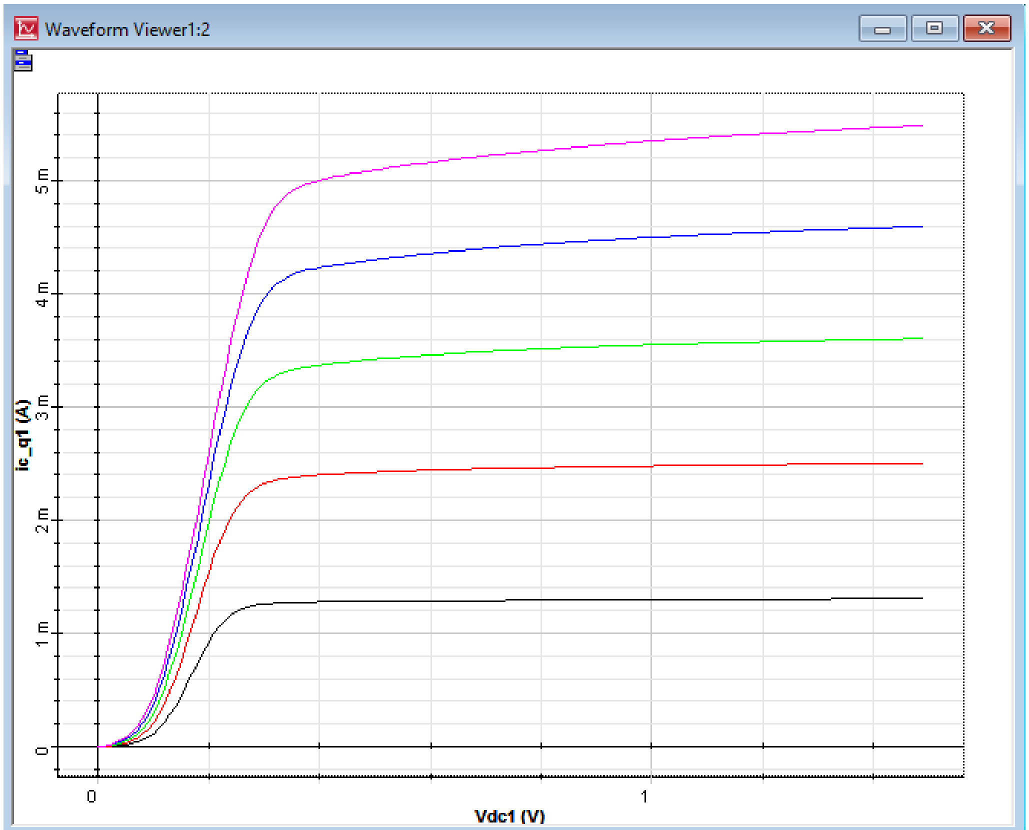


Figure 21 Parameter sweep of Ic vs Vdc



Wavelength Sweep Analysis

Wavelength sweep can be used to determine the frequency response of an optical circuit by sweeping the wavelength of the laser.

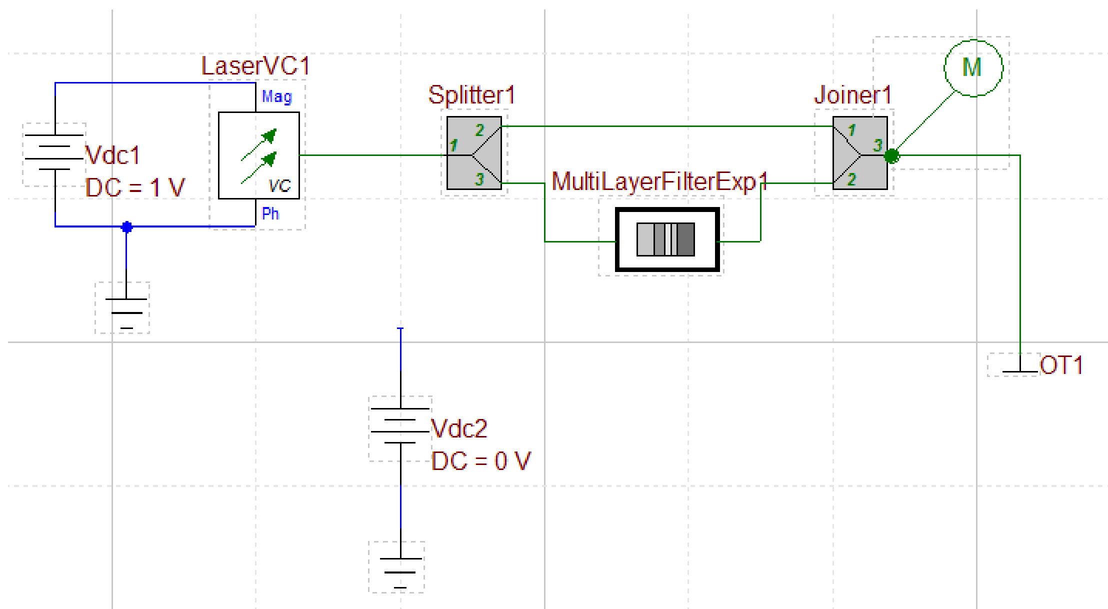
Note: The OptiSPICE Schematic associated with this example can also be found within the folder: *OptiSPICE 5.2 Samples\Tutorials\Basic\Wavelength Sweep*

Circuit design and set up

Perform the following steps to setup wavelength sweep

Step	Action
1	Drag and drop components and connect them as shown in Figure 22

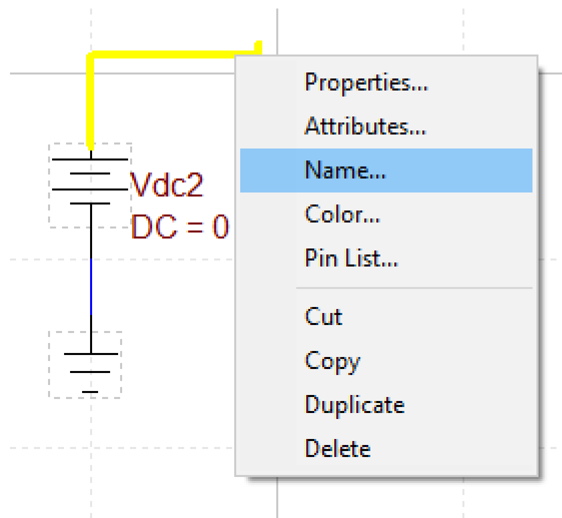
Figure 22 Example circuit for wavelength parameter sweep



2 Set the DC voltage of ***Vdc1*** to 1 V.

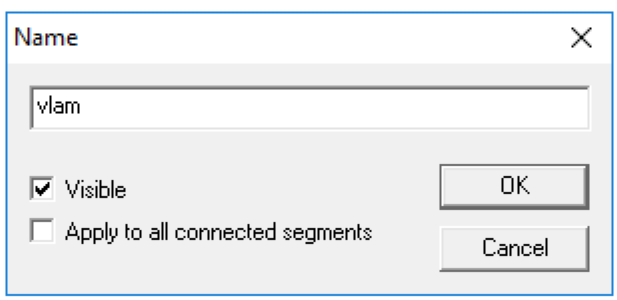
3 Right click on the signal of ***Vdc2*** and select name from the menu (see Figure 23)

Figure 23 Signal menu (connected to Vdc2)



4 Rename the signal as ***vlam*** (see Figure 24)

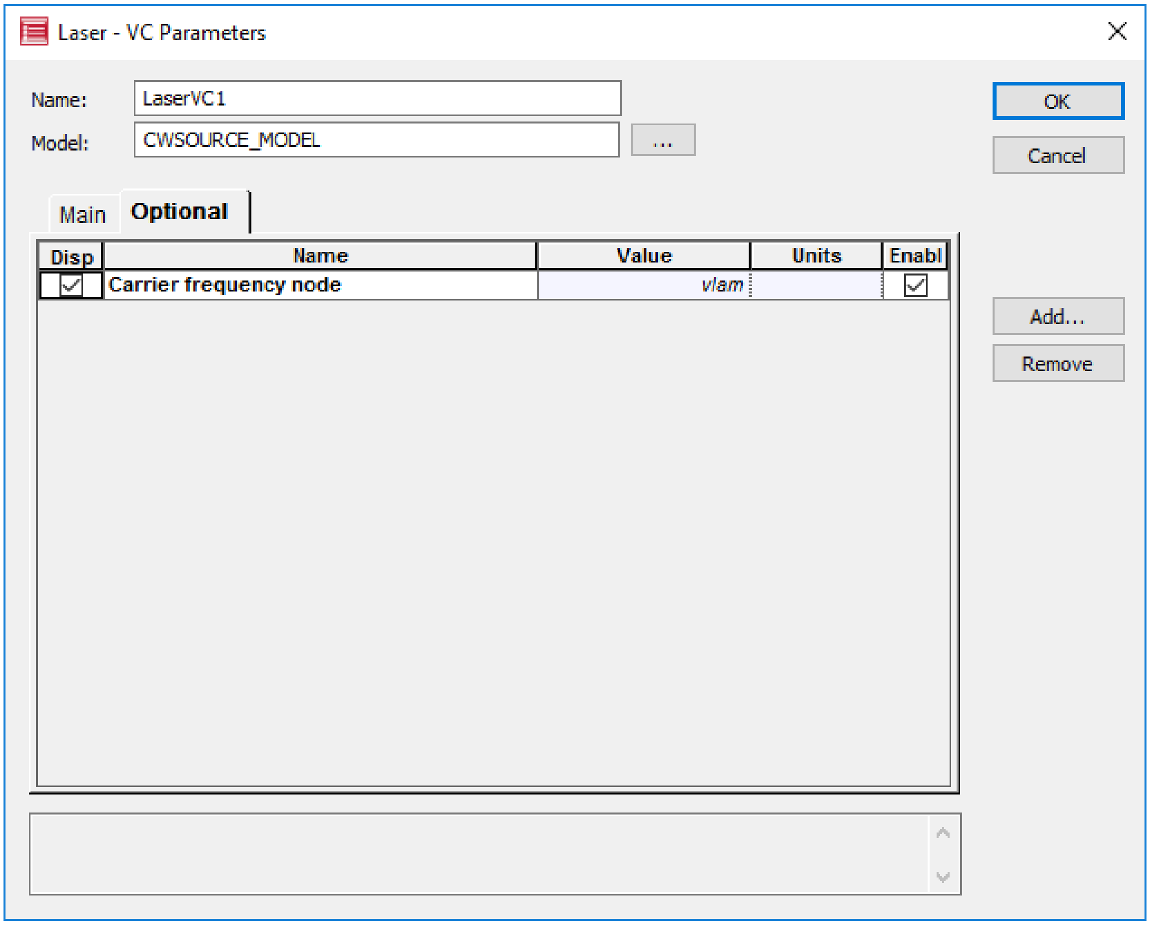
Figure 24 Changing the signal name



5 Double click on the LaserVC1 and select Optional tab

6 Enable ***Carrier frequency*** node and enter *vlam* as a value (see Figure 25)
Now the voltage of Vdc2 controls the wavelength of the laser.

Figure 25 Setting up the carrier frequency node

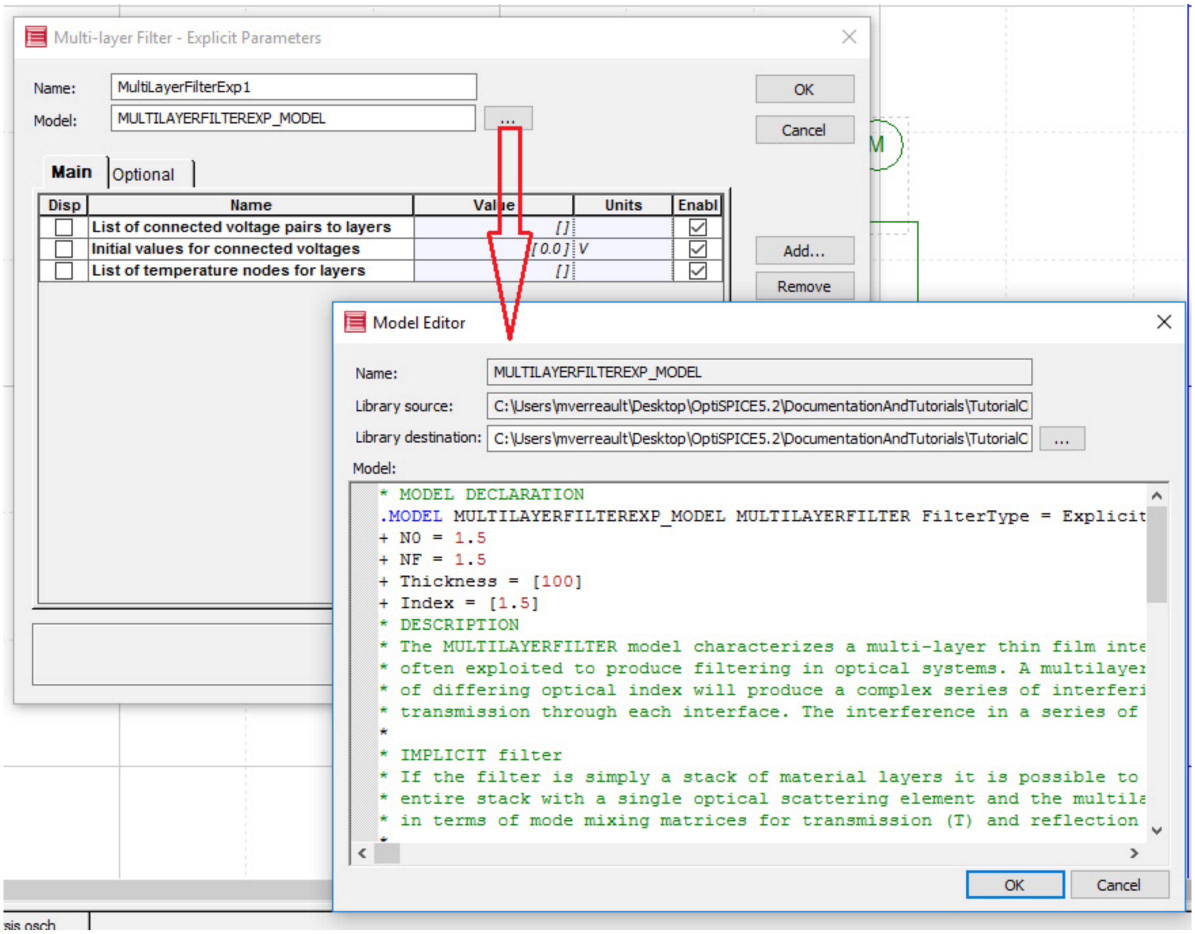


7 Double click on the ***MultiLayerFilterExp1*** and open the model editor (see Figure 26).

8 Enter the following values as model parameters and insert '**+**' for each new line (see Figure 26):

- **N0** = 1.5
- **NF** = 1.5
- **Thickness** = [100]
- **Index**: [1.5]

Figure 26 Adding model parameters to explicit multilayer filter

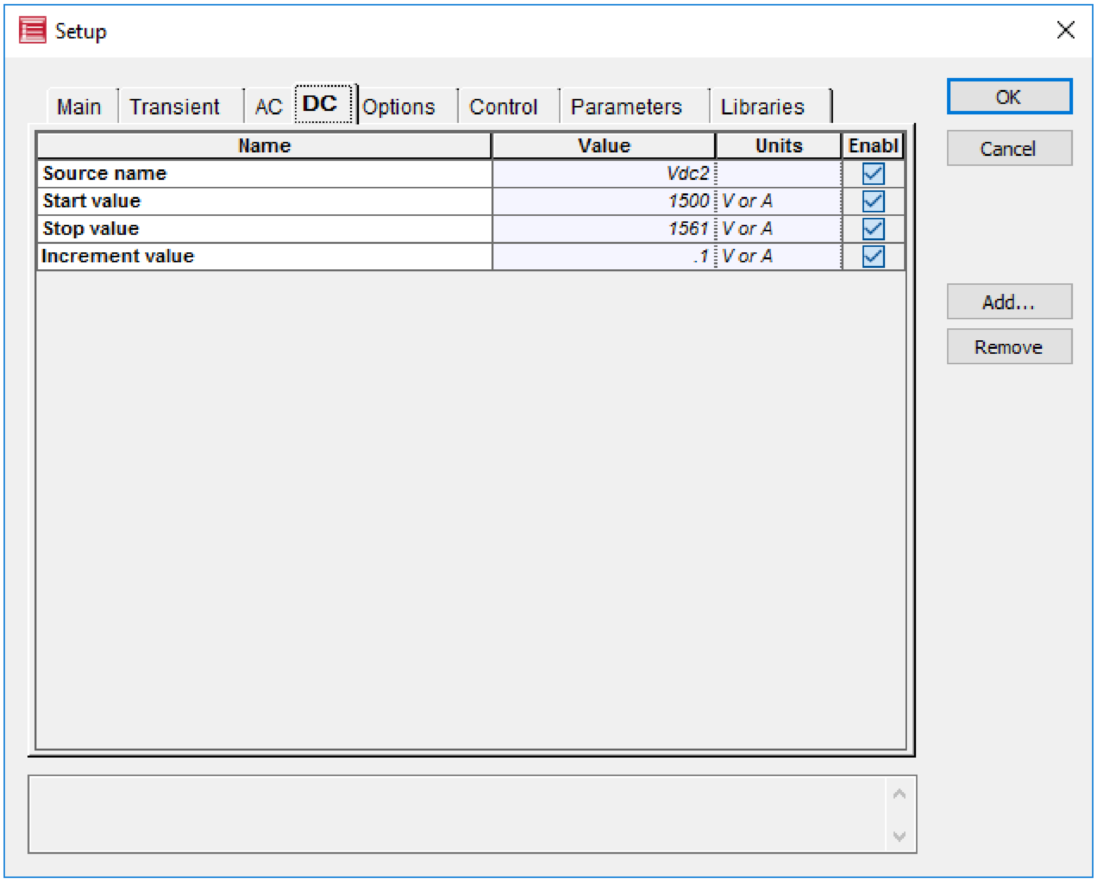


9 Select DC as a simulation type from **Analysis/Setup**

10 Select the **DC** tab and enter the following values (see Figure 27)

- **Source name** = Vdc2
- **Start value** = 1500
- **Stop value** = 1561
- **Increment value**: 0.1

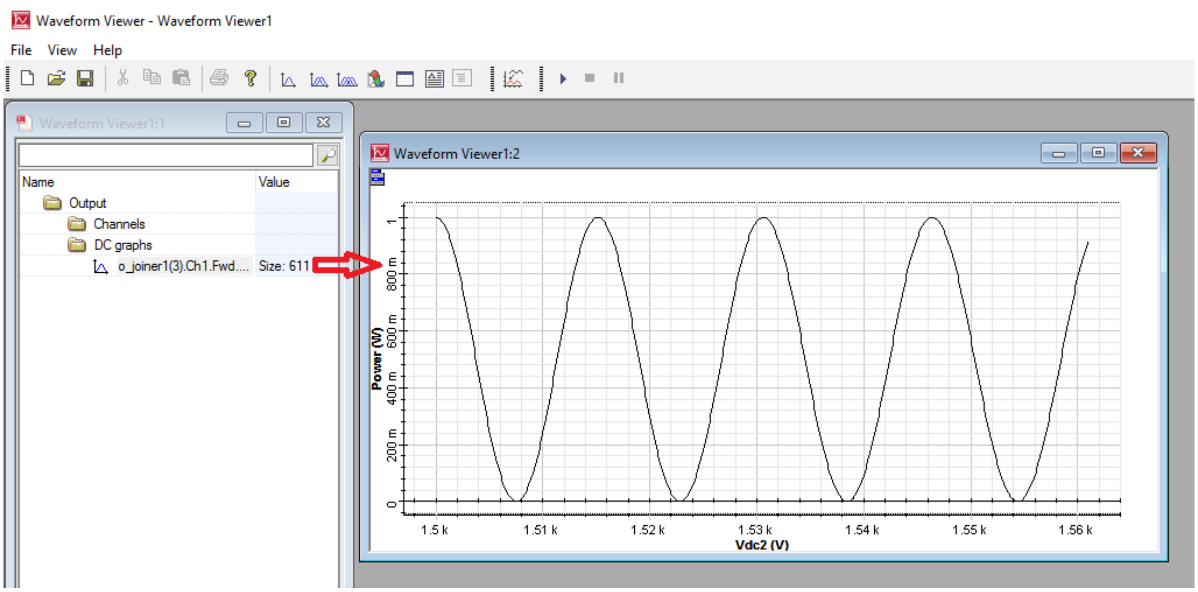
Figure 27 Setting up the wavelength sweep parameters



Running the simulation and visualizing results

Running simulation is same as for transient analysis. Save the design and select **Analysis > Run**. Click on ***Launch Waveform Viewer*** once the simulation ends. Figure 28 shows the output of Joiner1 vs. wavelength

Figure 28 Simulation results (Wavelength Sweep Analysis)



Transient and Noise Simulation

The following example shows how to run a transient simulation of an electrical circuit that includes electrical noise.

> ***Note:*** The OptiSPICE Schematic associated with this example can also be found within the folder: *OptiSPICE 5.2 Samples\Tutorials\Basic\Transient*

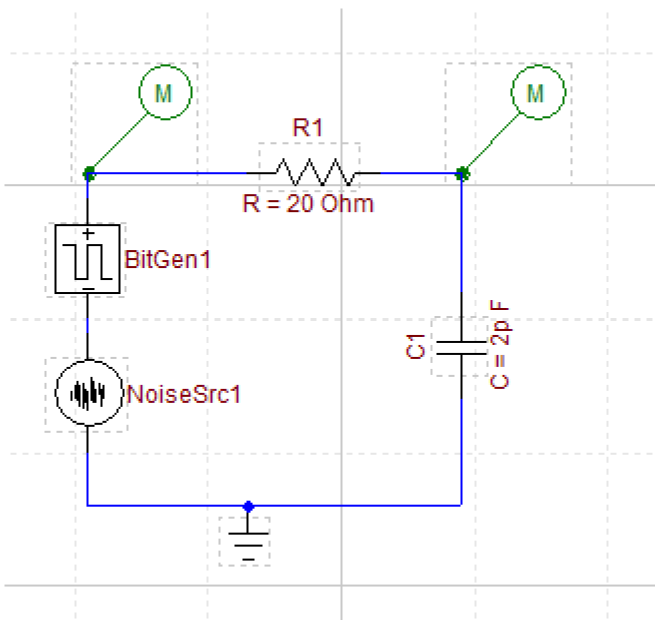
Circuit design and set up

Perform the following steps to setup wavelength sweep

Step **Action**

1 Drag and drop components and connect them as shown in Figure 29

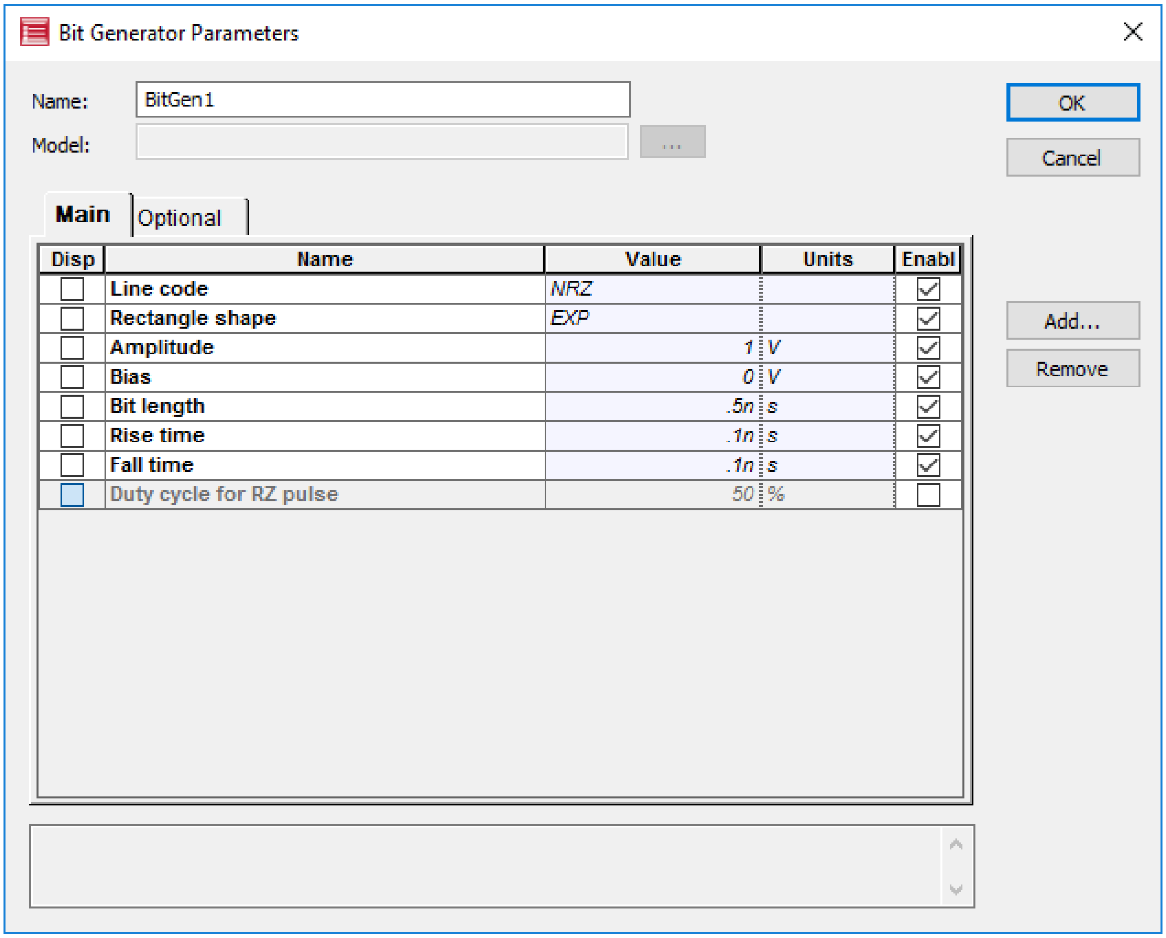
Figure 29 Example RC filter



2 Set up the following parameters for BitGen1 (see Figure 30):

- **Rise time** = 0.1n
- **Fall time** = 0.1n
- **Bit length** =0.5n

Figure 30 Signal setup for BitGen1



3 Set **R1** to 20 ohms and **C1** to 2 pF

4 Enter the following parameter values for the noise source (see Figure 31)

a. **Noise source type**: V

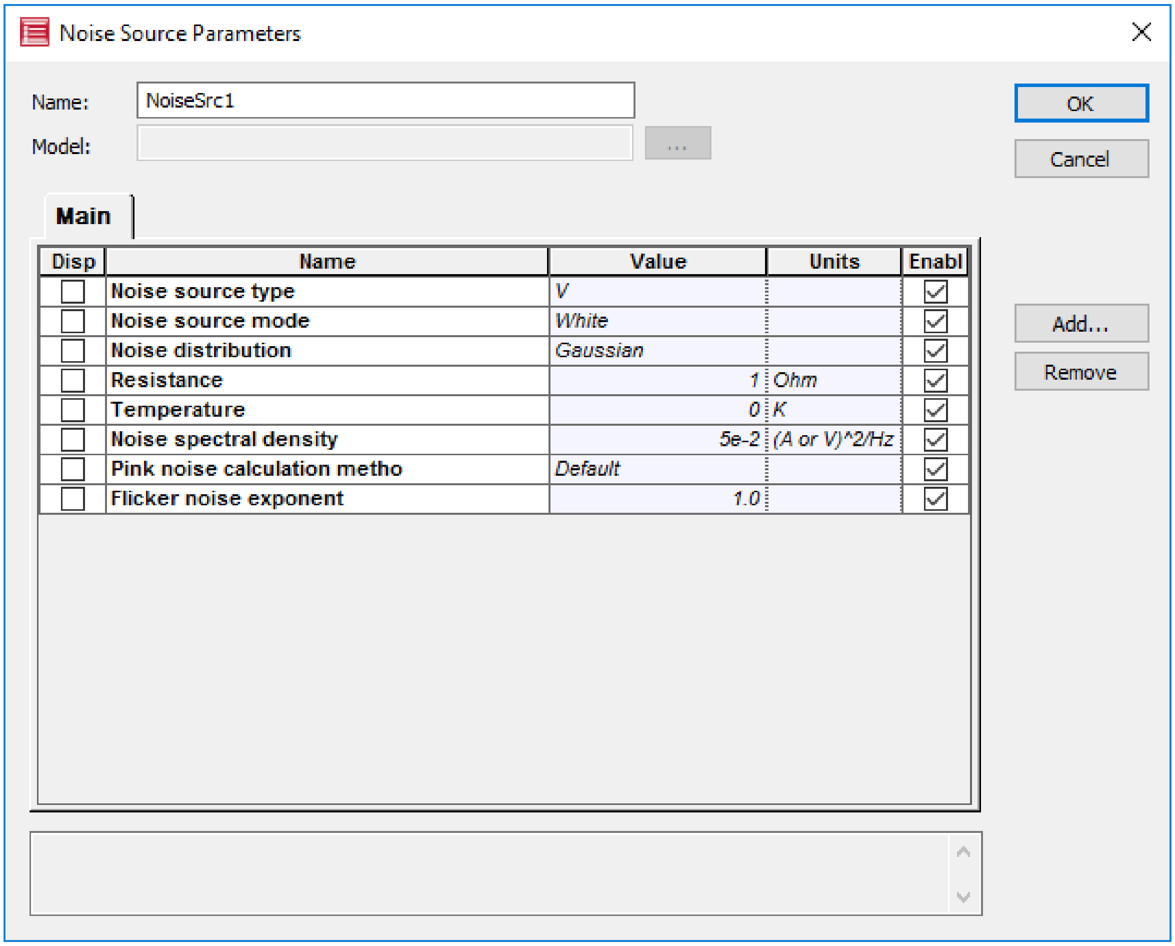
b. **Noise source mode**: White

c. **Noise source distribution**: Gaussian

d. **Resistance**: 1 ohm

e. **Noise spectral density**: 5e-2

Figure 31 Setting up the noise source

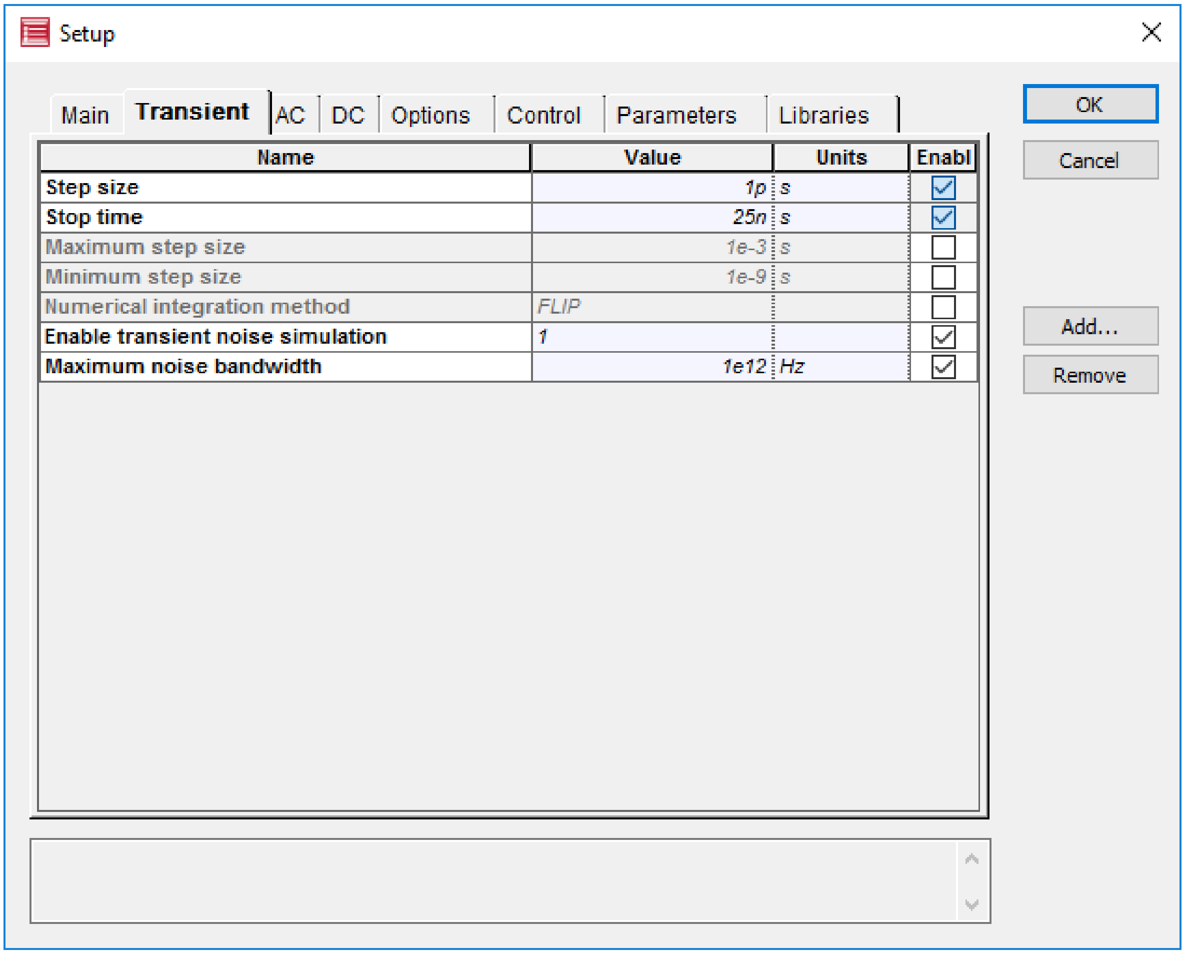


5 Go to **Analysis/Setup** and set up the simulation type as *Transient*

6 Go to the **Transient** tab and set the **Stop time** to 25 n, enable **Transient noise simulation** (set field to 1) and set the **Maximum noise bandwidth** to

1e12 Hz (see Figure 32)

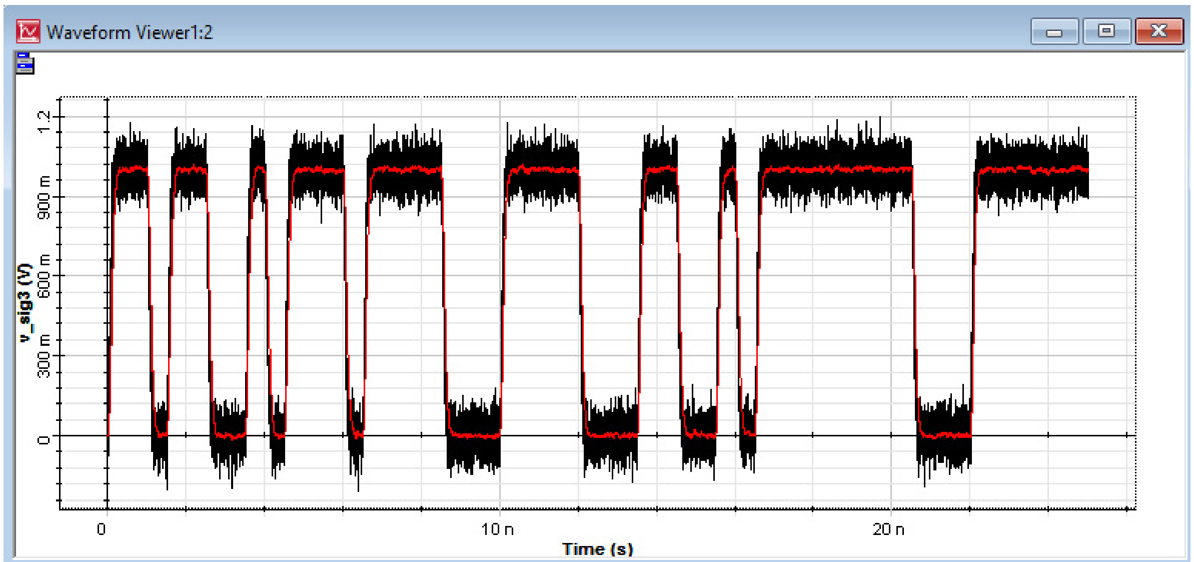
Figure 32 Transient simulation parameters



Running the simulation and visualizing results

Save the design and select **Analysis > Run**. Click on ***Launch Waveform Viewer*** once the simulation ends. Figure 33 shows the noise of the signal being filtered by RC the circuit (v_sig3 in RED) and the noise before filtering (v_sig7 in BLACK)

Figure 33 Simulation results (Transient and Noise Analysis)



Optiwave
7 Capella Court
Ottawa, Ontario, K2E 7X1, Canada

Tel.: 1.613.224.4700
Fax: 1.613.224.4706

E-mail: support@optiwave.com
URL: www.optiwave.com