## **OptiSPICE** Tutorials - Basic

Opto-Electronic Circuit Design Software

Version 5.2



# OptiSPICE

## Tutorials - Basic

**Opto-Electronic Circuit Design Software** 

#### Copyright © 2016 Optiwave

All rights reserved.

All OptiSPICE documents, including this one, and the information contained therein, is copyright material.

No part of this document may be reproduced, stored in a retrieval system or transmitted in any form or by any means whatsoever, including recording, photocopying, faxing, etc., without prior written approval of Optiwave.

#### Disclaimer

Optiwave makes no representation or warranty with respect to the adequacy of this documentation or the programs which it describes for any particular purpose or with respect to its adequacy to produce any particular result. In no event shall Optiwave, its employees, its contractors, or the authors of this documentation be liable for special, direct, indirect, or consequential damages, losses, costs, charges, claims, demands, or claim for lost profits, fees, or expenses of any nature or kind.

### **Table of Contents**

Basic Tutoria	als	1
AC An	alysis	1
	Circuit design	1
	Setup AC analysis	3
	Running the simulation	4
	Viewing results	4
DC An	alysis	6
	Circuit design	6
	Setup DC analysis	7
	Running the simulation	8
	Post processing	8
Param	eter Sweep Analysis	9
	Circuit design	9
	Setup parameter sweep	.11
	Running the simulation	.14
	Visualizing results	.14
Wavele	ength Sweep Analysis	.17
	Circuit design and set up	.17
	Running the simulation and visualizing results	.22
Transie	ent and Noise Simulation	.23
	Circuit design and set up	.23
	Running the simulation and visualizing results	.27

#### **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.

📕 Vac Par	ameters			×
Name: Model: <b>Main</b>	Vac1			OK Cancel
	Name C magnitude C phase	Units V Deg.	Enabl	Add Remove
			Ŷ	

Figure 2 AC source values

#### Setup AC analysis

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

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

N	Control Parameters	Libraries	· · · · ·	
Name Simulation type	AC Value	Units	Enabl	Cancel
Enable sweep				
Spice Command				
Python File Name		_		
Script Engine Execution Path				Add
				Remove

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.

Setup				>
Main Transient AC DC Options C		Libraries		ОК
Name	Value	Units	Enabl	Cancel
AC sweep type	LIN 1000			
Number of frequency points	1000	-		
Start frequency Stop frequency	20k	-		
				Add Remove
			$\sim$	

#### 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

Ē	Setup										$\times$
	Main	Transient	AC	DC Options	Co	ontrol	Parameters	Libraries	)		ОК
			Name				Value	Units	Enabl		Cancel
- [3	Simulat	ion type				DC					
E	inable a	sweep									
	Spice C	ommand									
											Add
										F	Remove

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

🗮 Setup				×
Main Transient AC DC Options Co	ontrol Parameters	Libraries ]		ОК
Name	Value	Units	Enabl	Cancel
Source name	Vdc1			
Start value	0	V or A		
Stop value	1.5	V or A		
Increment value	0.01	V or A		
				Add Remove

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

📕 Idc Parameters			×
Name: Idc1 Model: Main			OK Cancel
Disp Name          Disp       DC current	Value Unit	s Enabl	Add Remove

#### Setup parameter sweep

Perform the following steps to setup Parameter sweep.

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

#### Figure 12 Enabling parameter sweep

E	Setup						×
	Main Transient	AC DC Options	Control Pa	ameters	Libraries		ОК
		Name	Va	lue	Units	Enabl	Cancel
	Simulation type		DC				
	Enable sweep		5				
	Spice Command						
							Add
							Remove

- 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
-----------	--------------	-----------

Add Parame	eter	×
Category:	Main	Add
Symbol:	Ival	Cancel
Name:	Ival	
Type:	SWEEP ~	
Value:	1e-5	
Unit:		



- 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).

Ē	Setup									×
	Main T	ransient	ACDC	Options	Co	ntrol	Parameters	Libraries	1,	ОК
			Name				Value	Units	Enabl	Cancel
	lval						1e-00 🛯			
										Add
										Remove

#### Figure 14 Sweep icon



	Nested Level 1		OK
Sweeps	Ival		
1		1e-005	Cancel
			Tabal
			Total Sweeps
			Spread Too
			Assign
			Linear
			Exp



- 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

Set Total Sweeps	×
Total sweeps:	OK
Nested level: 1	Cancel

- **10** Click **OK**. This sets the five sweeps with the value 1e-5.
- 11 Select all five sweeps as shown in Figure 17



Parameter	Sweep —	
	Nested Level 1	OK
Sweeps	Ival	
1	1e-005	Cancel
2	1e-005	
3	1e-005	
4	1e-005	
5	1e-005	
		Total Sweeps

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

#### Figure 18 Entering linear variation

Dialog		×
Start	1e-005	ОК
End value:	5e-005	Cancel



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

Parameter Sweep			
	Nested Level 1		ОК
Sweeps	Ival		
1		1e-005	Cancel
2		2e-005	
3		3e-005	
4		4e-005	
5		5e-005	
			Total Sweeps
			Spread Tools Assign Linear

#### 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).



Name	Value
Cutput	
Channels	
DC graphs	
a Ival	1.00000e-005
[∧_ ic_q1	Size: 150
a Ival	2.00000e-005
[∧_ic_q1	Size: 150
😑 Ival	3.00000e-005
[∆_ ic_q1	Size: 150
a Ival	4.00000e-005
📐 ic_q1	Size: 150
a Ival	5.00000e-005
📐 ic_q1	Size: 150

#### 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

Name	×
vlam	
Visible	OK
Apply to all connected segments	Cancel

- 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.



📕 Laser -	VC Parameters				×
Name: Model:	LaserVC1 CWSOURCE_MODEL				OK Cancel
Main Disp	Optional Name	Value	Units	Enabl	
	arrier frequency node	vlam			Add Remove
				< >	

#### 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]



ame:	MultiLayerFilterExp1	ОК
odel:	MULTILAYERFILTEREXP_MODEL	Cancel
Main	Optional	
Disp	Name	Value Units Enabl
	List of connected voltage pairs to	
	Initial values for connected volta	
	List of temperature nodes for la	yers [] Remove
		Model Editor
		Name: MULTILAYERFILTEREXP MODEL
		Name: MULTILAYERFILTEREXP_MODEL
		Library source: C:\Users\mverreault\Desktop\OptiSPICE5.2\DocumentationAndTutorials\TutorialC
		Library destination: C:\Users\mverreault\Desktop\OptiSPICE5.2\DocumentationAndTutorials\TutorialC
		Model:
		* MODEL DECLARATION
		.MODEL MULTILAYERFILTEREXP MODEL MULTILAYERFILTER FilterType = Explicit
		+ NO = 1.5
		+ NF = 1.5
		+ Thickness = [100]
		+ Index = [1.5] * DESCRIPTION
		* The MULTILAYERFILTER model characterizes a multi-layer thin film inte
		* often exploited to produce filtering in optical systems. A multilayer
		* of differing optical index will produce a complex series of interferi
		* transmission through each interface. The interference in a series of
		* * IMPLICIT filter
		* If the filter is simply a stack of material layers it is possible to
		* entire stack with a single optical scattering element and the multila
		* in terms of mode mixing matrices for transmission (T) and reflection
		* < >

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



Setup				×
Main Transient AC DC Options C	ontrol Parameters	Libraries	<u> </u>	ОК
Name	Value	Units	Enabl	Cancel
Source name	Vdc2			
Start value		V or A		
Stop value		V or A	$\overline{\mathbf{v}}$	
Increment value	.1	V or A		
				Add
				Remove
			~	
			$\sim$	

#### 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





#### **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

- Set up the following parameters for BitGen1 (see Figure 30):
  - Rise time = 0.1n
  - Fall time = 0.1n

2

• Bit length =0.5n



odel:					Cancel
<b>Main</b> Disp	Optional Name	Value	Units	Enabl	
	Line code	NRZ			
	Rectangle shape	EXP			Add
	Amplitude		V		Demous
	Bias		V		Remove
<u>Ц</u>	Bit length	.5n			
	Rise time	.1n			
Ц	Fall time	.1n			
	Duty cycle for RZ pulse	50	%		

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



📕 Nois	e Source Parameters				×
Name: Model: Main	NoiseSrc1				OK Cancel
Disp	Name	Value	Units	Enabl	
	Noise source type	V			
	Noise source mode	White			Add
	Noise distribution	Gaussian			
	Resistance		Ohm		Remove
	Temperature		К		
	Noise spectral density		(A or V)^2/Hz		
	Pink noise calculation metho	Default	-		
	Flicker noise exponent	1.0			
				$\hat{}$	

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)

E Setup				×
Main Transient AC DC Options Co			1	OK
Name	Value	Units	Enabl	Cancel
Step size	1p			
Stop time	25n			
Maximum step size	1e-3		부님네	
Minimum step size	1e-9	S		
Numerical integration method	FLIP			Add
Enable transient noise simulation Maximum noise bandwidth	1			,
Maximum noise bandwidth	1e12	HZ		Remove
			Ŷ	

#### 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)







**BASIC TUTORIALS** 





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