OptiSPICE

Getting Started

Opto-Electronic Circuit Design Software

Version 5.2

OptiSPICE

Getting Started

Opto-Electronic Circuit Design Software

Table of Contents

Installing OptiSPICE

Before installing OptiSPICE, ensure the system requirements described below are available.

Hardware and software requirements

OptiSPICE requires the following minimum system configuration:

- Minimum PC configuration: PC with Pentium 4 processor or equivalent with 2 GB RAM.
- Recommended PC configuration: PC with a clock speed > 2 GHz with 2-4 cores (e.g. Intel i7 3rd/4th Gen, AMD Athlon/Athlon II) and 8 GB RAM.
- Operating Systems: Microsoft Windows 7/8/10 (64-bit).
- 400 MB free hard disk space
- 1024 x 768 graphic resolution, minimum 65536 colors
- Internet Explorer 5.5 or higher (to enable VBScript functionality)

Protection key

A hardware protection key is supplied with the software.

> ***Note:*** Please ensure that the hardware protection key is NOT connected during the installation of OptiSPICE.

To ensure that OptiSPICE operates properly, verify the following:

- The protection key is properly connected to the parallel/USB port of the computer.
- If you use more than one protection key, ensure that there is no conflict between the OptiSPICE protection key and the other keys.

> ***Note:*** Use a switch box to prevent protection key conflicts. Ensure that the cable between the switch box and the computer is a maximum of one meter long.

OptiSPICE directory

By default, the OptiSPICE installer creates an OptiSPICE directory on your hard disk. The OptiSPICE directory contains the following subdirectories:

- **\Bin** — executable files, dynamic linked libraries, and help files
- **Libraries** — OptiSPICE symbol library files
- **Documentation** — OptiSPICE documentation files
- **\Models** — OptiSPICE device model library files
- **\Samples** — OptiSPICE example files
- **\Templates** — OptiSPICE template files
- **\Tools** — OptiSPICE tools
- **\Utility Scripts** — OptiSPICE utility scripts

Installation

We recommend that you exit all Windows programs before running the setup program. To install OptiSPICE, perform the following procedure.

Step	Action
1	Log on as the Administrator, or log onto an account with Administrator privileges.
2	Insert the OptiSPICE CD into your CD ROM drive (if you have received your version of OptiSPICE via CD shipment)
3	Run setup.exe located in folder “OptiSPICE 5 Setup” on the installation CD” **Note:** If the software was provided in electronic (downloaded) format, run the downloaded executable file after saving it to your local disk.
4	Click **OK** and follow the on screen instructions and prompts.
5	When the installation is complete, reboot your computer.

Introduction

OptiSPICE is the first circuit design software for analysis of integrated circuits including interactions of optical and electronic components. It allows for the design and simulation of optoelectronic circuits at the transistor level, from laser drivers to trans-impedance amplifiers, optical interconnects and electronic equalizers. OptiSPICE produces self-consistent solutions of optoelectronic circuits that contain feedback spanning both optical and electrical parts. OptiSPICE is a fully-integrated solution for parameter extraction, schematic capture, circuit simulation and waveform analysis.

Description

With the imminent coexistence of electrical and optical components at the chip and board level, it is important to provide designers with a reliable simulation framework that can accurately and efficiently predict signal behavior in optoelectronic integrated circuits and boards.

Simulation of optical and electrical components simultaneously is a multi-disciplinary problem. Equations governing optical and electrical components are incompatible for co-simulation within traditional numerical engines. In addition, optoelectronic components are temperature dependent and this dependence needs to be incorporated into the simulation to obtain correct results. Temperature effects are particularly important for the simulation of optical devices integrated on a common substrate with high power electrical components such as laser drivers. In this case the close proximity of the electronics and optical elements such as lasers produces significant thermal coupling. The coupling of the electrical, optical and thermal domains in such a situation requires a self-consistent solution.

OptiSPICE is the only circuit design software for the self-consistent simulation of optical, electrical and thermal energy domains. The optical elements are represented by delay-differential equations, the electrical circuit by algebraic differential equations and the thermal circuit by a set of first order nonlinear heat diffusion equations. Supporting a wide variety of electrical circuit elements such as diodes, transistors, BJTs and MOSFETS along with optical components such as laser diodes, optical fibers and photo-diodes; OptiSPICE provides transient time domain, small-signal frequency, and noise analysis.

Benefits

- Significantly reduce product development costs and boost productivity through OptiSPICE comprehensive design environment to simulate optical and electrical circuits in one simulation engine
- Run state-of-the-art transient time domain, small-signal frequency, and noise analysis to accurately predict behavior of advanced optoelectronic circuits
- OptiSPICE Schematics offers direct schematic entry in an intuitive graphical user interface. It allows for greater ease of schematic capture, parameter specification, waveform probing and usage
- Waveform analysis and post-processing using the OptiSPICE Waveform Viewer, OptiSPICE Python interface, or OptiSystem advanced visualization tools (OSA spectra, signal chirp, eye diagrams, polarization state, constellation diagrams and much more).
- Includes parameter extraction tools for OptiSPICE model creation. From measurement data, parameter extractors are used to find the best set of OptiSPICE model parameters to fit the measurement

Applications

OptiSPICE supports a wide range of applications including:

- Design and simulation of optoelectronic circuits at the transistor level, from laser drivers to trans-impedance amplifiers, optical interconnects and electronic equalizers.
- Signal integrity analysis of optoelectronic circuits, including eye diagram analysis with BER patterns.

Main features

The main features of the OptiSPICE include:

Feature	Description
Device Symbol Editor	Integrated Device Symbol Editor allows you to create custom symbols for devices or hierarchical blocks using standard drawing tools.
Hierarchical Design	Hierarchical Design with unlimited levels is fully supported. Any symbol on a schematic can contain another schematic of arbitrary size. Blocks can be nested to any desired depth. Any number of hierarchical blocks can be open for editing at any time.
Custom Report Generator	OptiSPICE Schematics includes a powerful Custom Report Generator tool for netlist and text report generation. The report format is driven by a "form file" which contains formatting commands and constant text. Form file features allow you to control: Overall report structure, e.g. netlist formats by signal or by device, listings by device for bills of materials, etc.
Powerful Script language	OptiSPICE includes several powerful technologies for scripting and customization that allows full access to all design data and virtually every program function.
Graphics export	The schematic editor can save diagrams in the standard PDF (Acrobat), WMF (Windows Metafile) and DXF (AutoCAD) graphics formats. This capability allows you to pass graphics to other programs for plotting, enhancement, or incorporation into other documentation.
HSPICE compatible	Generate OptiSPICE or HSPICE compatible Netlists. Active and passive device model compatibility with industry HSPICE standard. Users can easily import external models and Netlists written in HSPICE format to OptiSPICE.
Simulation engine	OptiSPICE simulator incorporates equations governing optical components directly into an electrical simulation framework, thus forming a single-engine optoelectronic simulation software. Able to handle integrated optics, multiple optical channels (WDM), and multimode signals. Advanced numerical techniques for superior convergence. Advanced solver automatically selects the best convergence algorithm for reliable transient simulation convergence.
Thermal models	Includes thermal macro models that model the thermal behavior of the devices. Users can incorporate them into the optoelectronic simulation to provide reliable simulation results.

Feature	Description
Device library	Supports a wide variety of electrical circuit elements such as diodes, transistors, BJTs and MOSFETS along with optical components such as laser diodes, optical fibers and photodiodes. Enables accurate simulations by supporting BSIM3 models. Provides accurate implementation of different frequency dependent models including S-parameters, pole/residue expressions and transmission line models.
Waveform and signal integrity analysis	OptiSPICE includes the **Waveform Viewer,** a post processing analysis tool that allows users to view the data captured from all probes placed in an OptiSPICE circuit. It can be used to review the data captured following transient, AC, DC and operating point analysis of electrical, optical and optoelectronic circuits. OptiSPICE can also inter-work with the **Python** scripting language. This powerful scientific scripting language can be used to initiate OptiSPICE simulations and collate all the results for advanced post-processing and graphing functions. In addition OptiSPICE simulations can be run from **MathWorks ® MATLAB** technical computing language thus allowing for the post-processing of simulation results using its feature rich graphing and calculation tools. Finally, it is possible to view results directly within **OptiSystem** with the **OptiSPICE Output** component. This capability can be optionally launched after completion of a simulation and provides access to **OptiSystem** extensive set of visualization tools.
Parameter extraction	Laser parameter extractor allows users to generate models by extracting and fitting parameters from static and dynamic measurements of lasers. Filter parameter extractor allows users to translate S-parameters into compact and efficient pole/residue representations. Multimode fiber parameter extractor includes an optical fiber mode solver that allows users to generate libraries of fibers from a user defined refractive index profile.

Quick Start

This chapter is divided into three sections: *Starting OptiSPICE*, *Quick Start Part 1* and *Quick Start Part 2*.

Starting OptiSPICE provides quick instructions on how to create a new design schematic

Quick Start Part 1 will show you how to perform schematic editing, setup device parameters, place electrical probes, setup a simulation and an introduction to the OptiSPICE Netlist.

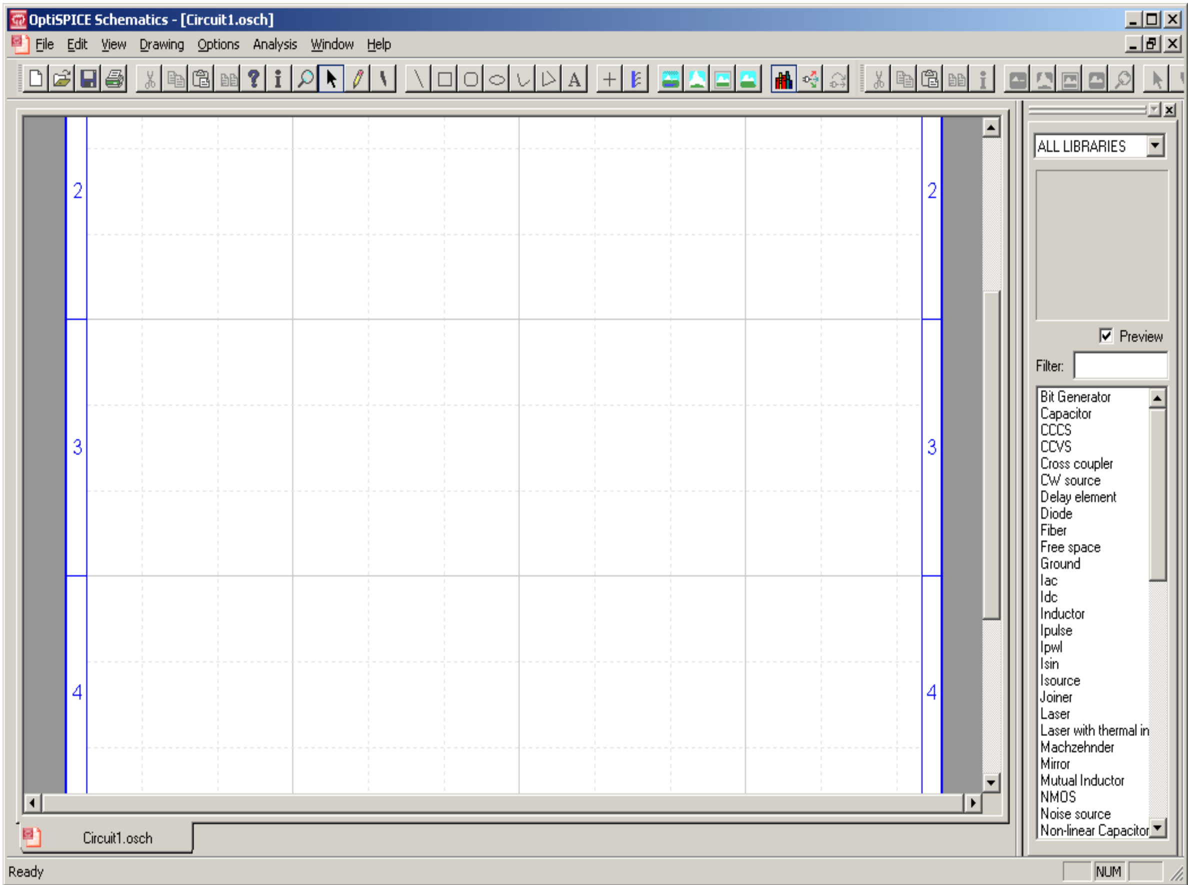
Quick Start Part 2 will show you how to setup designs that include optical and optoelectronic devices, place optical probes, create and edit models, setup global parameters and how to review and filter post-processing results.

Starting OptiSPICE

To start **OptiSPICE**, perform the following action.

Step	Action
1	From the **Start** menu, select **Programs** > **Optiwave Software** > **OptiSPICE 5**> **OptiSPICE Schematics.**
2	Click on the **Create/Open Design** tab.
3	Select **OptiSPICE** in the list.
4	Click the **OK** button *OptiSPICE loads and the graphical user interface appears (see Figure 1).*
5	Select **File > Save Design As**. *The **Save Design** dialog box appears.*
6	Save the schematic file. ***Note:*** Schematic file must be saved before viewing Netlist or running simulation.

Figure 1 OptiSPICE graphical user interface (GUI)



Main parts of the GUI

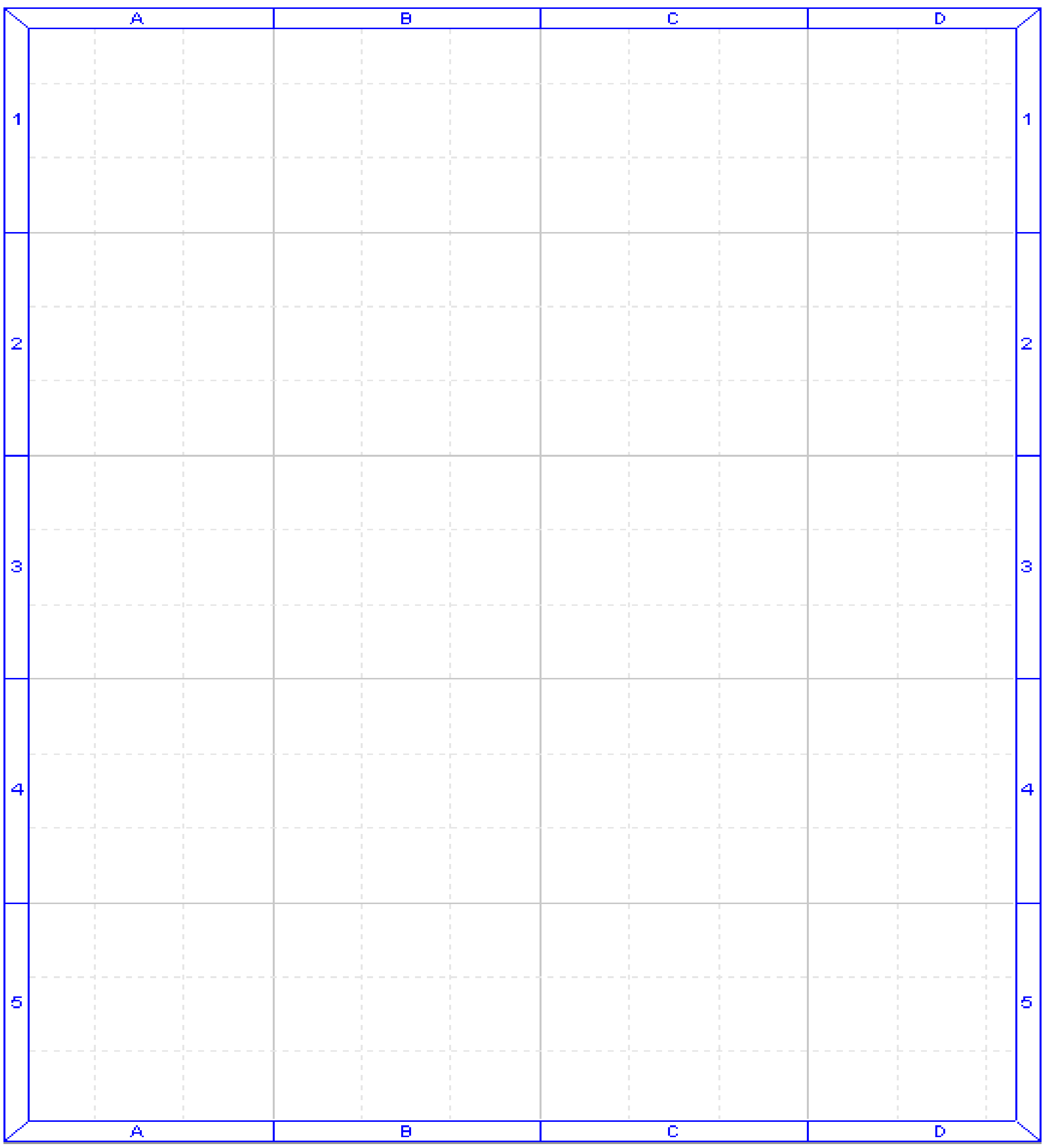
The OptiSPICE GUI contains the following main windows:

- Schematic layout
- Parts Palette

Schematic layout

The main working area where you insert devices into the layout, edit devices, and create connections between devices (see Figure 2).

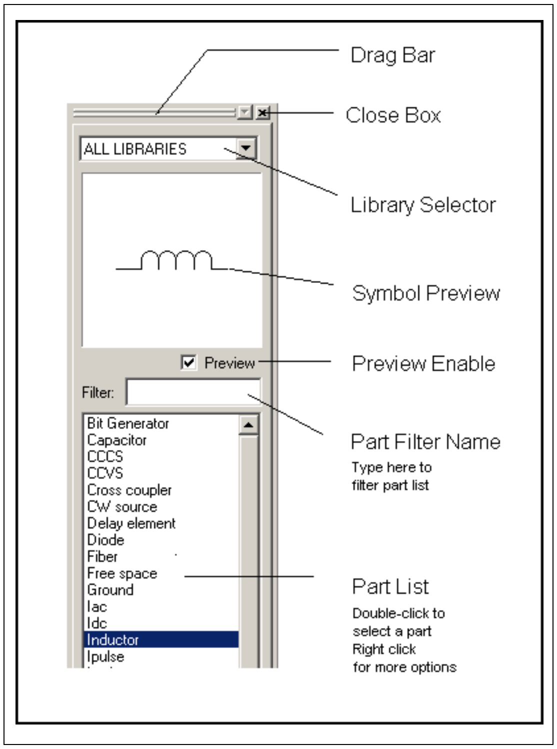
Figure 2 Schematic layout window



Parts Palette

Use Parts Palette, located in the main layout, to access devices to create the system design (see Figure 3).

Figure 3 Parts Palette



Quick Start Part 1 (Schematic Editing, Device Parameters, Probes, Simulation Setup, OptiSPICE Netlist)

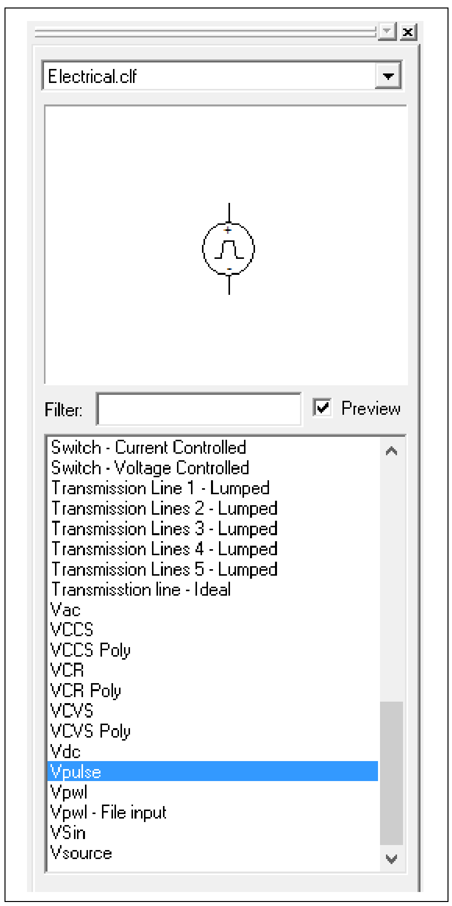
Schematic Editing

To place and connect devices in the schematic, perform the following procedures. For more details about schematic editing see the OptiSPICE Schematics book.

Placing devices onto the schematic

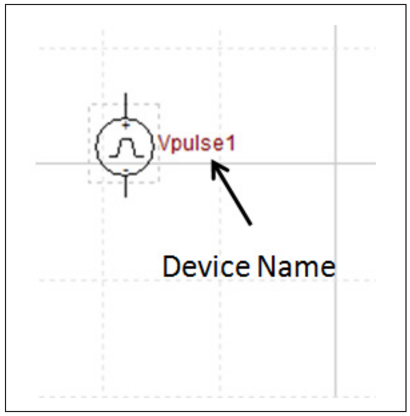
Step	Action
1	Click the library selector drop-down list in the Parts Palette and choose the library ***Electrical***. *Devices in the **Electrical** library will now be displayed in the part selection list.*
2	Click on the item **Vpulse** to select it. *You will see a preview image of the voltage source appear in the box above the list (see Figure 4)*

Figure 4 Vpulse device symbol preview image



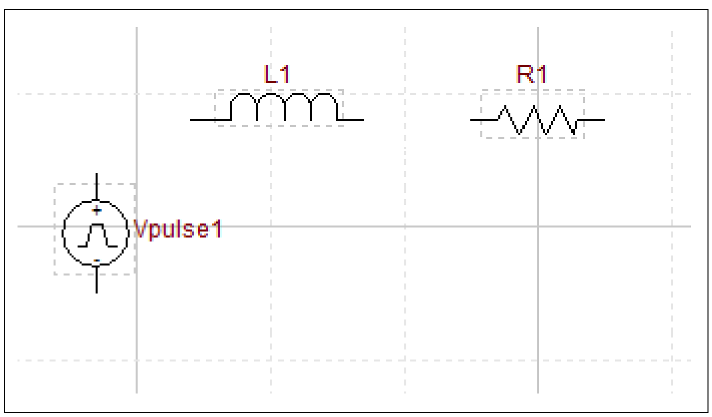
3 Drag and drop the selected **Vpulse** device onto the schematic layout
__Vpulse__ is placed in the schematic with an auto device name __Vpulse1__ (see Figure 5).

Figure 5 Placing a device onto the Schematic layout



4 Repeating Steps 2 - 3, place an **Inductor** and **Resistor** from the ***Electrical*** library as shown in Figure 6.

Figure 6 Placing devices

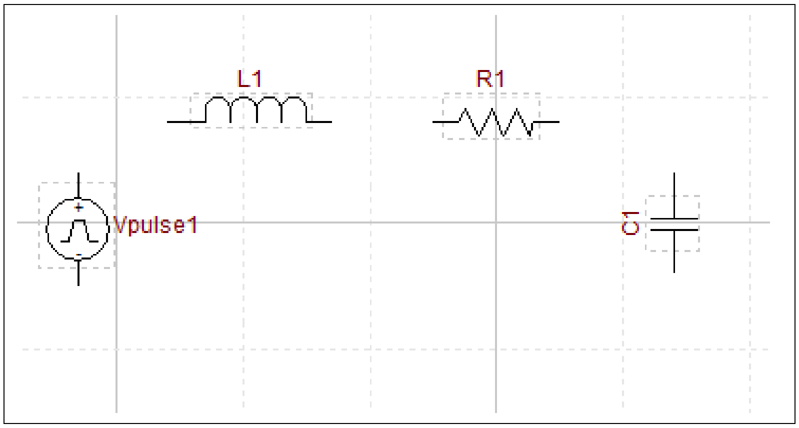


Changing the device orientation (flipping and rotating)

5 Place a **Capacitor** device on to the schematic layout

6 Right click on the capacitor **C1** and select *Rotate left*.
Capacitor __C1__ is rotated to the left (see Figure 7).You can also flip devices horizontally and vertically from the same menu

Figure 7 Rotate left operation



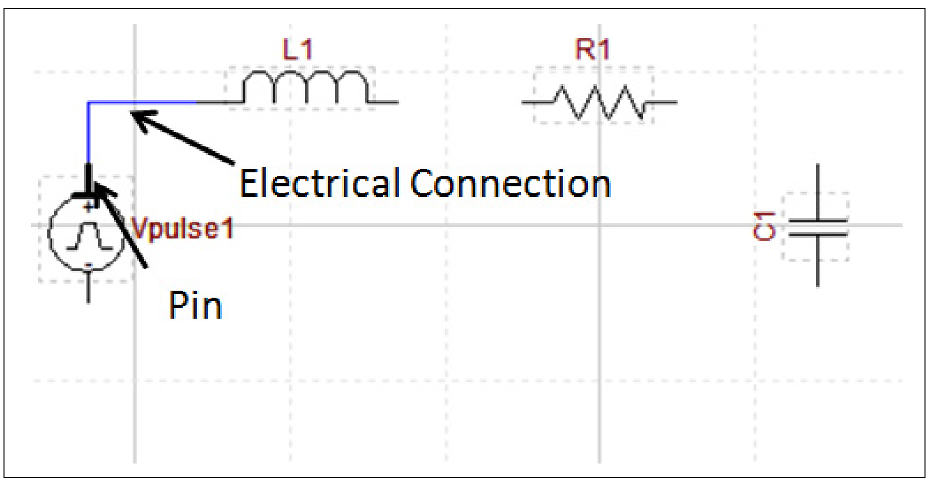
Connecting devices

Electrical devices have pins that can be connected using a signal (wire or net)

7 Connect and hold the top pin of **Vpulse1**

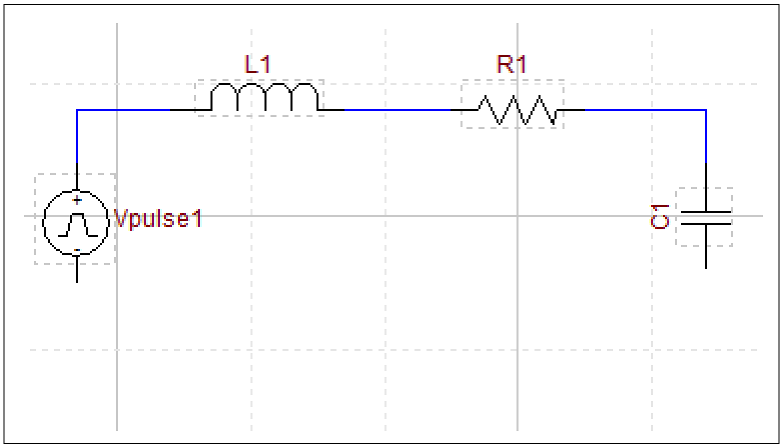
8 With the mouse button still held down, move to the end of the left pin of the inductor **L1** as shown in *Figure 8*
*You have now completed the connection between **Vpulse1** and inductor **L1***

Figure 8 Connecting devices (Vpulse1 to L1)



9 Repeat step 8 to connect **L1** to **R1** and **R1** to **C1** (see *Figure 9*)

Figure 9 Connecting devices (L1-R1-C1)



Connecting to ground

The ***Ground*** symbol automatically names the attached signal to 0 and causes it to be logically connected to all other ground signals in the circuit.

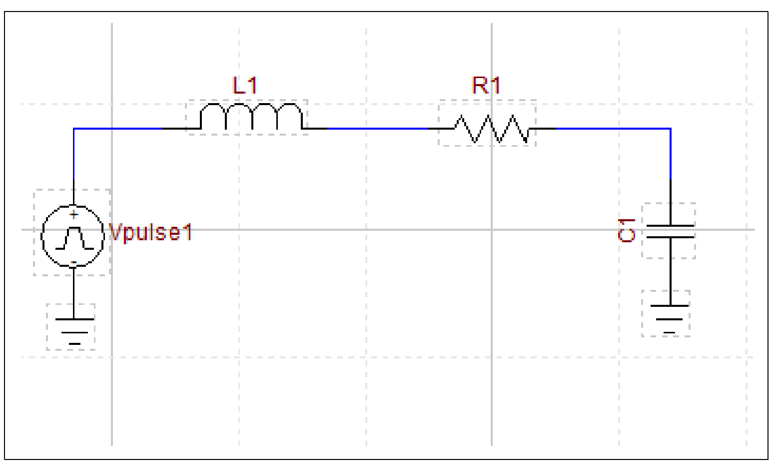
10 Double click on the **Ground** device from the ***Electrical*** library.

11 Move the cursor to the end of the bottom pin of **Vpulse1** such that the pin of **Vpulse1** just touches the pin of the **Ground** and click the mouse button at this point

This is called automatic pin connection where a signal connection is made between pins automatically

12 Repeat steps **10**-**11** to connect the ground for **C1** (see Figure 10).

Figure 10 Connecting to ground



Device parameters

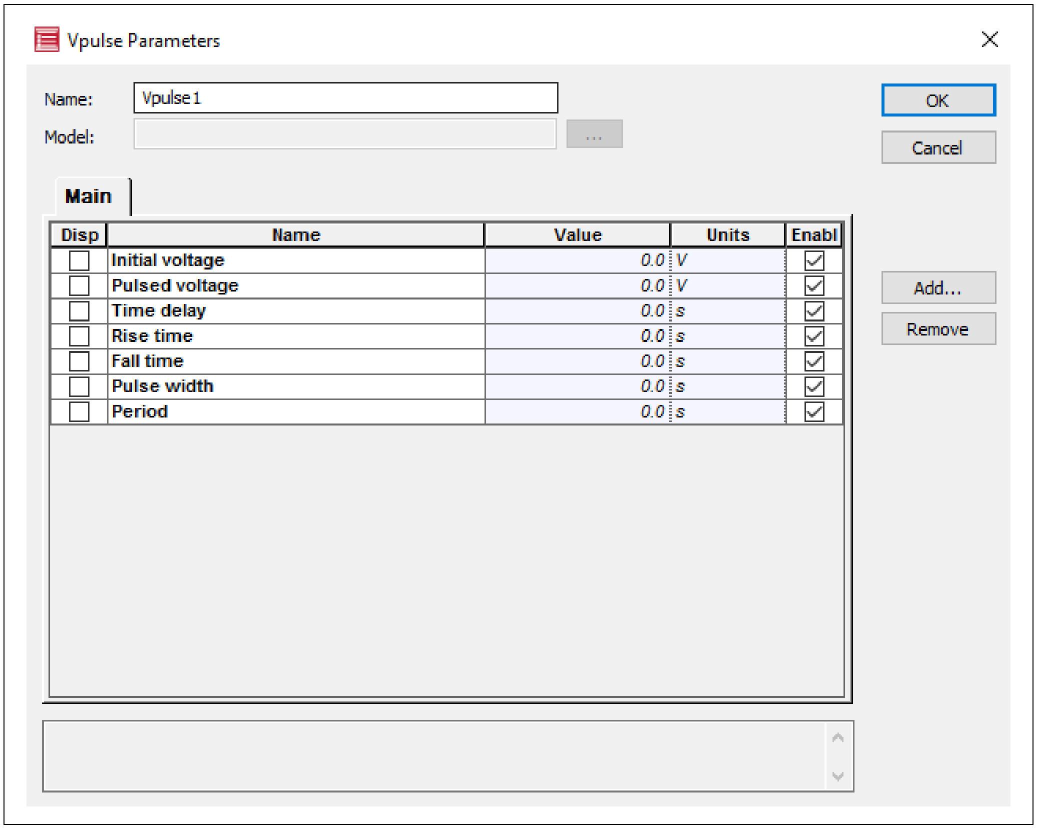
Launching the device parameter dialog box

Double-click a component to view and edit the parameters for the component. To view the properties for **Vpulse1**, perform the following action.

Action

- In the **Schematic layout**, double-click the **Vpulse1** device.
  *The **Vpulse Parameters** dialog box appears (see Figure 11).*

Figure 11 Device parameters – Vpulse



Device parameters are organized by categories. Typically a device has two parameter categories, each represented by a tab in the dialog box.

- **Main** - parameters that are necessary for the simulation.
- **Optional** - parameters that are optional for the simulation.

Each category has a set of parameters. Parameters have the following properties:

- **Disp** - check box to display the parameter in the schematic layout
- **Name** - parameter name
- **Value** - parameter value initially filled with default value

- **Units** - units for the parameter
- **Enabl** - check box to include or omit the parameter in the Netlist.

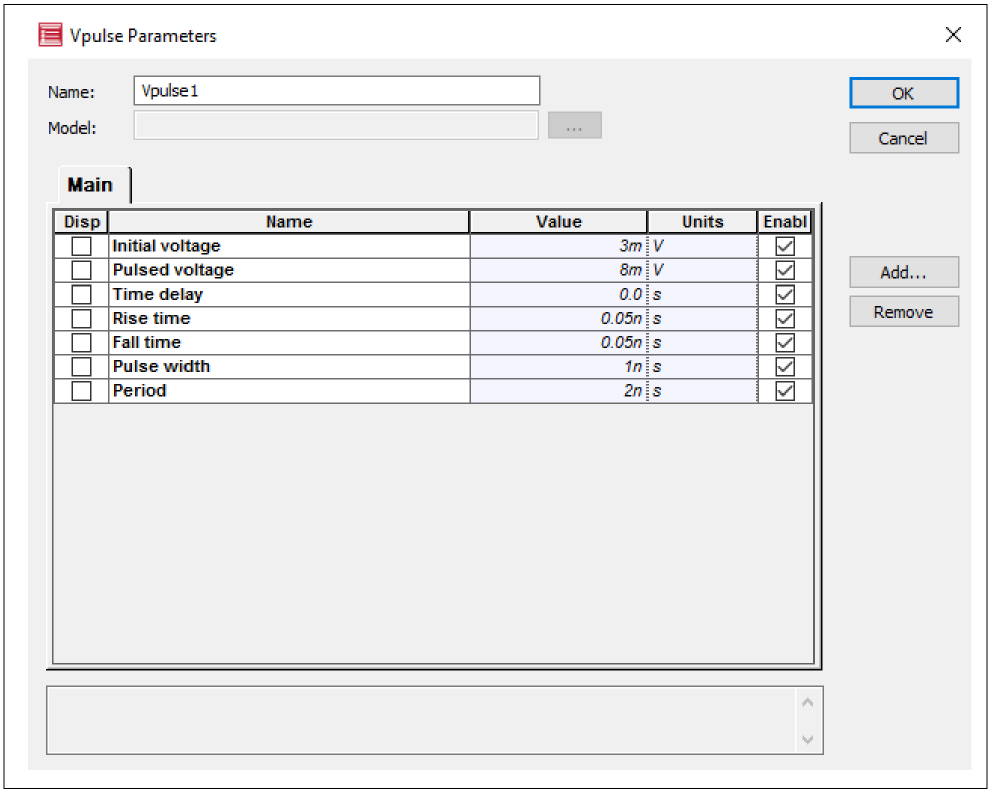
For the parameter descriptions and details see the OptiSPICE Device Library book.

Editing device parameters

In this design, you have to change some of the device parameters from the default values. First edit the **Vpulse1** device parameters, by performing the following procedure.

Step	Action
1	Double-click the **Vpulse1** in the **Schematic layout.** *The **Vpulse Parameters** dialog box appears.*
2	Click in the **Value** cell beside **Initial Voltage**
3	Enter "3m" in that cell
4	Repeating steps 2 - 3, enter the following parameters for the **Vpulse1** device. • **Pulse Voltage**: 8m • **Rise time**: 0.2n • **Fall time**: 0.05n • **Pulse width**: 1n • **Period**: 2n ***Note:*** The character 'n' and 'm' following the numbers are the unit abbreviations for *nano* (1e-9) and *milli* (1e-3), respectively.
5	Click **OK**. *This completes the parameter editing for the **Vpulse1** device.*

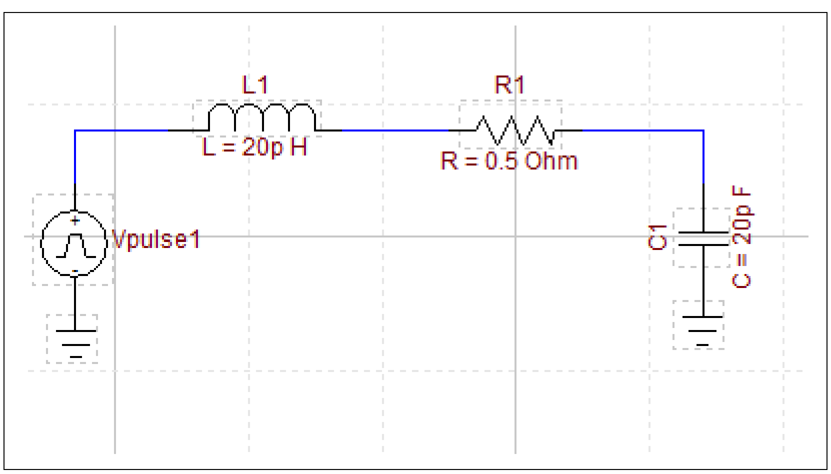
Figure 12 Editing Device parameters Vpulse



6 Enter the values for **L1**, **R1** and **C1** by using the device parameter dialog box

- **L1**: 20p
- **R1:** 0.5
- **C1**: 20p

Figure 13 Setting up device values for L1, R1, and C1



Probes

The Probe is a special device in OptiSPICE Schematics that allows you to monitor simulation results such as voltage, current, and optical field. You have to place it on a signal or a pin where you want to monitor the simulation results. It can take the following signal types depending on where it is placed.

- Voltage probe - when placed on an electrical signal (wire or net).
- Current probe - when placed on a device pin through which an electrical current flows.
- Optical probe - when placed on a device pin through which an optical signal (light) passes through.

Placing Probes

Step	Action
1	Click the library selector drop-down list in the Parts Palette and choose the library **Probe**.
2	Click on the device **Probe** to select it.
3	Drag and drop the selected **Probe** exactly on to the signal connecting **Vpulse1** and **L1** as shown in Figure 14 (green hot spot should touch the signal). *These probes are voltage probes. Double clicking shows an empty dialog box, the title of which confirms it is a voltage probe (see Figure 15).*

Figure 14 Placing Probes

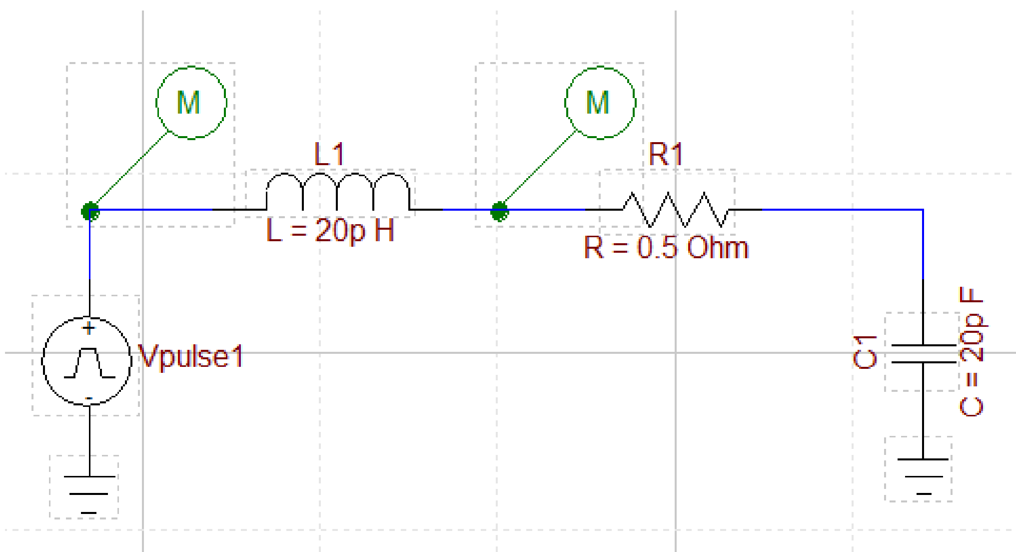
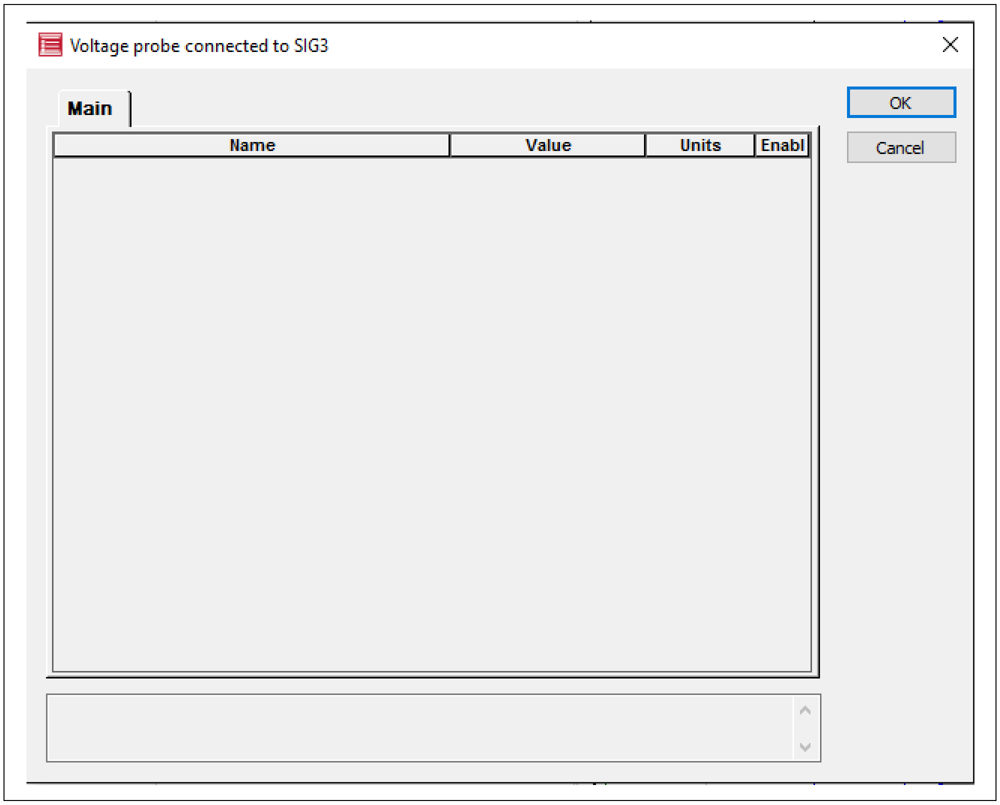
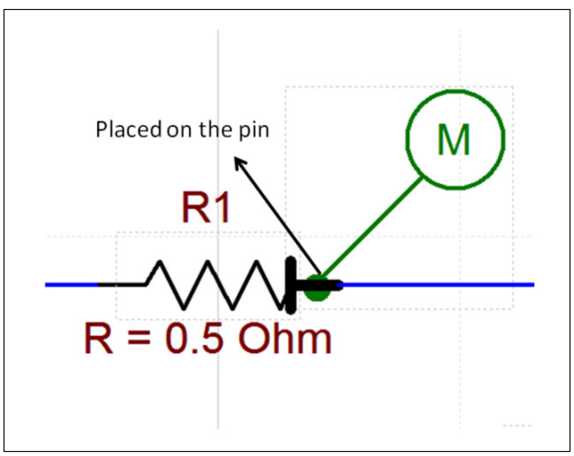


Figure 15 Voltage probe dialog box



4 Drag and drop the second **Probe** exactly on the right pin of **R1** as shown in Figure 16.
This is a current probe.

Figure 16 Placement of current probe



Note: Placing the Probe on the pin is somewhat difficult task compared to placing probe on a signal because the size of the pin is smaller than the signal. Also, when you are placing near the spot where a pin and a signal touching, there are chances that the Probe might lie on a signal rather than on a pin. Zooming in may help to easily visualize where the Probe is connected. You can use the arrow keys to make fine movements to the placement of Probe. It is always safer to verify the type of the Probe by double clicking on it

Simulation Setup

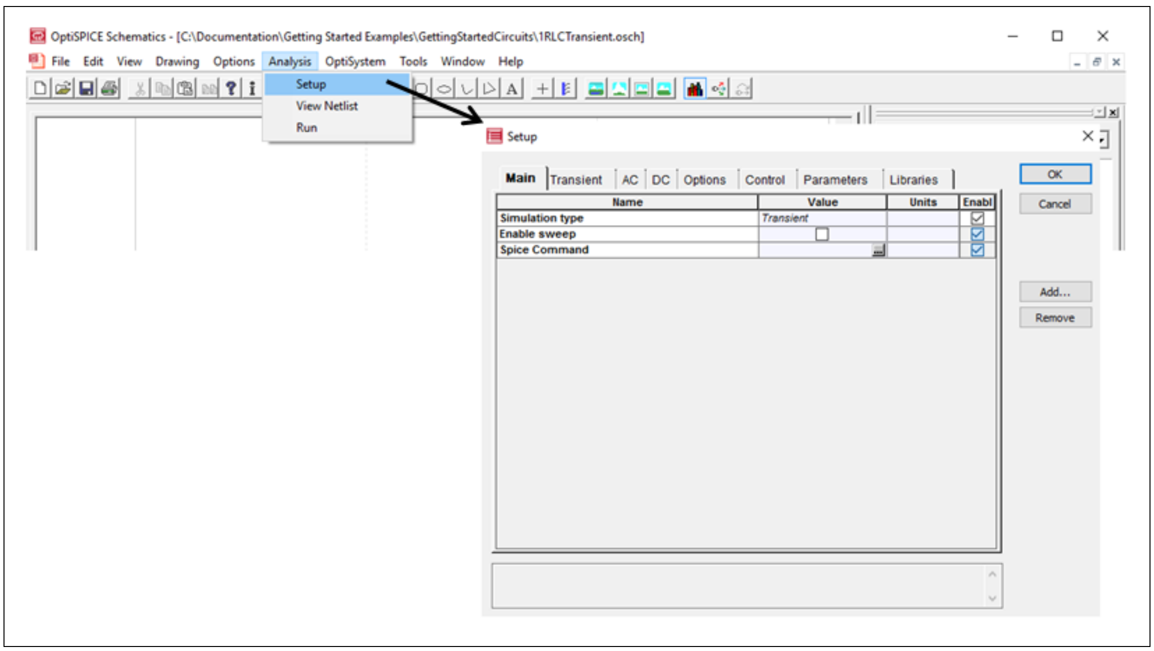
Before running simulation, you need to complete all the necessary specifications such as type of simulation (AC, DC, or transient), simulation options, global parameters, etc.

Opening the simulation setup dialog box

Action

- Select **Analysis > Setup**.
  Setup dialog box opens (see Figure 17).

Figure 17 Simulation setup dialog box



Setup parameters are organized by categories.

- **Main** - to specify the analysis type
- **Transient** - to specify transient analysis settings
- **AC** - to specify AC analysis settings
- **DC** - to specify DC analysis settings
- **Options** - to specify simulation options
- **Control** - to control simulator options and output settings
- **Parameters** - to manage global parameters

- **Libraries** - to add/remove library paths.

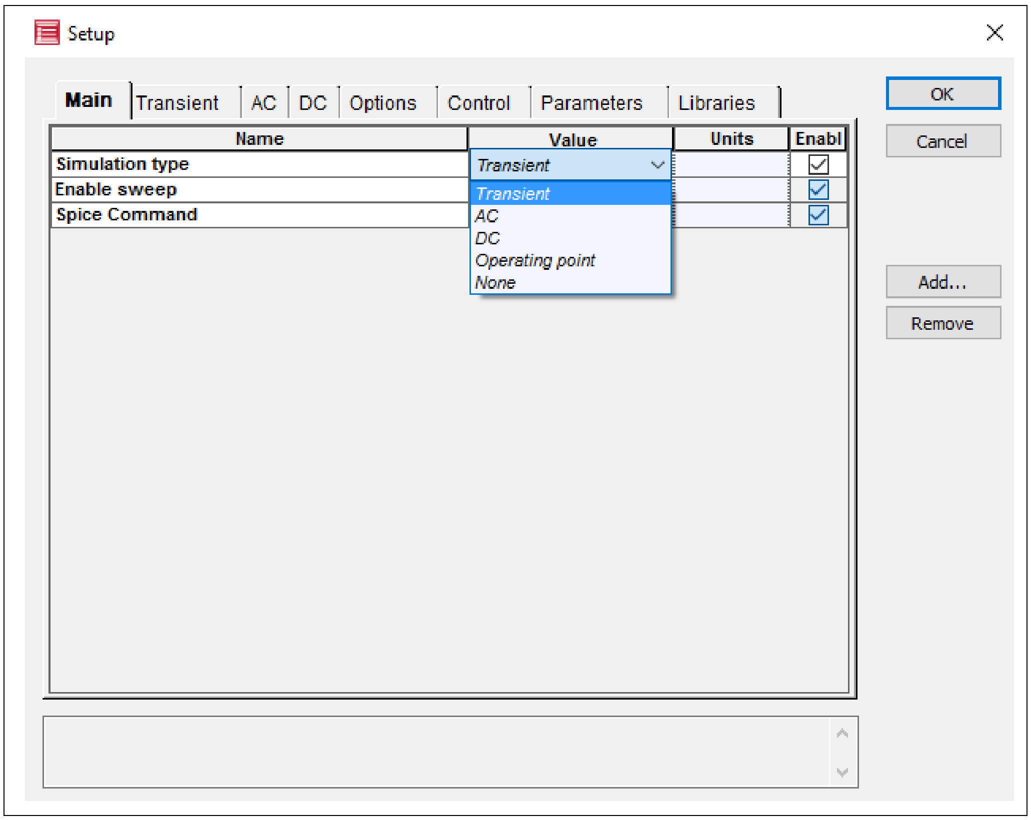
Performing transient analysis

In this design example, we will perform a transient analysis.

Step **Action**

1 Select the *Transient* analysis from the drop down menu next to the simulation type

Figure 18 Setting up a Transient simulation

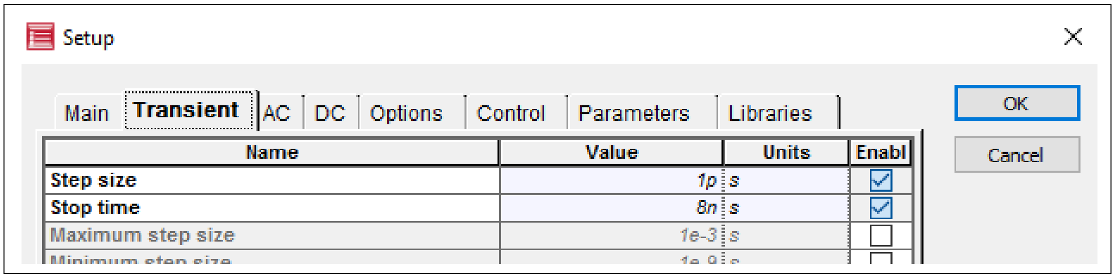


2 Click the **Transient** tab.

3 In the Transient tab, type the following values

- **Step size**: 1p
- **Stop time**: 8n

Figure 19 Transient simulation parameters



OptiSPICE Netlist

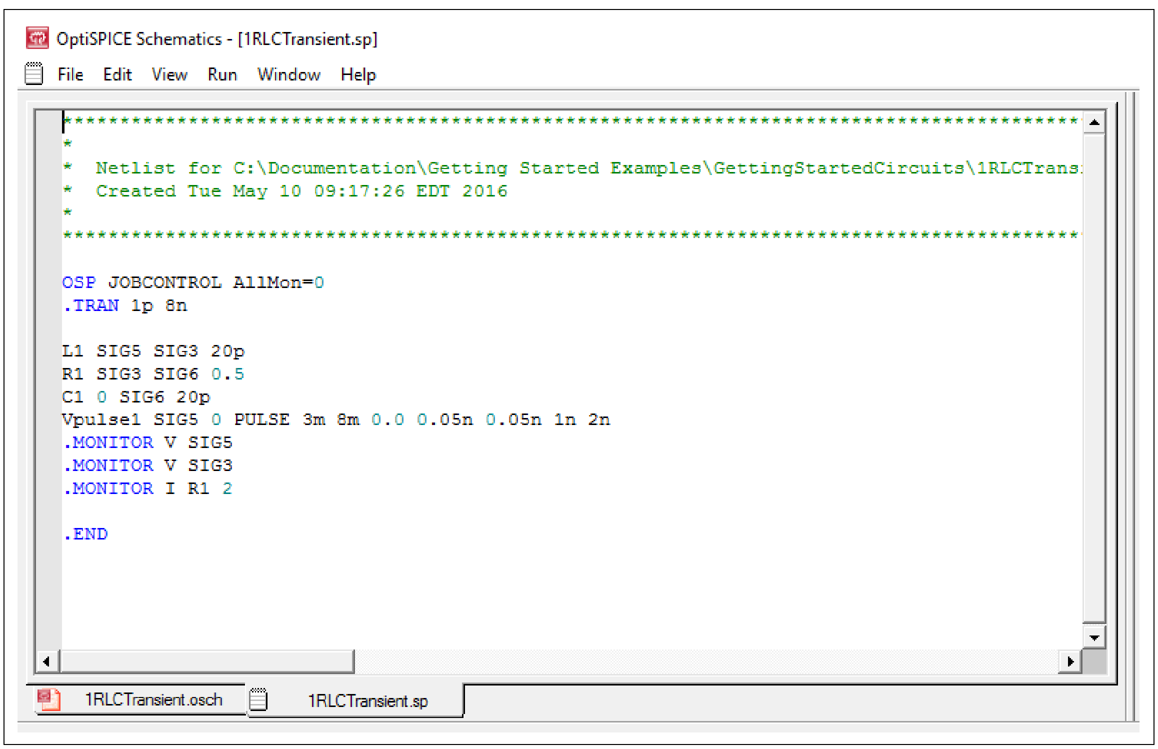
The OptiSPICE Netlist is the text file representation of your schematics design and it is used by the OptiSPICE simulator to run a simulation. You can view and edit the Netlist. See the OptiSPICE Simulator Command Reference book for more information.

Viewing the OptiSPICE Netlist

Action

- Select **Analysis > View Netlist**.
  The Netlist is opened in a new window as shown by Figure 20.

Figure 20 OptiSPICE Netlist

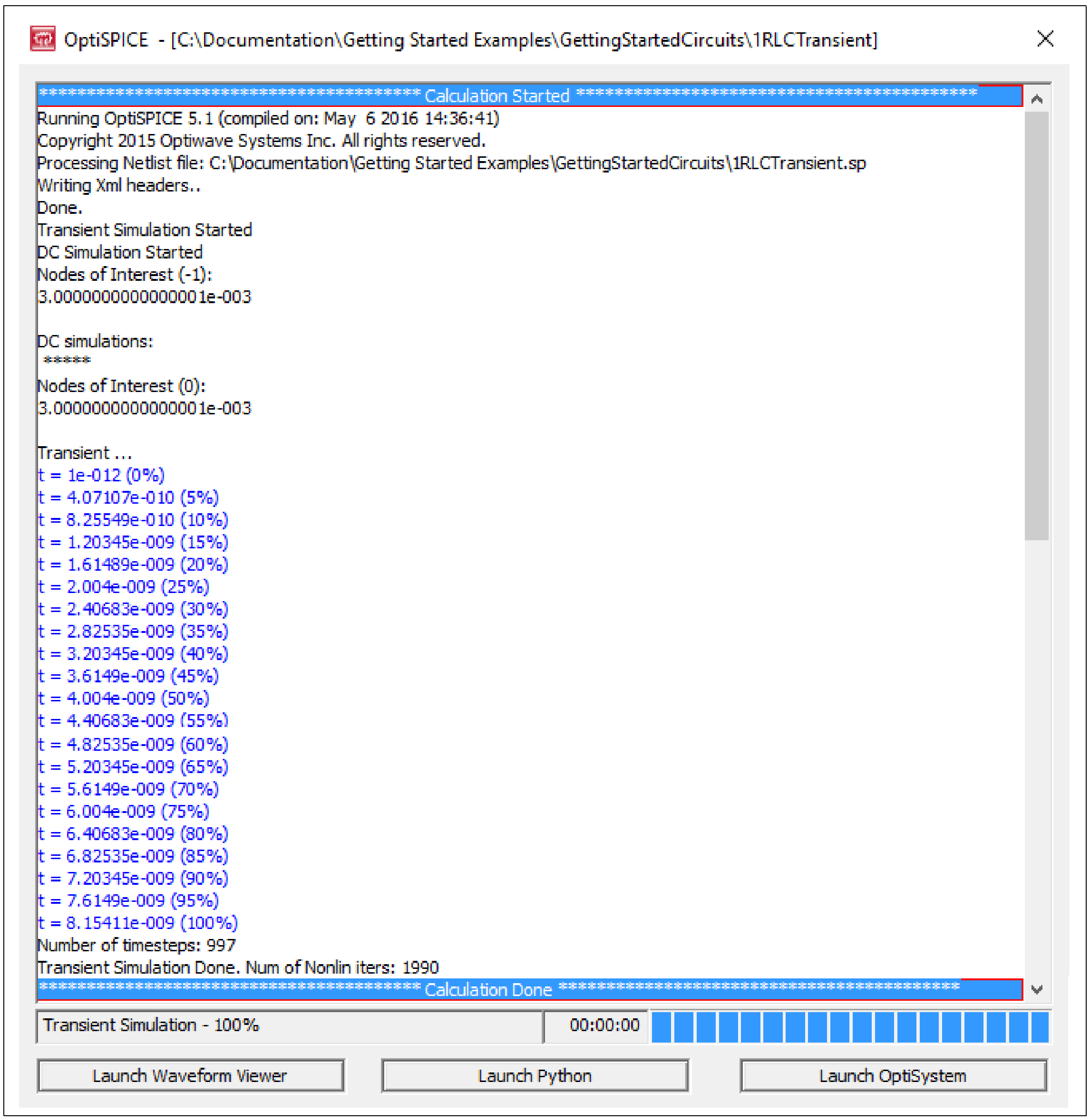


OptiSPICE Simulation

Running Simulation

Step	Action
1	Switch back to the schematic design window if you are still in the Netlist window.
2	Select **Analysis > Run**. *OptiSPICE window appears (see Figure 21). Here you can see the simulation progress and other simulation related details.*

Figure 21 Simulation in progress



3 Once the simulation is done, select “Launch Waveform Viewer” to view results with the Waveform Viewer feature.

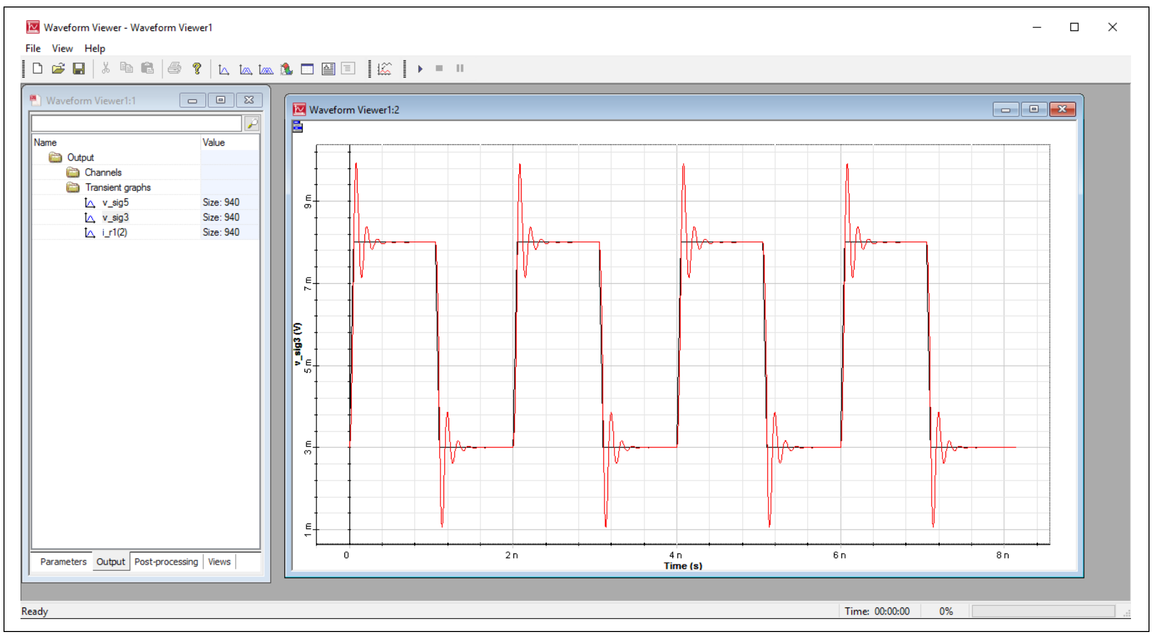
4 In Waveform Viewer, it is possible to visualize the results by double clicking on a signal (see Figure 22). For further information on how to use the Waveform Viewer please see *OptiSPICE Waveform Viewer.pdf*

5 You can also use “Launch OptiSystem” to view results within OptiSystem

If OptiSystem is launched, the OptiSPICE Output component is automatically created within the OptiSystem layout.

Note: Python can also be used to post process the results and create custom graphs (please see *OptiSPICE Python Post Processing.pdf* for further details in how to use this feature)

Figure 22 Simulation results (Waveform Viewer)



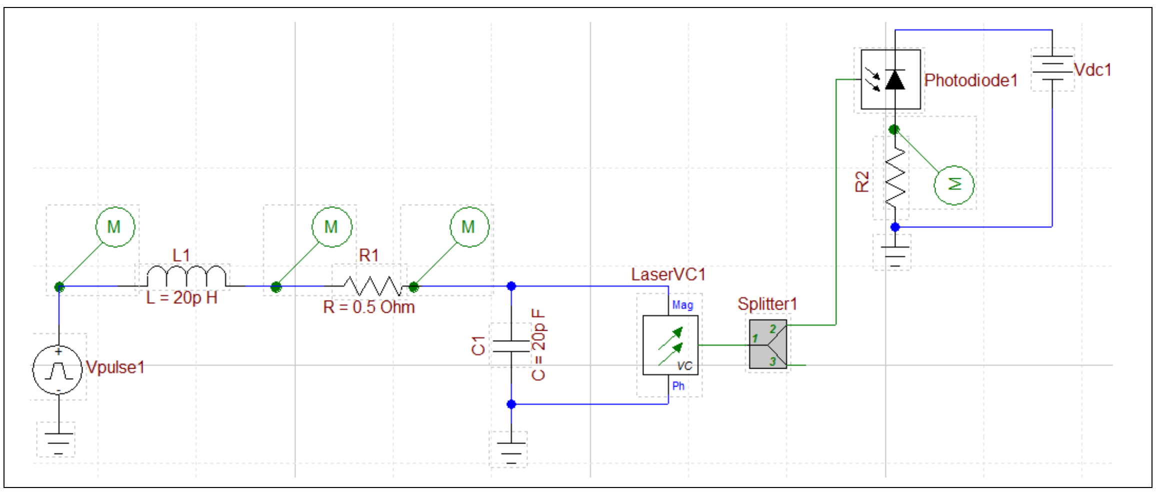
Quick Start Part 2(Optical/Optoelectronic devices, Editing Models, Filtering results)

OptiSPICE simulates Optical signals by keeping track of the magnitude of the signal envelope and the phase of the carrier. This method allows for faster transient simulations and the accurate representation of various optical effects such as reflections and interference

Adding Optical and Electro-Optical devices

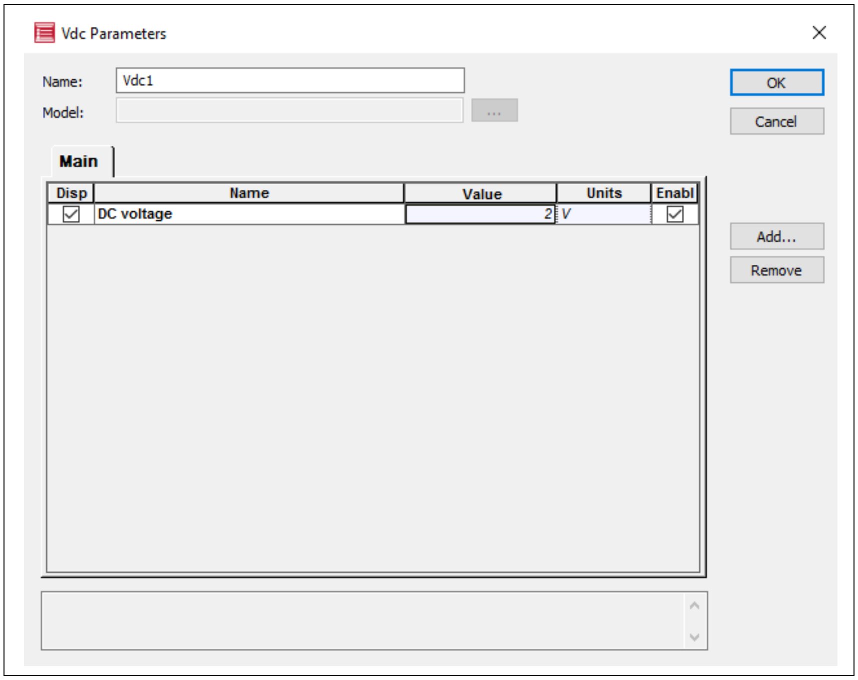
Step	Action
1	Drag and drop the devices, Laser-VC, Optical Power Splitter, Photodiode, Vdc, resistor, probe (current), ground on the schematic layout
2	Connect them as shown in Figure 23

Figure 23 Layout which includes optical and electro-optical devices



3 Double click on vdc1 and set the DC voltage to 2 (see Figure 24).

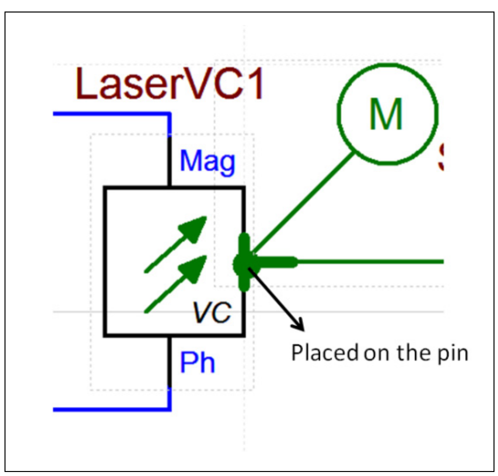
Figure 24 Vdc Device parameters



Placing an optical probe

Step	Action
1	Place a probe on the right pin of the laser source *Probes need to be placed on the pin (Figure 25) of the optical devices to measure optical data*

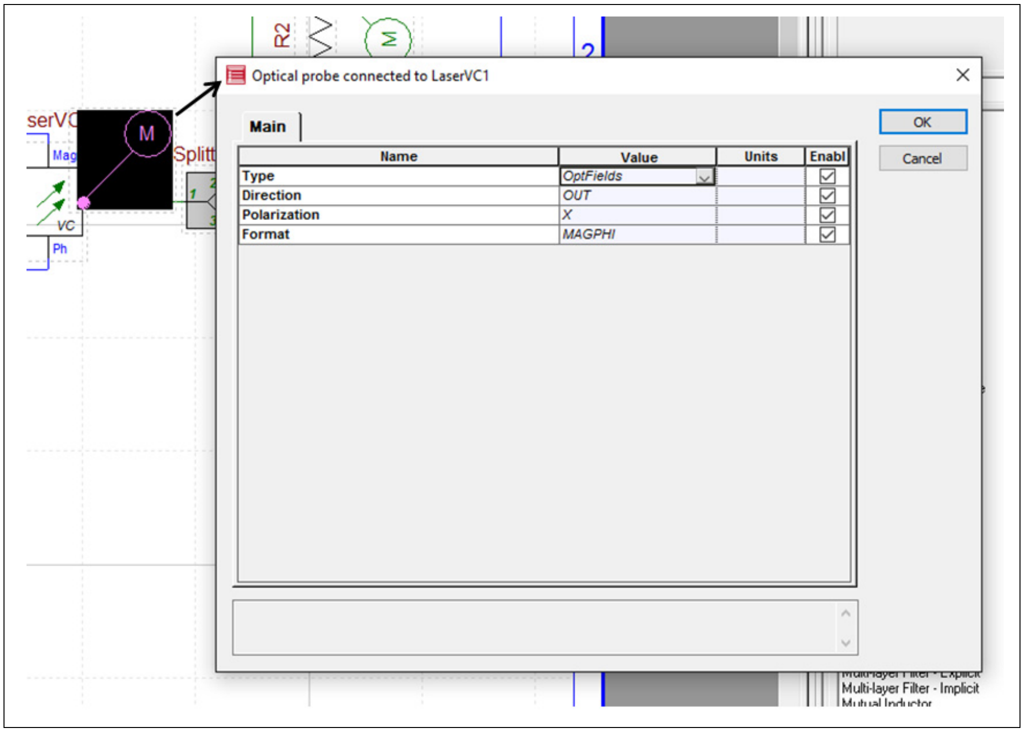
Figure 25 Probe placement on optical device



2 Double click on the probe placed on the LaserVC1

3 Set the output type as *OptFields* and click OK (Figure 26).

Figure 26 Optical probe settings



Adding Global parameters

Step **Action**

1 Click on analysis and select setup from the drop-down menu (Figure 27).

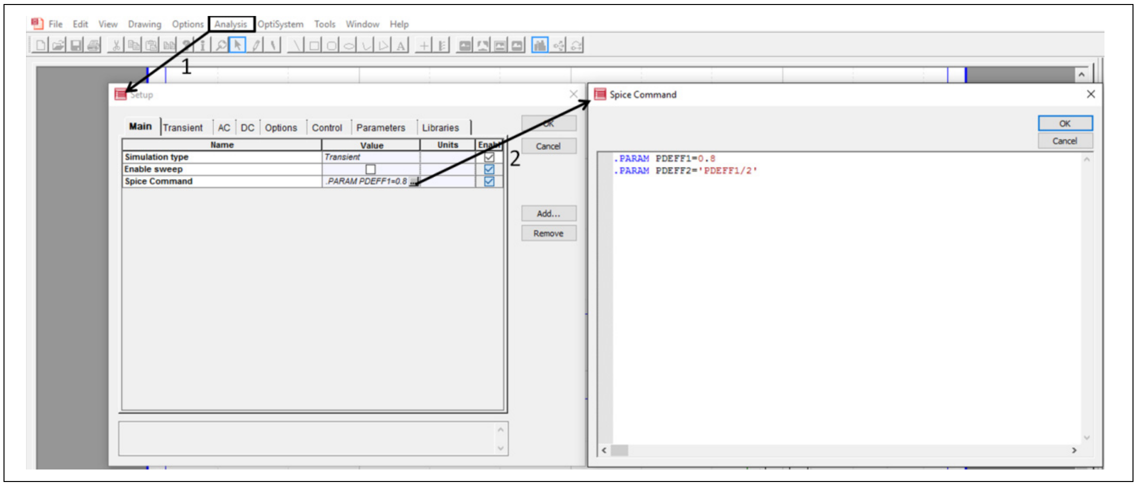
2 Click on the button with on the Spice Command value box.

3 Enter the following commands in the Spice Command text box.

- **.PARAM PDEFF1=0.8**
- **.PARAM PDEFF2='PDEFF1/2'**

.PARAM command in OptiSPICE supports basic mathematical operations such as addition (+), subtraction (-), division (/), multiplication (*) as well as log and power (^).

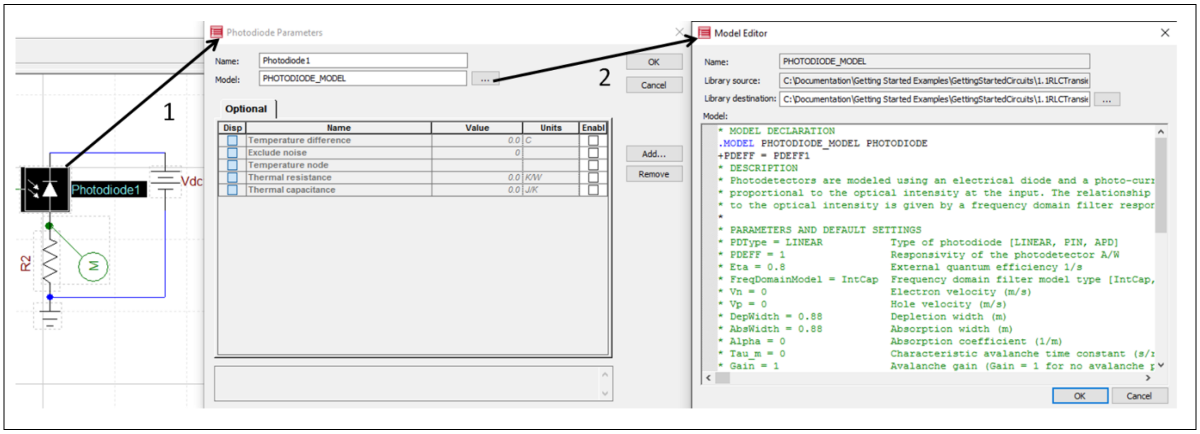
Figure 27 SPICE Command text box



Model Editor

Step	Action
1	Double click on the Photodiode1 to open the Photodiode parameter menu (Figure 28)
2	Click on the button next to the Model Name to open the model editor menu
3	Add the following parameter to the Photodiode model: **+ PDEFF = PDEFF1** *'+' is used to define a new line in the model editor menu. PDEFF is a Photodiode model parameter which sets the responsivity of the device*

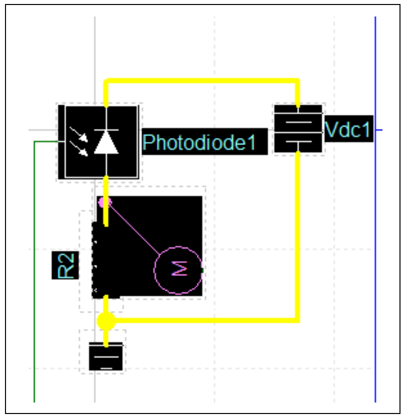
Figure 28 Photodiode model editor



Copy and paste devices

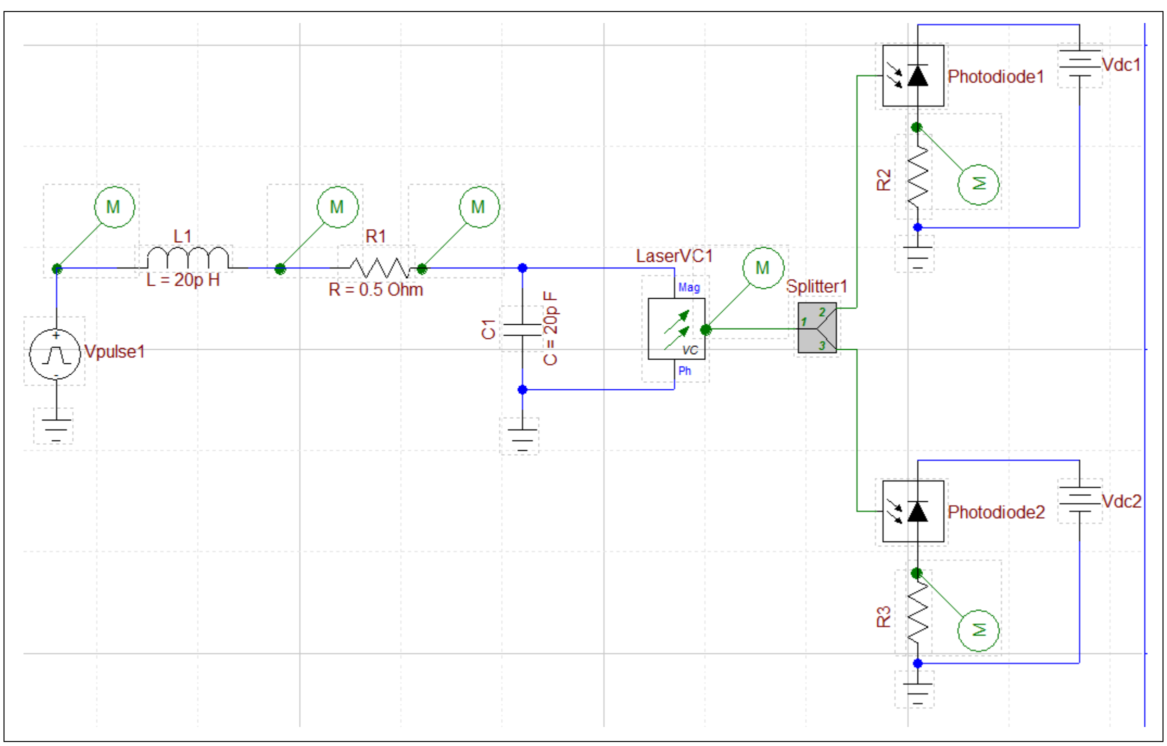
Step	Action
1	Select the devices shown in Figure 29 using the cursor.
2	Copy the selected devices by pressing **Ctrl** and **C** keys simultaneously

Figure 29 Copying and pasting devices



3	Paste the selected devices by pressing ctrl and v keys simultaneously
4	Connect the photodiode to the bottom arm of the Splitter1 as shown in Figure 30

Figure 30

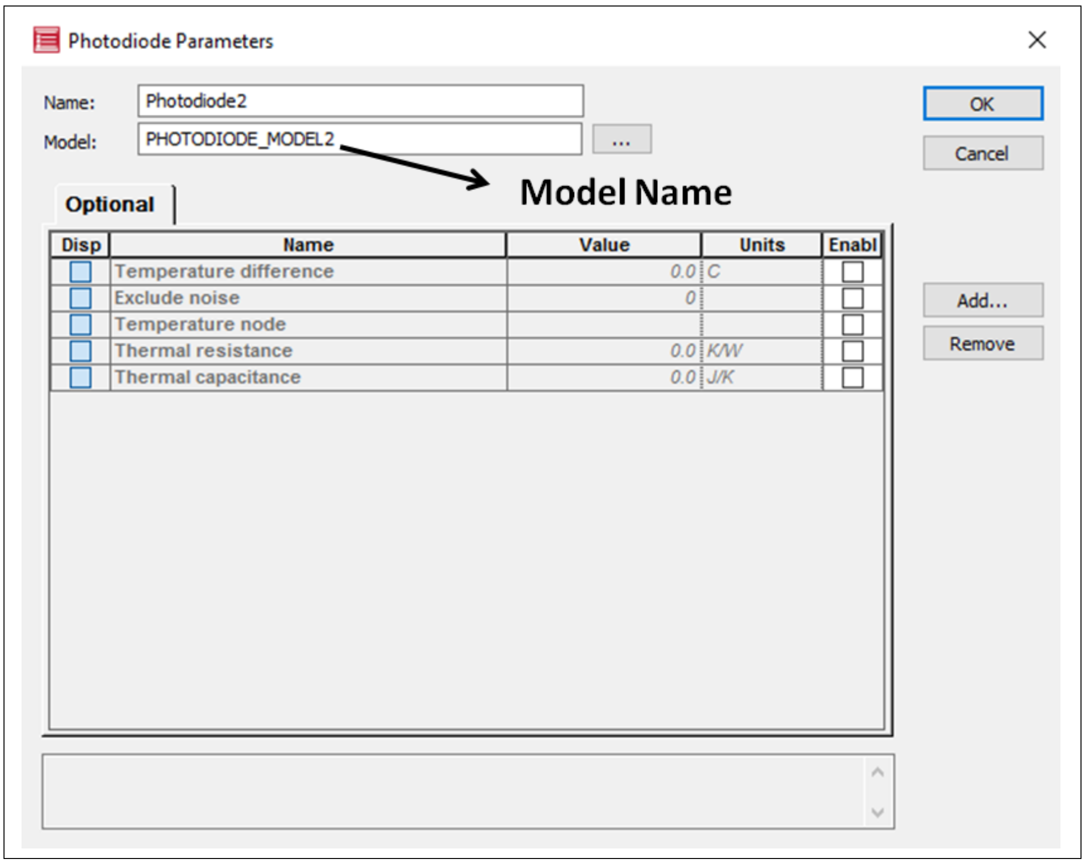


Creating a new model

In OptiSPICE models sharing the same name share the model parameters. It is important to remember that once a parameter in a shared model is modified, the change will occur across all the devices sharing the same model.

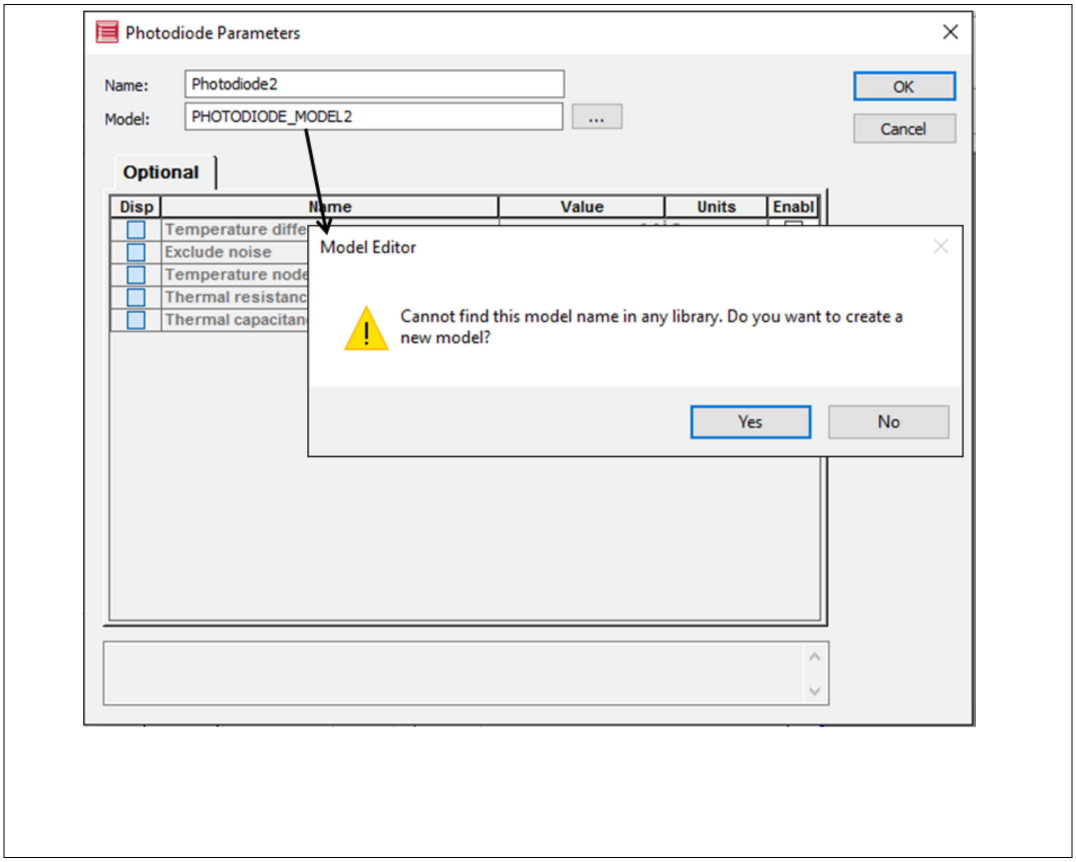
Step	Action
1	Double click on the device Photodiode2
2	Change the model name to PHOTODIODE_MODEL2 (Figure 31).

Figure 31 Launching model editor



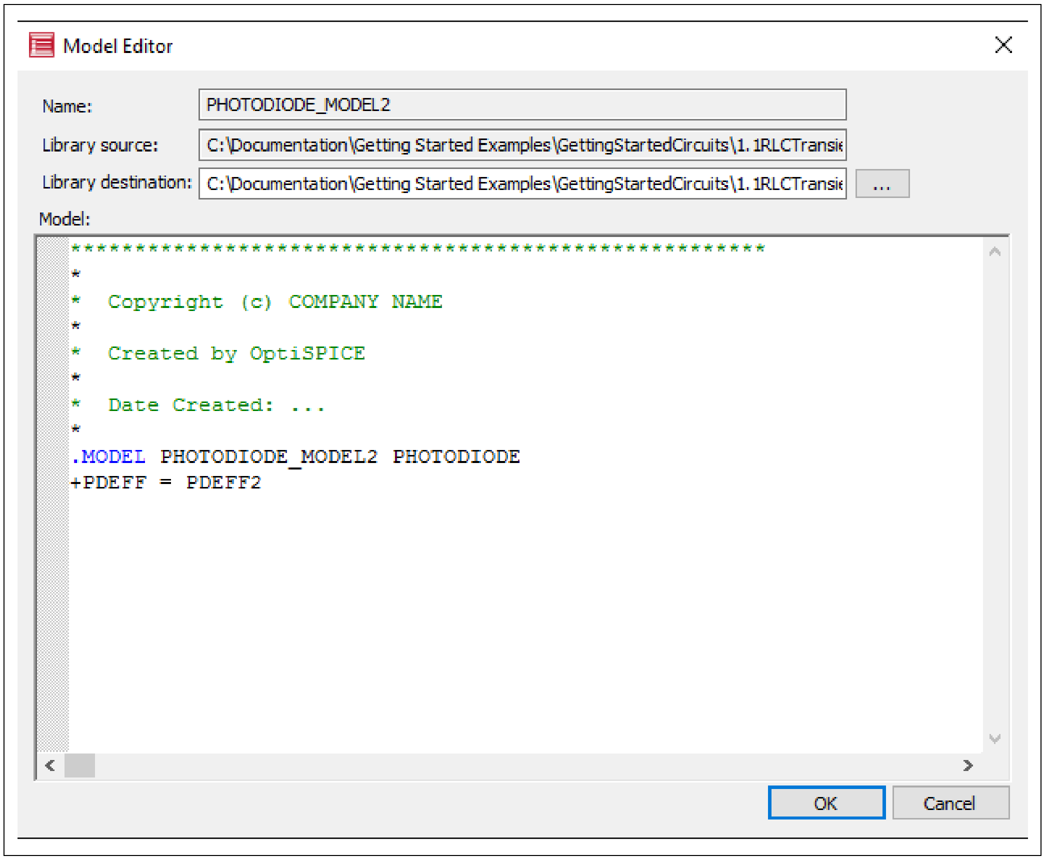
3 Click on yes when prompted to create a new model (Figure 32).

Figure 32 Launching model editor



4 Add the following device parameter to the photodiode model (Figure 33):
+ PDEFF = PDEFF2

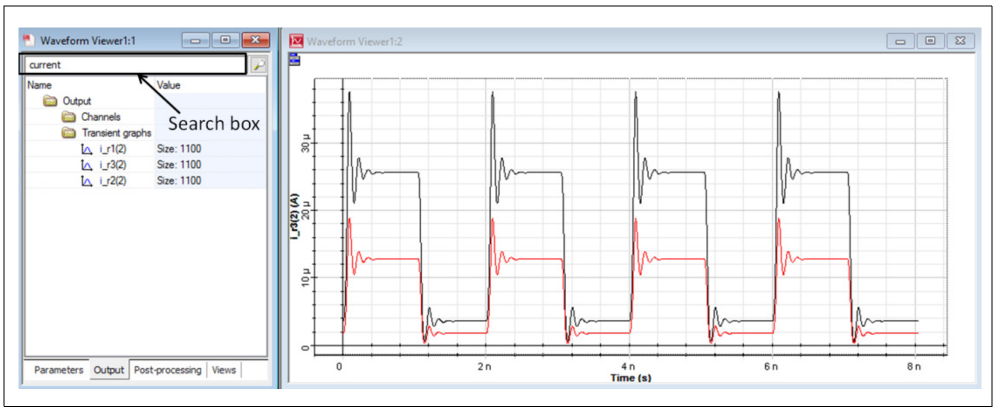
Figure 33 Photodiode model editor



Filtering the simulation results

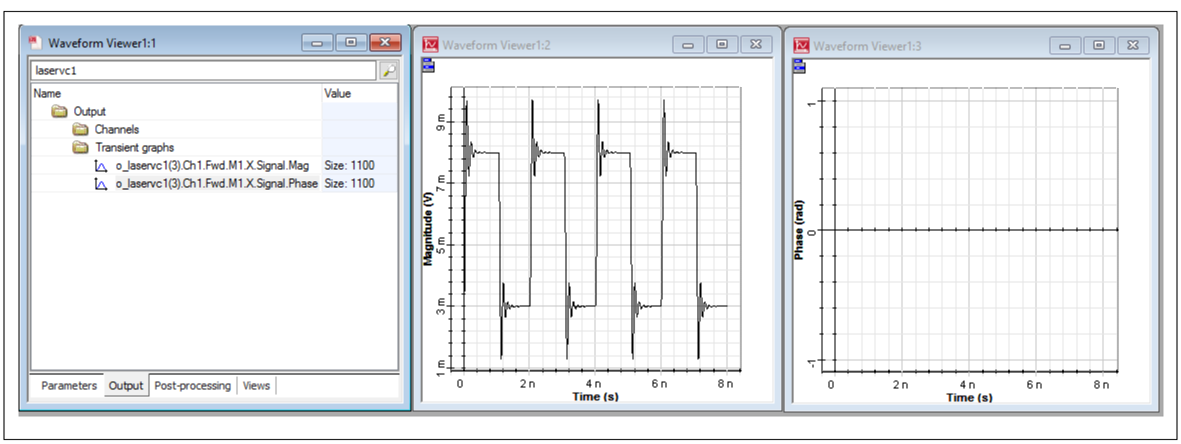
Step	Action
1	Run the simulation by clicking on analysis and selecting run from the drop-down menu.
2	Once the simulation is complete click on the launch waveform viewer button to launch the waveform viewer
3	Type current in the search box (Figure 34) and then press enter to filter the simulation results
4	Double click on i_r2(2) to see the time vs. current plot of the resistor r2
5	Drag and drop i_r3(2) onto the figure to plot both results on the same graph

Figure 34 Current versus time plot of “i_r1(2)” and “i_r3(2)”



Step	Action
6	Clear the filter by deleting current from the search box
7	Type laservc1 into the search box and then press enter to see the simulation results from laservc1 (Figure 35).
8	Double click on each result to visualize the magnitude and the phase of the laser output

Figure 35 Magnitude and phase of Laser “laservc1”



You have completed the OptiSPICE quick start tutorial!

OptiSPICE User Tips

The following section provides a few tips on how to deal with common problems or issues that may occur when designing and simulating optical and electrical circuits in OptiSPICE.

Simulation Engine

Floating nodes

Any portion of a circuit which is electrical requires at least one ground in the circuit. If a circuit contains multiple separate electrical circuits (for example a laser driver associated with an optical modulator and an electrical amplifier associated with a photodetector circuit) then each independent electrical circuit must be connected to a ground.

Long simulation times

It is not uncommon for SPICE simulations to require long simulation times due to the requirement to take extremely small time steps when modeling fast changing events in amplitude or phase. If you find that your simulations are taking to long to complete, it is recommended to reduce the ***Stop time*** of the simulation. For example if you are modeling multiple input pulses to a modulator, reducing the ***Stop time*** will create a smaller set of impulse functions.

Another way to increase the speed of your simulations is to force the simulator to not go below a minimum step size (this is controlled by the parameter ***Minimum step size***). This will likely increase the speed of your simulations but it may also reduce the resolution of your results. It is recommended to try different settings for the minimum step size to find the right balance between simulation time and required resolution.

Non uniform time step

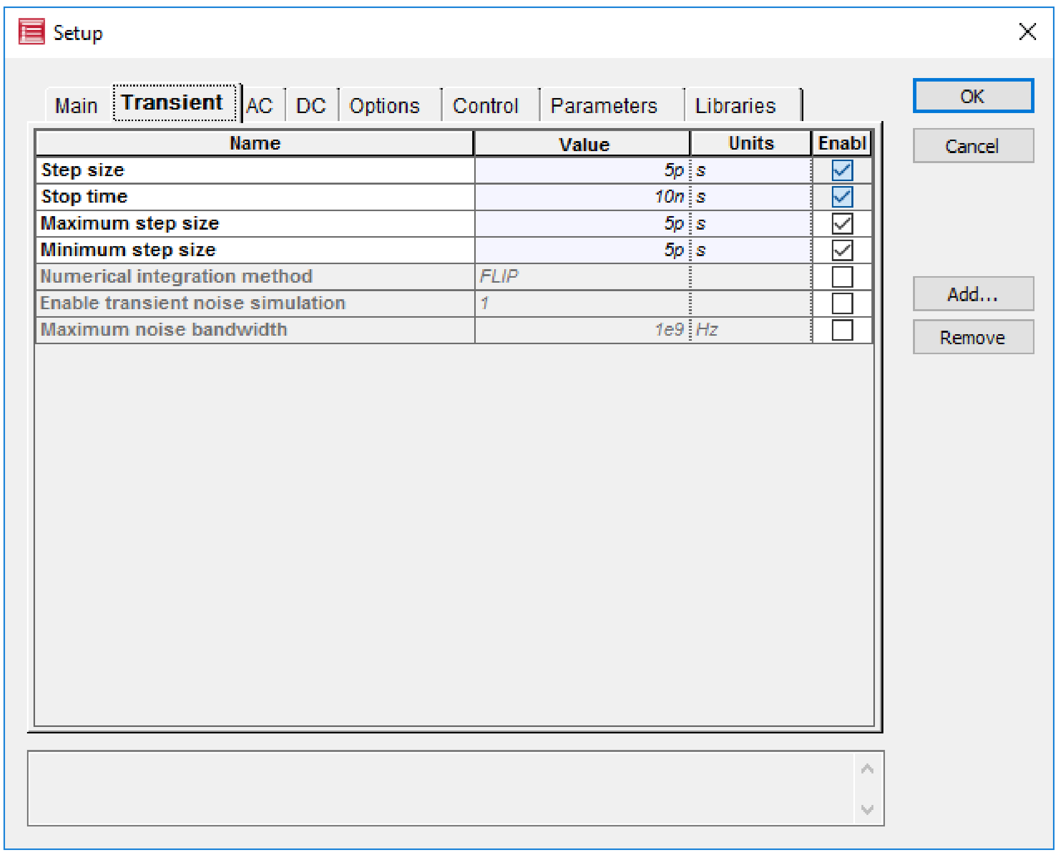
The simulation engine takes non uniform time steps by default (this ensures that the simulation runs more quickly where possible). This may however not work in all cases. If the simulation fails to converge, it is possible to make the simulation engine take smaller time steps by limiting the maximum and the minimum step size.

To regulate the minimum and maximum allowed time steps, enable the ***Minimum step size*** and ***Maximum step size*** parameters under the Transient tab of the Setup menu and set their values to be the same. For example (see Fig 1) if a simulation is setup with an initial time step (***Step size***) of 5 psec, set the minimum and maximum step sizes to 5 psec (or to a smaller value such as 1 psec). If this doesn't work, try to reduce the minimum and maximum step sizes even further.

> ***Note:*** The ***Minimum step size*** and ***Maximum step size*** parameters, when equal, will force the simulator to take uniform time steps. This may result in loss of resolution within portions of the transient simulation where there are rapid fluctuations in the waveform over time. To increase the resolution simply decrease the size of the minimum allowed and maximum allowed time steps.

If these changes do not fix the issue then contact Optiwave technical support at support@optiwave.com.

Figure 1 Setting the minimum and maximum allowed time steps



Undefined Phase

During the simulation of optical signals if the magnitude of a signal goes to zero it may cause convergence issues since the phase of the signal becomes ill-defined. Off-setting the electric field by a small amount (~1e-5) such that the magnitude is not equal to zero during simulation may solve this issue.

Equation parser

Certain functions in the equation parser may not work in the MS Windows environment. For example the square root operation (“sqrt”) may not be properly parse. In these cases it is recommended to use “^0.5” instead.

There may also be issues with long and complex expressions. It is thus recommended to keep the equations as short as possible by defining multiple parameters and associated expressions.

If these changes do not fix the issue then contact Optiwave technical support at support@optiwave.com.

User Interface

Unconnected ports

Sometimes the connections on the pins can be misleading where visually it looks like there is a connection but physically there is not. The connections can be double-checked by slightly moving the device. All the connections should be moving with the device.

Copy paste devices/sub-circuits

When a device or sub-circuit is copied and pasted into a new schematic, the model names and definitions are not carried over (the models will go back to their default settings in the new schematic). They have to be copied to the new schematic.

Creating Sub-circuits

Port and pin types are set by default to electrical. For optical connections they need to be defined as OPTICAL. This needs to be done in two separate places.

The ***Port type*** can be set while creating the sub-circuit after placing the ports. Right click on the port and then go to Attributes and the Type field and explicitly write OPTICAL in the text box.

Also the ***Pin type*** has to be set in the part editor. Right click on the pin and then go to Attributes and the Type field and write OPTICAL in the text box.

User defined parameters

It is recommended to define your parameters explicitly (.PARAM var1=...) in the SPICE command text box. This will make it easier to copy and paste parameter settings from one schematic/circuit design to another.

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