
Introductory Tutorials

This section contains the following introductory tutorials.

- [Lesson 1: Transmitter — External modulated laser](#)
- [Lesson 2: Subsystems — Hierarchical simulation](#)
- [Lesson 3: Optical Systems — WDM Design](#)
- [Lesson 4: Parameter Sweeps — BER x Input power](#)
- [Lesson 5: Bidirectional Simulation — Working with multiple iterations](#)
- [Lesson 6: Time-Driven Simulation — Working with individual samples](#)
- [Lesson 7: Optical Amplifiers — Designing optical fiber amplifiers and fiber lasers](#)
- [Lesson 8: Optical Systems — Working with multimode components](#)



Notes:

Lesson 1: Transmitter — External modulated laser

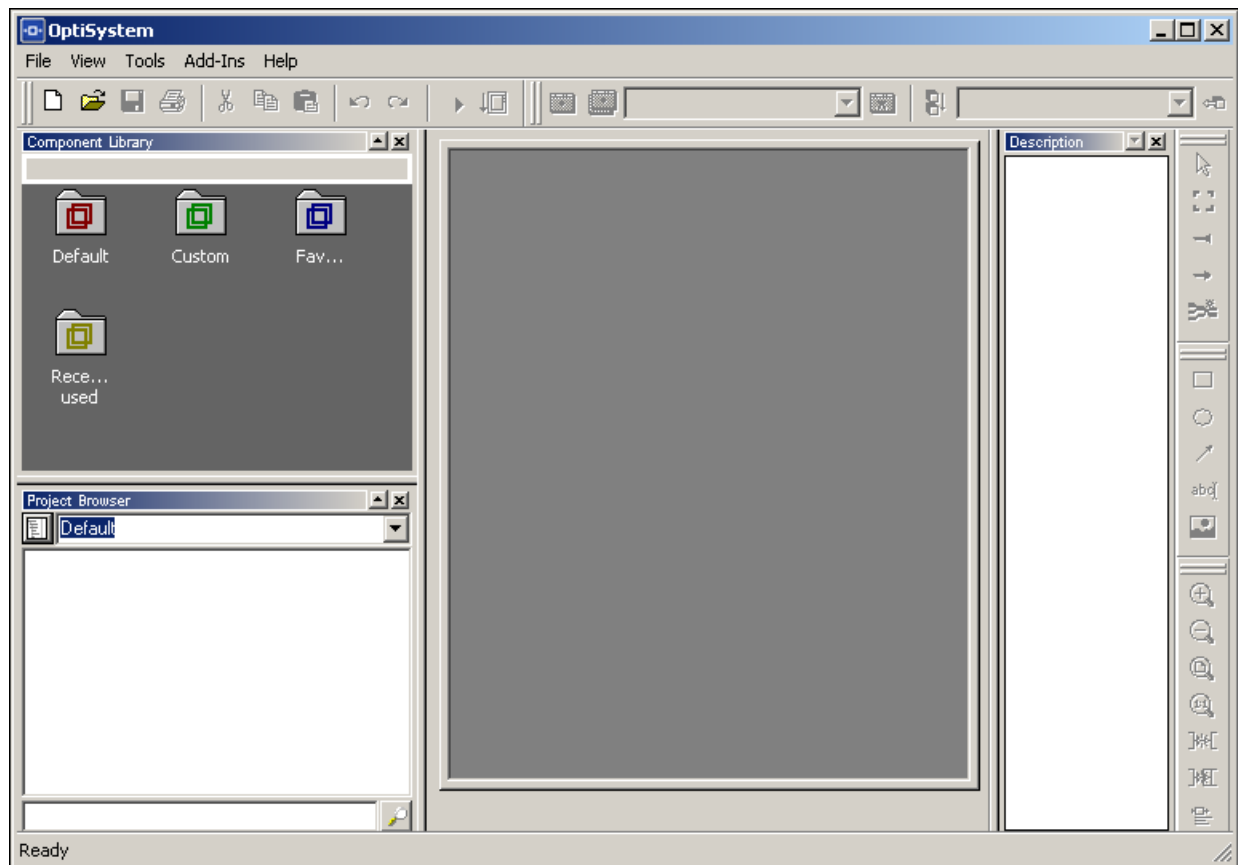
This lesson describes how to create a transmitter using an external modulated laser. You will become familiar with the Component Library, the **Main layout**, component parameters, and visualizers.

To start OptiSystem, perform the following procedure:

Starting OptiSystem

- | Step | Action |
|------|--|
| 1 | On the Taskbar, click Start . |
| 2 | Select Programs > Optiwave Software > OptiSystem 9> OptiSystem .
<i>OptiSystem opens and the graphical user interface appears (see Figure 1).</i> |

Figure 1 OptiSystem graphical user interface



Main parts of the GUI

The OptiSystem GUI contains the following main windows:

- [Project layout](#)
- [Dockers](#)
 - [Component Library](#)
 - [Project Browser](#)
 - [Description](#)
- [Status Bar](#)

Project layout

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

Figure 2 Project layout window

Name	Value
Layout 1	10
Global	
BER Analyzer	BER A
Optical Time Domain Visualizer	Optical
Optical Time Domain Visualizer...	Optical
Spatial Connector	Spatial
Spatial Optical Receiver	Spatial
Spatial Optical Transmitter	Spatial
Spatial Visualizer	Spatial
Spatial Visualizer_1	Spatial

Dockers

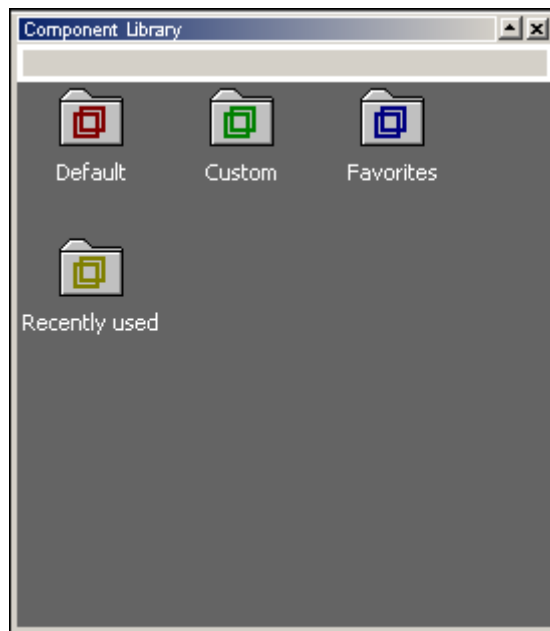
Use dockers, located in the **Main layout**, to display information about the active (current) project:

- [Component Library](#)
- [Project Browser](#)
- [Description](#)

Component Library

Access components to create the system design (see [Figure 3](#)).

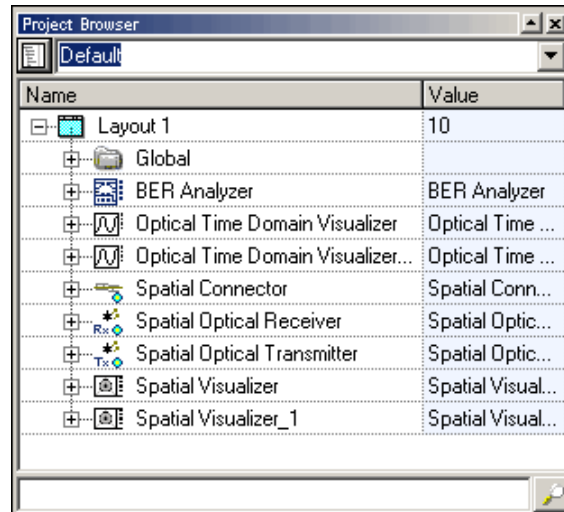
Figure 3 Component Library window



Project Browser

Organize the project to achieve results more efficiently, and navigate through the current project (see [Figure 4](#)).

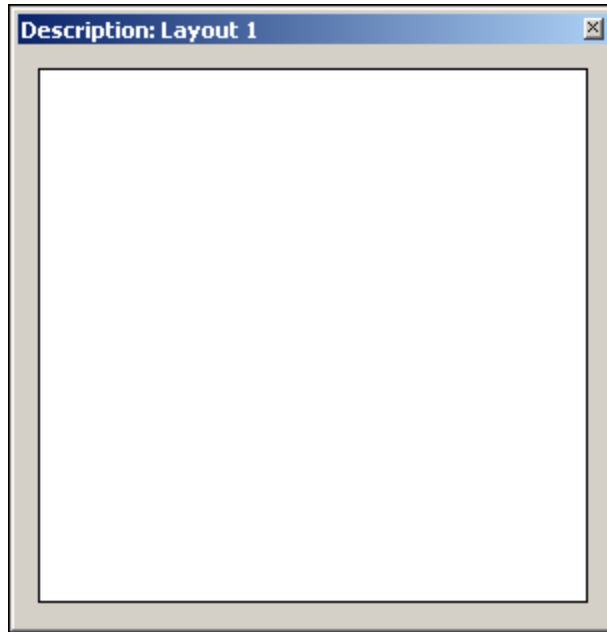
Figure 4 Project Browser window



Description

Display detailed information about the current project (see [Figure 5](#)).

Figure 5 Description window

**Status Bar**

Displays useful hints about using OptiSystem, and other help. Located below the **Project layout** window.

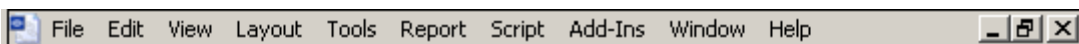
Figure 6 Status Bar

[CTRL] - Duplicate, [SHIFT] - Add to selection, [CTRL + SHIFT] Resize layout.

Menu bar

Contains the menus that are available in OptiSystem (see [Figure 7](#)). Many of these menu items are also available as buttons on the toolbars or from other lists.

Figure 7 Menu bar



Using the Component Library

In the following example, you design the external modulated transmitter. You will select components from the Component Library and place them in the **Main layout**.

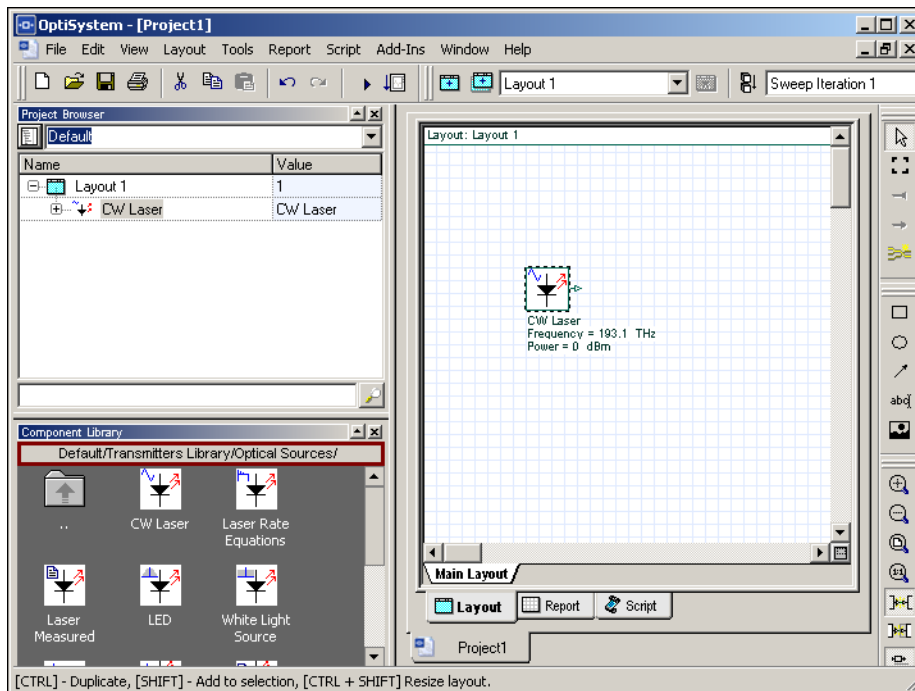
Note: OptiSystem provides a set of built-in default components.

To use the Component Library, perform the following procedure.

Step Action

- 1 To start a new project, from the **Main** toolbar, select **File > New**.
*A blank **Main layout** appears in the **Project layout** window.*
- 2 From the Component Library, select **Default > Transmitters Library > Optical Sources**.
- 3 Drag the **CW Laser** to the **Main layout** (see [Figure 8](#)).

Figure 8 Adding a CW Laser to the Main layout



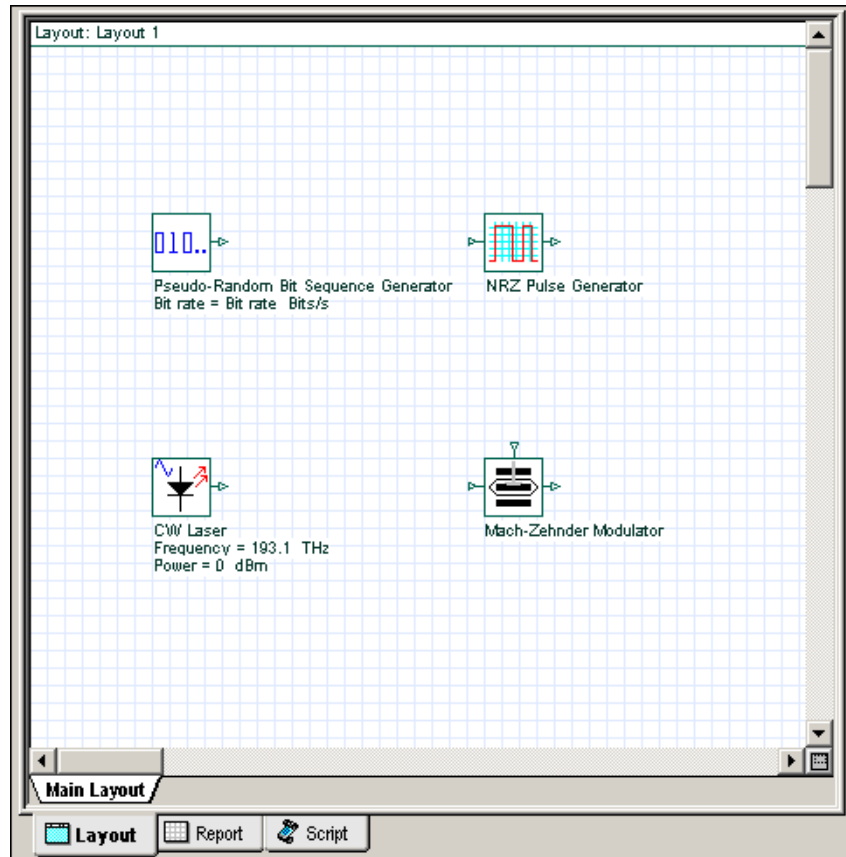
- 4 From the Component Library, select **Default > Transmitters > Modulators > Optical**.
- 5 Drag the **Mach-Zehnder Modulator** to the **Main layout** (see [Figure 9](#)).

Note: By default, the auto connect feature is on. When you place a component to the **Main layout**, the input port of the component connects automatically to the output port of a component that it can connect to. To turn the auto connect feature off, see “Turning the Auto connect feature off and on” on page 12.



- 6 From the Component Library, select **Default > Transmitters Library > Bit Sequence Generators**.
- 7 Drag the **Pseudo-Random Bit Sequence Generator** to the **Main layout**.
- 8 From the Component Library, select **Default > Transmitters > Pulse Generators > Electrical**.
- 9 Drag the **NRZ Pulse Generator** to the **Main layout** (see [Figure 9](#)).

Figure 9 Adding components to the Main layout



Auto connect feature

By default, the **Auto Connect on Drop** feature is active. There are two ways that components can auto connect:

- **Auto Connect on Drop:** When you place a component from the Component Library in the **Main layout**, the input port of the component connects automatically to the nearest output port of another component.
- **Auto Connect on Move:** When you move a component in the **Main layout**, the input port of the component connects automatically to the nearest output port of another component.

Turning the Auto connect feature off and on

To turn the **Auto connect** feature off and on, perform the following procedure.

- | Step | Action |
|------|---|
| 1 | To turn the Auto connect feature off, click the active Auto Connect on Drop button and the Auto Connect on Move button on the Layout Operations toolbar.
<i>The buttons are inactive (see Figure 10) and the components no longer connect automatically to each other.</i> |

Figure 10 Inactive Auto connect buttons



- | | |
|---|--|
| 2 | To turn the auto connect feature back on, click the inactive Auto Connect on Drop button and the Auto Connect on Move button on the Layout Operations toolbar.
<i>The buttons are active (see Figure 11) and the components connect automatically to each other.</i> |
|---|--|

Figure 11 Active Auto connect buttons



Connecting components manually

The only connectable ports are those which have the same type of signal being transferred between them.

The exception to this rule is the ports that can be added to a sub-system and certain components in the library that have ports, which support any type of signal (for example, **Forks**).

Note: You can only connect output to input ports and vice versa.

Figure 12 Rubber Band cursor



The rubber band cursor appears when you place the cursor over a port.

To connect components using the layout tool, perform the following procedure.

Step Action

- 1 Place the cursor over the initial port.
The cursor changes to the rubber band cursor (chain link) (see Figure 12). A tool tip appears that indicates the type of signal that is available on this port (see Figure 13).
- 2 Click and drag to the port to be connected.
The ports are connected.

Figure 13 Connecting components symbol

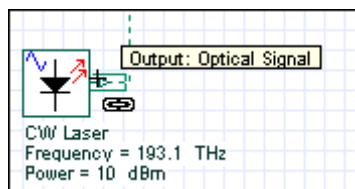
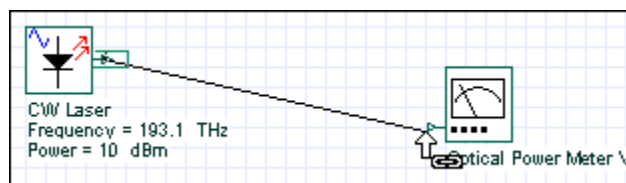


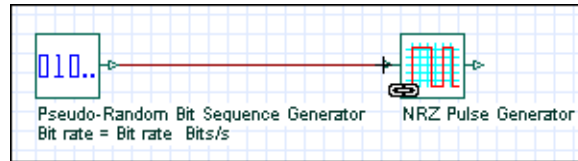
Figure 14 Connecting ports



- To connect the components, click on the port of one component and drag the connection to the port of a compatible component (see Figure 16).
 - a. Connect the **Pseudo-Random Bit Sequence Generator** output port to the **NRZ Pulse Generator Bit Sequence** input port.

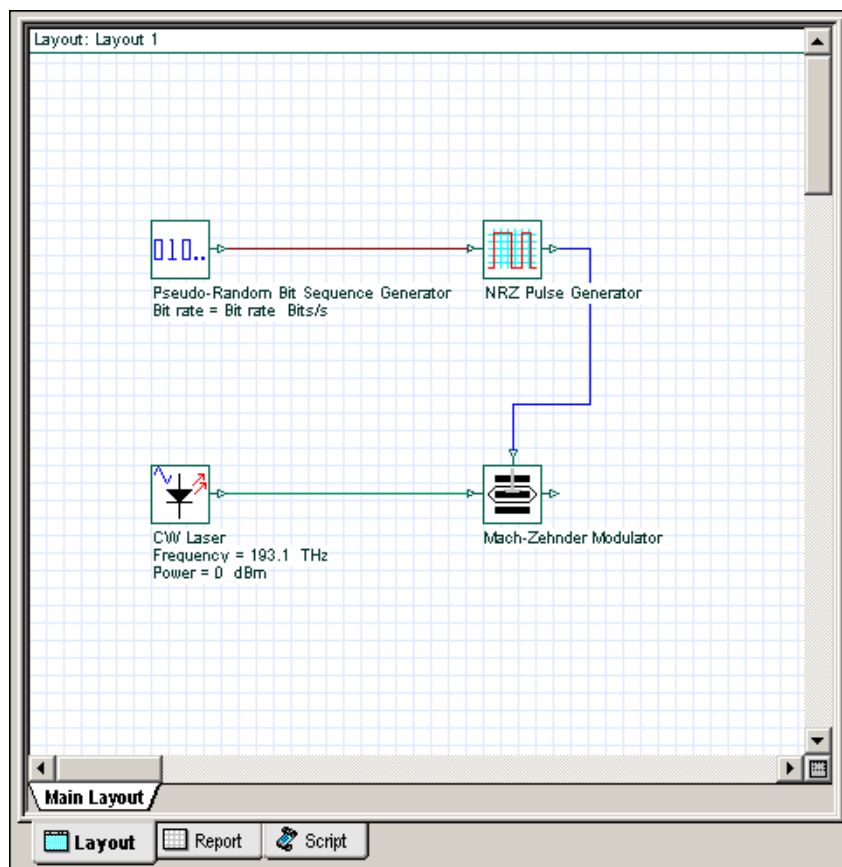


Figure 15 Connecting components



- b. Connect the **NRZ Pulse Generator** output port to the available **Mach-Zehnder Modulator** input port.
- c. Connect the **CW Laser** output port to the **Mach-Zehnder Modulator** input port.

Figure 16 Connecting components



Visualizing the results

OptiSystem allows you to visualize simulation results in a number of ways. The **Visualizer Library** folder in the Component Library allows you to post process and display results from a simulation. The visualizer is categorized as an electrical or optical visualizer according to the input signal type.

For example, to visualize the electrical signal generated by the **NRZ Pulse Generator** in time domain, use an **Oscilloscope Visualizer**.

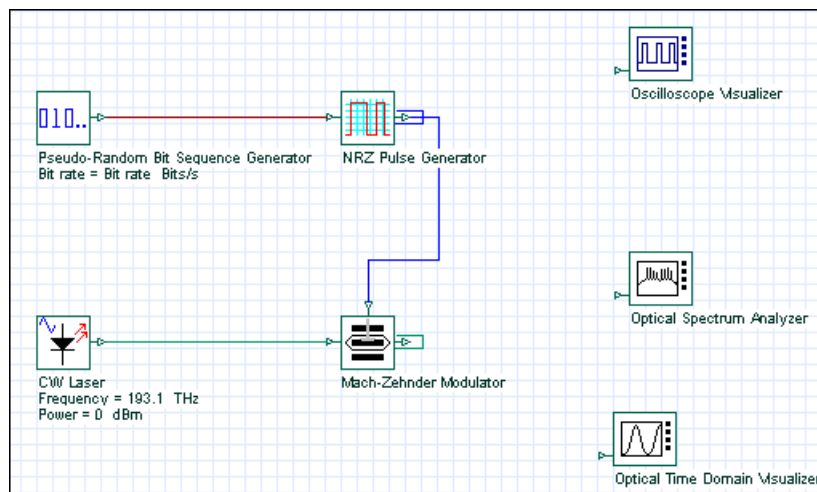


Visualizing the results

To visualize the results, perform the following procedures.

- | Step | Action |
|------|--|
| 1 | From the Component Library, select Default > Visualizer Library > Electrical . |
| 2 | Drag the Oscilloscope Visualizer to the Main layout .
The optical signal can be also displayed by selecting visualizers from the library. For example, use an Optical Spectrum Analyzer and an Optical Time Domain Visualizer to visualize the modulated optical signal in time domain. |
| 3 | From the Component Library, select Default > Visualizer Library > Optical . |
| 4 | Drag the Optical Spectrum Analyzer to the Main layout . |
| 5 | Drag the Optical Time Domain Visualizer to the Main layout (see Figure 17). |

Figure 17 Visualizers in Main layout



Connecting visualizers

To visualize the signal from one component, you must connect the component output port to the visualizer input port.

You can connect more than one visualizer to one component output port. As a result, you can have multiple visualizers attached to the same component output port.

To connect visualizers, perform the following action.

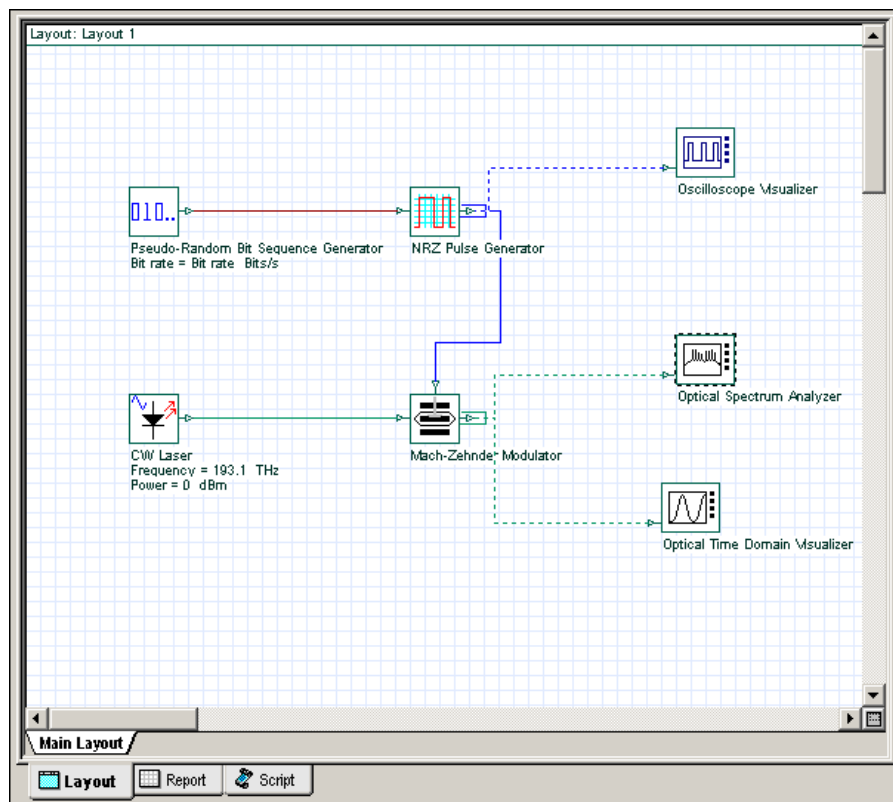
Action

- To connect the component and visualizers, click on the output port of the component and drag it to the input port of the visualizer (see [Figure 18](#)).

Note: You can only connect one component to a visualizer input port.

- Connect the **NRZ Pulse Generator** output to the **Oscilloscope Visualizer** input port.
- Connect the **Mach-Zehnder Modulator** output to the **Optical Spectrum Analyzer** input port and to the **Optical Time Domain Visualizer** input port.

Figure 18 Connecting components to visualizers

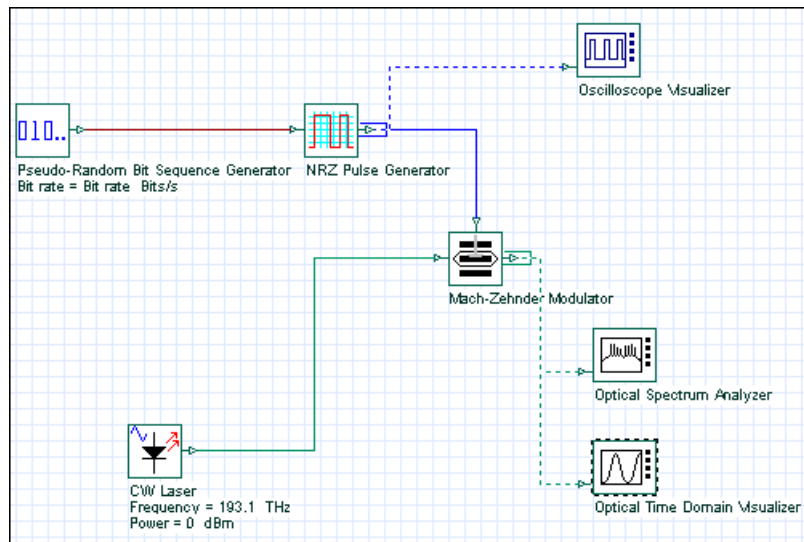


Visualizers and data monitors

When you connect a visualizer to a component output port, OptiSystem inserts a default data monitor to the component output port. You can then connect the component to the visualizer.

Note: Visualizers always connect to monitors. You can have multiple visualizers attached to the same component output port, because they are actually attached to the monitor, not to the component (see the **Mach-Zehnder Modulator** with connections to the OTDV and the OSD in [Figure 19](#)). Data monitors are represented by a rectangle around the component output port (see [Figure 19](#) for examples of monitors on components).

Figure 19 Visualizers and monitors



Connecting a monitor to a port

Visualizers post-process the data that the data monitor saves. You can connect a monitor to a port without connecting to a visualizer, and the monitor will save the data after the simulation ends. If you insert a monitor into a port before the simulation starts, you can connect visualizer to this monitor after the simulation ends without having to run the simulation again.

To connect a monitor to a port, perform the following procedure.

Step	Action
-------------	---------------

- | | |
|----------|--|
| 1 | From the Layout Tools toolbar, select the Monitor Tool .
<i>The cursor changes to the Monitor tool.</i> |
| 2 | Click the CW Laser output port. |
| 3 | To disable the Monitor tool , on the Layout Tools toolbar, click the Layout tool .
<i>The monitor saves the signals at the CW Laser output.</i> |

Note: If a port already has a monitor, clicking the port with the **Monitor tool** removes the existing monitor.



Component parameters

Viewing and editing component properties

To view the properties of a component, perform the following action.

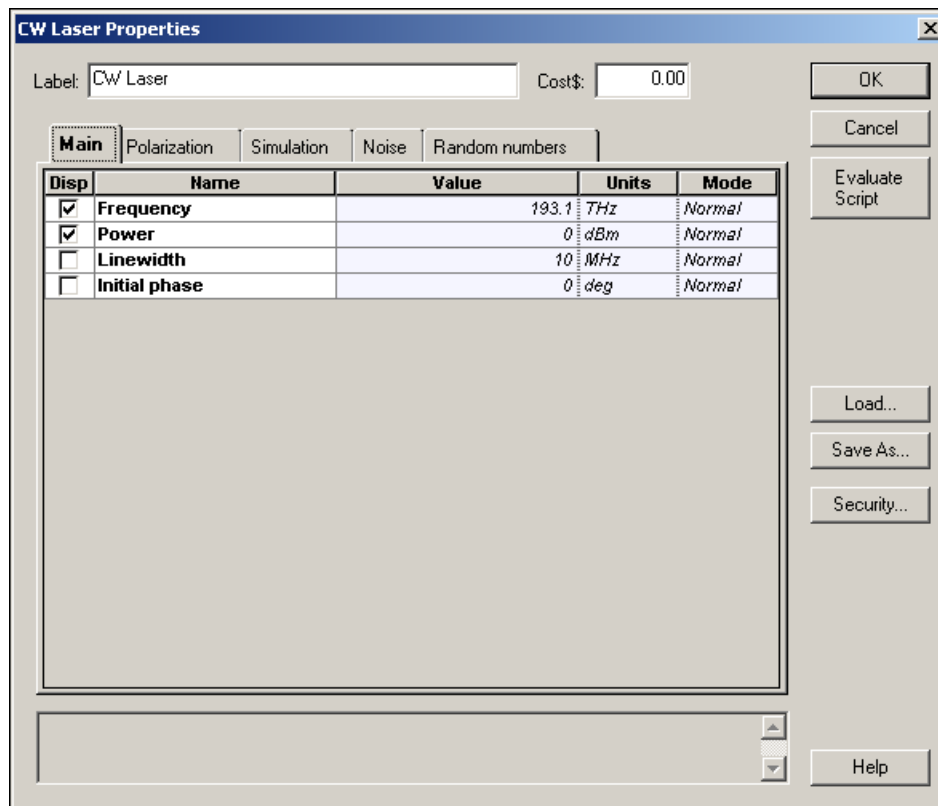
Action

- In the **Main layout**, double click the **CW Laser**.
The **CW Laser Properties** dialog box appears (see [Figure 20](#)).

Component parameters are organized by categories. The **CW Laser** has five parameter categories.

- Main — includes parameters for accessing a laser (Frequency, Power, Line width, Initial phase)
- Polarization
- Simulation
- Noise
- Random numbers

Figure 20 Component parameters



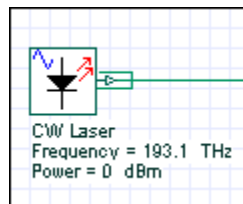
Each category has a set of parameters. Parameters have the following properties:

- Disp
- Name
- Value
- Units
- Mode

Display parameters in the layout

The first parameter property is **Disp**. When you select this property, the parameter name, value, and unit appear in the **Main layout**. For example, if you select **Disp** for **Frequency** and **Power**, these parameters appear in the **Main layout** (see [Figure 21](#)).

Figure 21 Displaying parameters in the Main layout

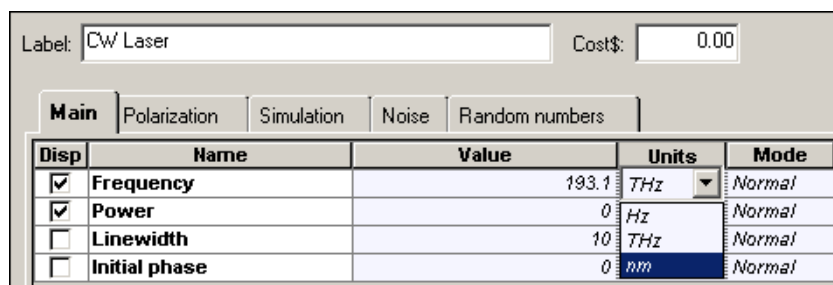


Parameter units

Some parameters, such as **Frequency** and **Power**, can have multiple units. **Frequency** can be in **Hz**, **THz** or **nm**, and **Power** can be in **W**, **mW**, or **dBm** (see [Figure 22](#)). The conversion is automatic.

Note: You must press **Enter** or click in another cell to update the values.

Figure 22 Parameter units



Changing parameter modes: arithmetic expressions

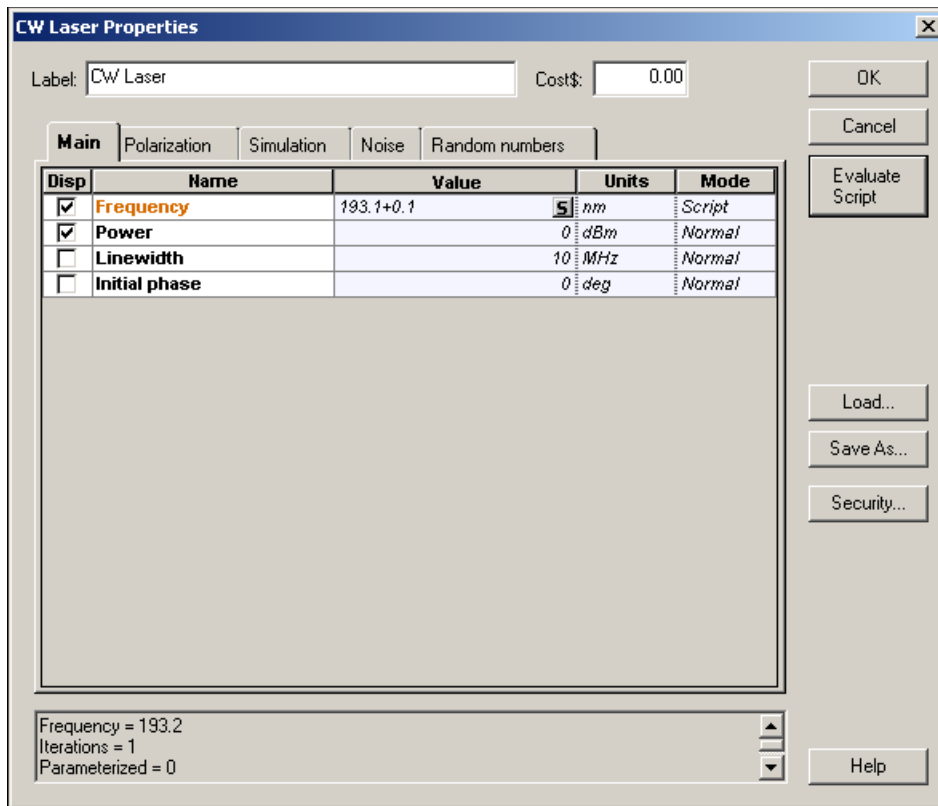
Each parameter can have three modes: Normal, Sweep and Script. Script mode allows you to enter arithmetic expressions and to access parameters that are defined globally.



To change parameter modes, perform the following procedure.

- | Step | Action |
|------|---|
| 1 | In the Main layout , double-click the CW Laser .
<i>The CW Laser Properties dialog box appears (see Figure 23).</i> |
| 2 | Click in the Mode cell beside Frequency .
<i>A drop-down list appears.</i> |
| 3 | From the drop-down list, select Script . |
| 4 | Click in the Value cell beside Frequency . |
| 5 | In the Parameter Script Editor dialog box, type:
193.1+0.1 |
| 6 | Click Evaluate , and OK . |
| 7 | Click Evaluate Script .
<i>The value of the script expression appears in the message window at the bottom of the CW Laser Properties dialog box. The Frequency result is 193.200000.</i> |
| 8 | To save the settings and return to the Main layout , click OK . |

Figure 23 Scripted parameters



Accessing visualizer parameters

To access the parameters for a visualizer, perform the following procedure.

Note: You must right-click on a visualizer to access the parameters. Double-clicking on a visualizer brings up a dialog box to visualize the graphs and results generated during the simulation, not the parameters.

- | Step | Action |
|------|--|
| 1 | In the Main layout , select and right-click on the Optical Spectrum Analyzer . A context menu appears (see Figure 24). |
| 2 | From the context menu, select Component Properties . The Optical Spectrum Analyzer Properties dialog box appears (see Figure 25). |

Figure 24 Layout menu

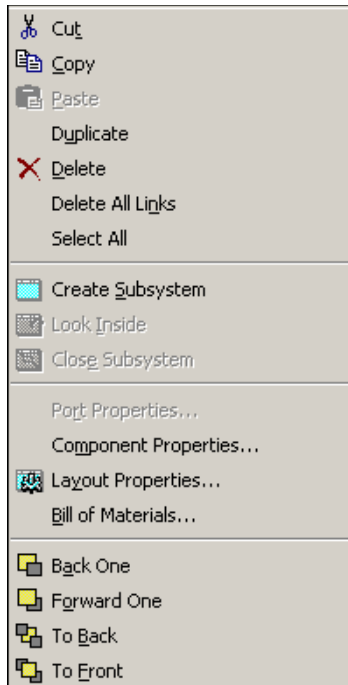
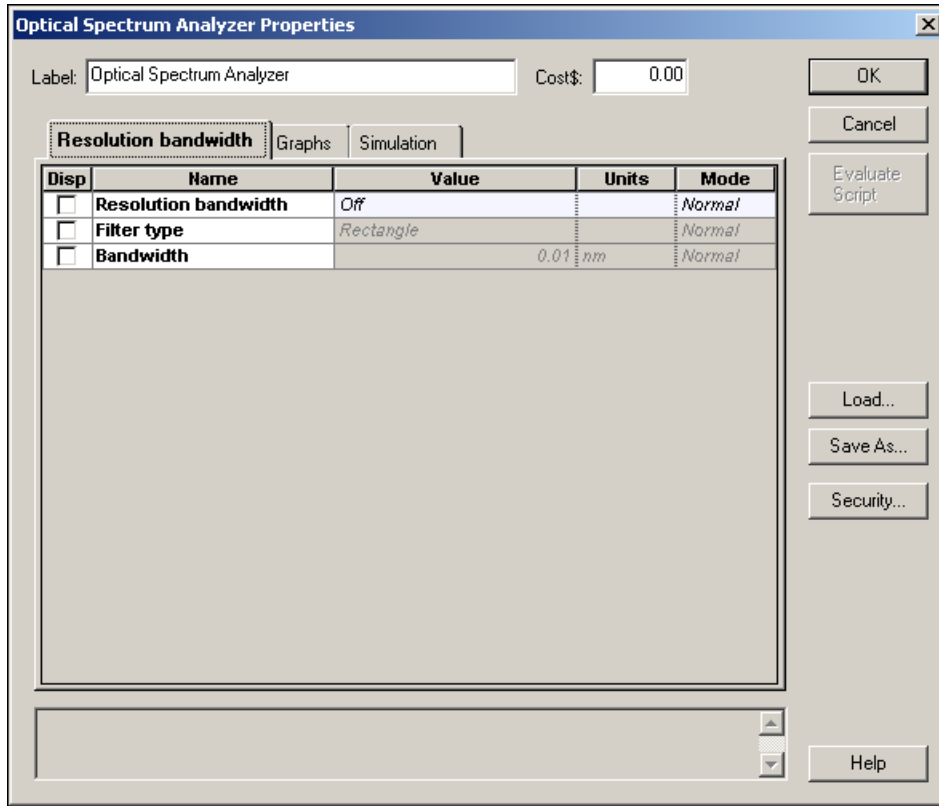


Figure 25 Visualizers parameter dialog



- 3 To save the settings and return to the **Main layout**, click **OK**.

Running the simulation

OptiSystem allows you to control the calculation in three different ways:

- **Calculate the whole project:** all sweep iterations for multiple layouts
- **Calculate all sweep iterations in the active layout:** all sweep iterations for the current layout
- **Calculate current sweep iteration:** current sweep iteration for the current layout

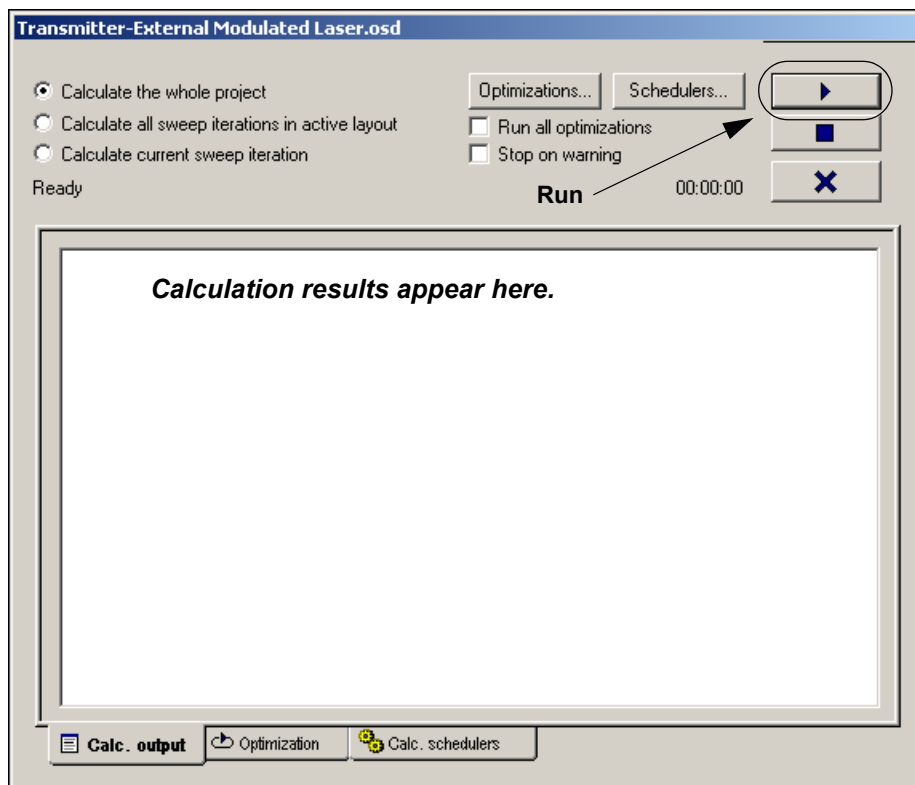
By default, you will calculate the whole project, since there are currently no multiple layouts and no sweep iterations.

To run a simulation, perform the following procedure.

Step Action

- 1 From the **File** menu, select **Calculate**.
*The **OptiSystem Calculations** dialog box appears (see [Figure 26](#)).*
- 2 In the **OptiSystem Calculations** dialog box, click **Run** to start the simulation (see [Figure 26](#)).
*The calculation output appears in the dialog box, and the simulation results appear below the components that were included in the simulation in the **Main layout**.*

Figure 26 Running the simulation



Saving the simulation results

OptiSystem allows you to save the data from the monitors. This allows you to save the project file with the signals from the monitors. Next time you load the file, the visualizers will recalculate the graphs and results from the monitors.

To save the simulation results, perform the following procedure.

Step Action

- 1 After the calculation ends, from the **Tools** menu, select **Options**.
*The **Options** dialog box appears (see [Figure 27](#)).*
- 2 Select **Save Monitor Data**, and click **Save**.
The results are saved with the project file.
- 3 Select **File > Calculate**.
*The **Optisystem Calculations** dialog box appears.*
- 4 In the **OptiSystem Calculations** dialog box, click **Run** to start the simulation (see [Figure 26](#)).
- 5 When the calculation is finished, select **File > Save As**.
*The **Save As** dialog box appears (see [Figure 28](#)).*
- 6 Save the project.

Note: When saving monitors the project file size may become very large. There is also a delay when reloading the project, due to the fact that visualizers will recalculate during the loading of the file.

Figure 27 Options dialog box—Save Monitor Data selection

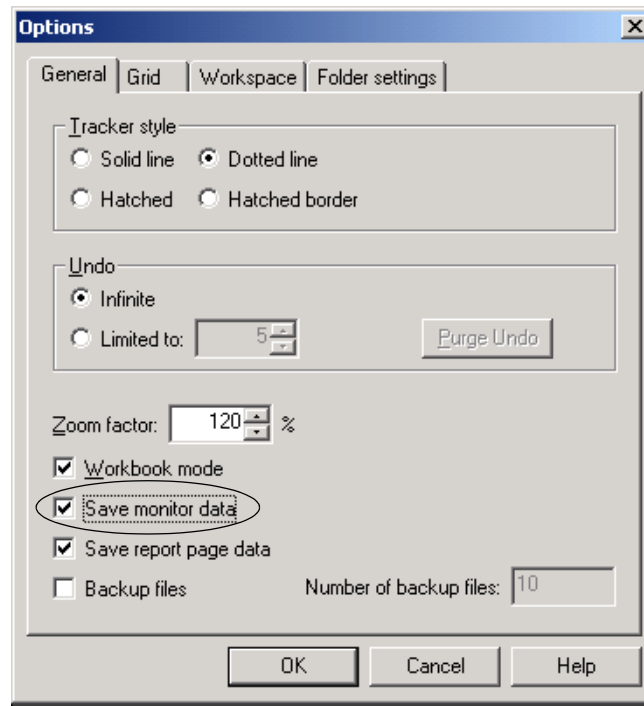
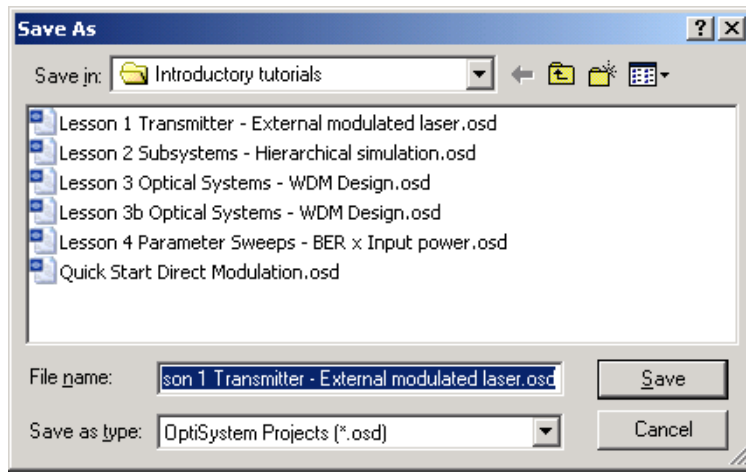


Figure 28 Save As dialog box



Displaying results from a visualizer

To display results from a visualizer, perform the following action.

Action

- Double-click a visualizer to view the graphs and results that the simulation generates.

Note: Double-click again to close the dialog box.

Oscilloscope

To visualize electrical signals in time domain with an oscilloscope (see [Figure 29](#)), perform the following action.

Action

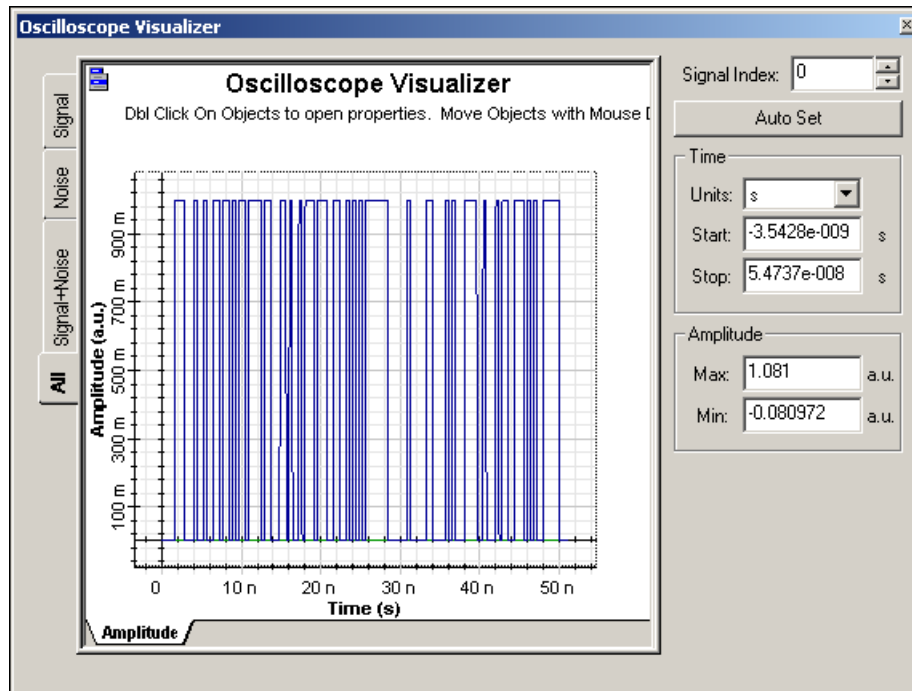
- Double-click the **Oscilloscope Visualizer**.
*The **Oscilloscope Visualizer** dialog box appears.*



Since OptiSystem can propagate the signal and noise separately, you can visualize the results separately. Use the tabs on the left side of the graph to select the representation that you want to view.

- Signal
- Noise
- Signal + Noise
- All

Figure 29 Oscilloscope



Optical Spectrum Analyzer

To visualize optical signals in frequency domain with an **Optical Spectrum Analyzer** (see [Figure 30](#)), perform the following action.

Action

- Double-click the **Optical Spectrum Analyzer**.
*The **Optical Spectrum Analyzer** dialog box appears.*

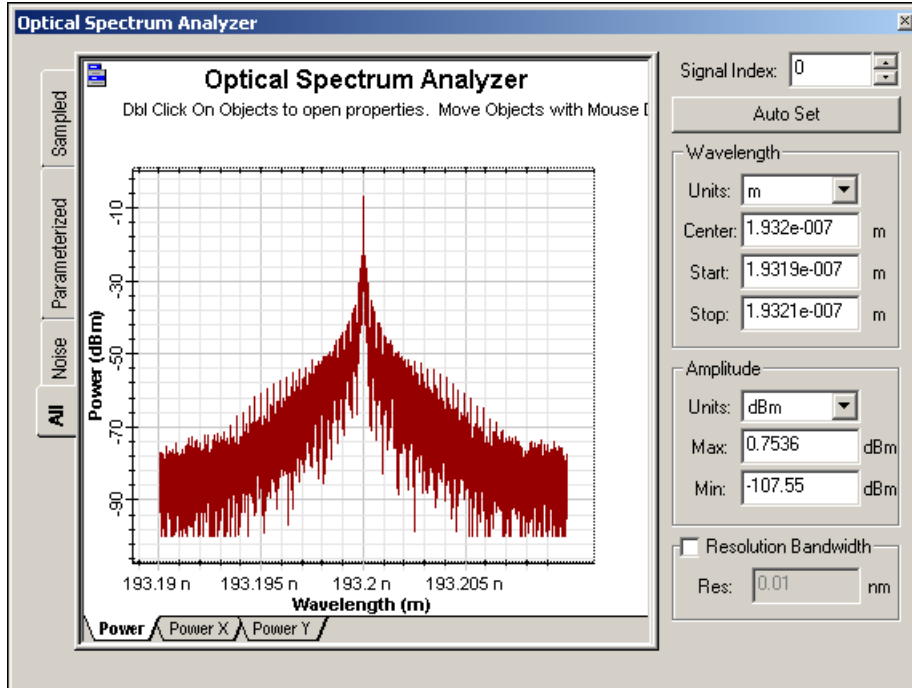
Since **OptiSystem** uses a mixed signal representation, you can visualize the signal according to the representation. Use the tabs on the left side of the graph to select the representation that you want to view.

- Sampled
- Parameterized
- Noise
- All

To access the optical signal polarization, use the tabs at the bottom of the graph.

- **Power:** Total power
- **Power X:** Power from polarization X
- **Power Y:** Power from polarization Y

Figure 30 OSA



Optical Time Domain Visualizer

To visualize optical signals in time domain with an **Optical Time Domain Visualizer** (see [Figure 31](#)), perform the following procedure.

Action

- Double-click the **Optical Time Domain Visualizer**.
*The **Optical Time Domain Visualizer** dialog box appears.*

In time domain, **OptiSystem** translates the optical signal and the power spectral density of the noise to numerical noise in time domain. Use the tabs at the bottom of the graph to select the representation that you want to view.

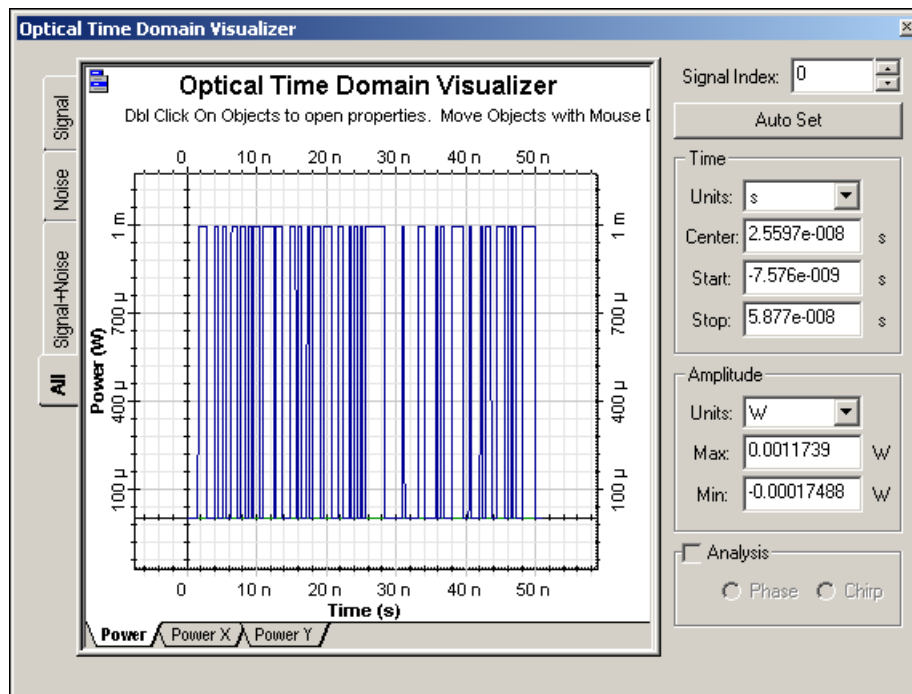
Power: Total power

Power X: Power from polarization X

Power Y: Power from polarization Y

Note: When you select polarization X or Y, you can also select to display the phase or chirp of the signal for that particular polarization.

Figure 31 Optical Time Domain Visualizer



Info-Window

When you open the **Info-Window**, it appears in the work area of a visualizer graph. By default, the **Info-Window** displays the current position (in database coordinates) of the cursor. When you zoom, pan, or trace the graph, the coordinates in the **Info-Window** change to reflect the coordinates of the cursor.

Accessing the Info-Window

To access the **Info-Window**, perform the following procedure.

Step Action

- 1 Double-click a visualizer to view the graph.
- 2 To open the **Info-Window**, right-click anywhere on the graph and select the **Info** tool from the graph toolbox that appears (see [Figure 32](#)),

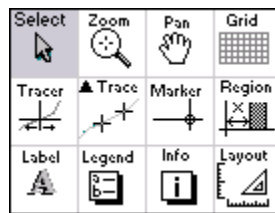
or

Click the **Graph Menu** button (top left corner of graph) and select **InfoWindow**.

*The **Info-Window** appears beside the graph.*

Note: Enlarge the graph window to increase the size of the graph.

Figure 32 Graph toolbox



Zoom, Pan, and Trace tools

The **Zoom**, **Pan**, and **Trace** tools help you analyze and extract information from the graphs.

Zooming into a graph

To zoom into a graph, perform the following procedure.

- | Step | Action |
|-------------|--|
| 1 | To open the Optical Spectrum Analyzer display, double-click the Optical Spectrum Analyzer . |
| 2 | Open the Info-Window (see Accessing the Info-Window). |
| 3 | To access the Zoom tool , right-click anywhere on the graph and select the Zoom tool from the graph toolbox that appears,
or
Click the Graph Menu button and select Zoom .
<i>The cursor changes into a magnifying glass.</i> |
| 4 | Click and drag the magnifying glass cursor over the area of the graph that you want to zoom into.
<i>The cursor coordinates appear in the Info-Window.</i>
Note: By default, the zoom operation zooms into the center of the center of the grid. |
| 5 | Double-click anywhere on the graph to return to the original size. |

Panning a graph

To pan a graph and view a specific area, perform the following procedure.

- | Step | Action |
|-------------|---|
| 1 | To open the Optical Spectrum Analyzer display, double-click the Optical Spectrum Analyzer . |
| 2 | Open the Info-Window (see Accessing the Info-Window). |
| 3 | To access the Pan tool, right-click anywhere on the graph and select the Pan from the graph toolbox that appears.
or
Click the Graph Menu button and select Pan .
<i>The cursor changes into a hand.</i> |
| 4 | Use the Pan tool to move the graph and view a specific area of the graph.
<i>The cursor coordinates appear in the Info-Window.</i> |



Tracing a graph

To trace a graph and obtain the values for each point in the graph, perform the following procedure.

Step	Action
-------------	---------------

- | | |
|----------|--|
| 1 | To open the Optical Spectrum Analyzer display, double-click the Optical Spectrum Analyzer . |
| 2 | Open the Info-Window (see Accessing the Info-Window). |
| 3 | To access the Tracer tool, right-click anywhere on the graph and select Tracer from the graph toolbox that appears.

or

Click the Graph Menu button and select Tracer .
<i>The cursor changes into a double-headed arrow</i> |
| 4 | To view the X and Y line coordinates, click anywhere in the graph.
<i>The X and Y line coordinates appear.</i> |
| 5 | Move the X and Y line coordinates to the position on the graph curve that you want to view the coordinates for.
<i>The cursor and Tracer coordinates appear in the Info-Window.</i> |



Saving graphs

You can obtain the table of points with the values for each point in the graph and then save this as a text file. Copy the graph to the clipboard as a bitmap, or export the graph in different file formats — for example, metafile or bitmap.

To save a graph, perform the following procedure.

- | Step | Action |
|------|---|
| 1 | From the Graph Menu button, select Export Data .
<i>The Data Table dialog box appears.</i> |
| 2 | To save the data as a text file, click Export Curve .
<i>The Save As dialog box appears (see Figure 33).</i> |
| 3 | Select the folder where you want to save the data. |
| 4 | Type a name for the file, and click Save . |
| 5 | To return to the graph, click OK . |

Figure 33 Graph menu dialog

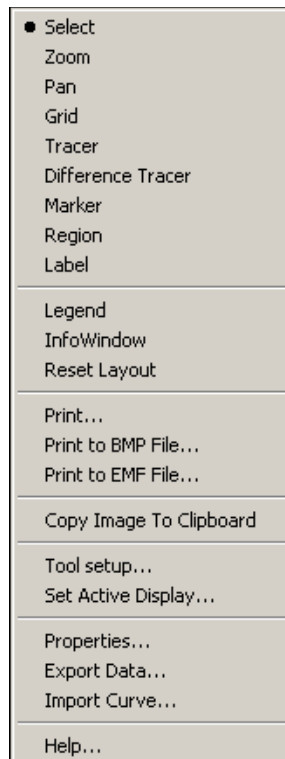
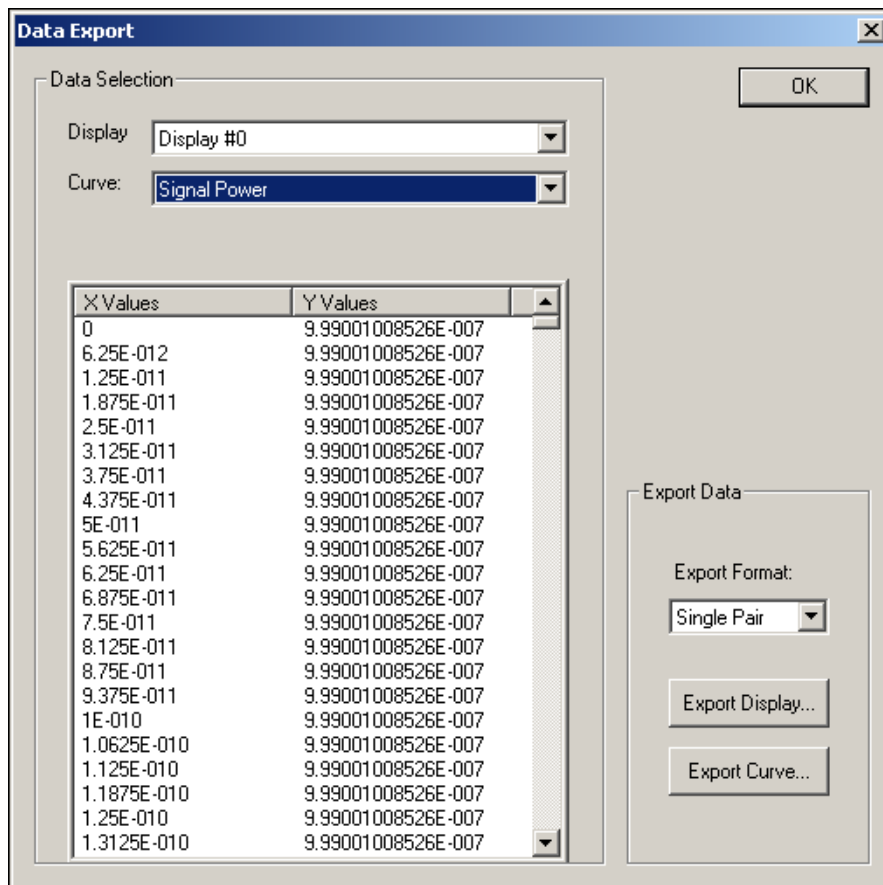


Figure 34 Data Export dialog



Notes:



LESSON 1: TRANSMITTER — EXTERNAL MODULATED LASER

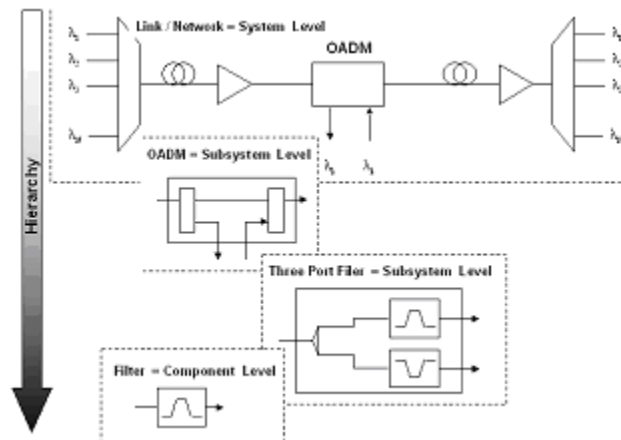


Lesson 2: Subsystems — Hierarchical simulation

A subsystem is like a component — it has an icon, parameters, and input and output ports. You build a subsystem using a group of components or other subsystems. You can easily create a subsystem by grouping selected components in the layout (see [Figure 1](#)).

Subsystems help you to create your own components based on the Component Library without programming, and to organize the layout in different hierarchical levels when there are a large number of components in different levels.

Figure 1 Hierarchical simulation



This lesson describes how to create a subsystem using the External Modulated Laser subsystem from [Lesson 1: Transmitter — External modulated laser](#). You will become familiar with subsystems and the Component Library.

Loading a sample file

To load a sample file, perform the following procedure.

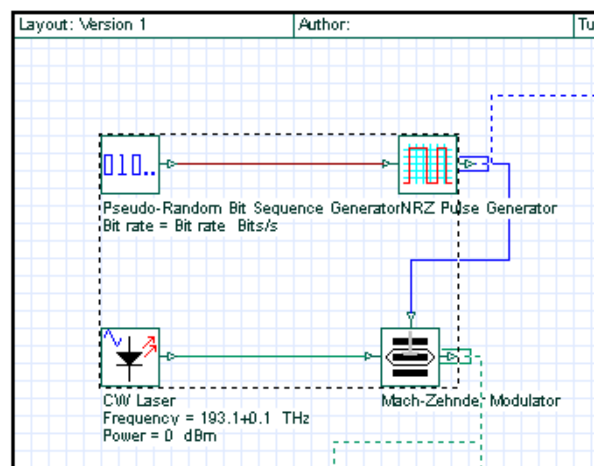
- | Step | Action |
|------|--|
| 1 | From the File menu, select Open . |
| 2 | From your C: drive, select Program Files > Optiwave Software > OptiSystem 9 > samples > Introductory tutorials > "Lesson 2 Subsystems - Hierarchical simulation.osd" . |

Creating a subsystem

To create a subsystem, perform the following procedure.

- | Step | Action |
|------|--|
| 1 | In the Main layout , select the components that you want to include in the subsystem.
<i>A box appears around the selected components (see Figure 2)..</i> |
| 2 | Right-click the selection.
<i>A context menu appears (see Figure 24 from Lesson 1).</i> |
| 3 | From the context menu, select Create Subsystem .
<i>The subsystem appears in a glass box. When you look inside the subsystem, a subsystem tab appears at the bottom of the Main layout (see Figure 3).</i> |
- Note:** The visualizers that are not included in the selection are disconnected. The subsystem does not add additional ports to connect the visualizers.

Figure 2 Creating a subsystem—Selected components in the Main layout



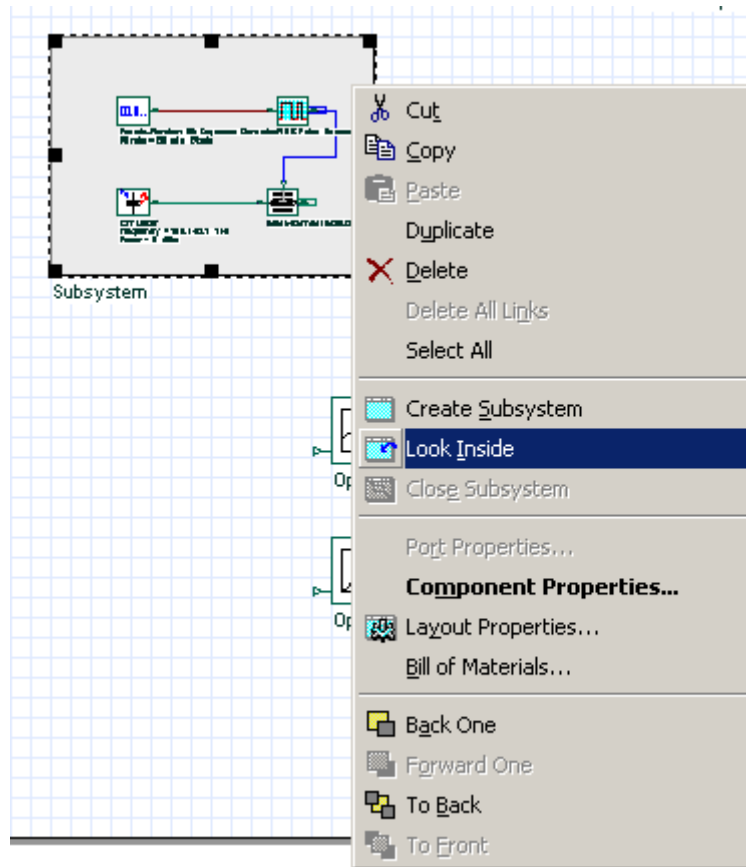
Looking inside a subsystem

To look inside a subsystem, perform the following procedure.

- | Step | Action |
|------|--|
| 1 | Select the subsystem glass box and right-click.
<i>A context menu appears (see Figure 3)</i> |
| 2 | From the context menu, select Look Inside .
<i>The subsystem opens and a Subsystem tab appears beside the Main layout tab at the bottom of the subsystem layout window (see Figure 3).</i> |



Figure 3 Looking inside a subsystem



- 3 To return to the **Main layout** and close the subsystem, right-click in the subsystem layout and select **Close Subsystem** from the context menu (see [Figure 24](#) from Lesson 1),
- or
- To return the **Main layout** and leave the subsystem open, click the **Main Layout** tab.


Creating a subsystem output port

You can change parameters and create ports in order to access the signals from the subsystem.

- | Step | Action |
|-------------|---|
| 1 | Select the subsystem glass box and right-click.
<i>A context menu appears.</i> |
| 2 | From the context menu, select Look Inside .
<i>The subsystem opens.</i> |
| 3 | From the Layout Tools toolbar, select Draw – Output Port Tool .
<i>The cursor changes into a cross-hatch.</i> |
| 4 | Position the cursor on the edge of the subsystem layout where you want to add the output port.
<i>The cross-hatch changes into an output port.</i> |
| 5 | Click to place the output port.
<i>The output port is created (see Figure 5).</i> |

Connecting a component output port to a subsystem output port

To connect the Mach-Zehnder Modulator output port to the subsystem output port, perform the following procedure.

- | Step | Action |
|-------------|---|
| 1 | From the Layout Tools toolbar, select Layout Tool  .
<i>The cursor changes into the Layout Tool.</i> |
| 2 | Click the Mach-Zehnder output port.
<i>The cursor changes into a link (see Figure 4).</i> |
| 3 | To connect the Mach-Zehnder output port to the subsystem output port, drag the cursor from the Mach-Zehnder output port to the subsystem output port (see Figure 5).
<i>The output ports are connected.</i> |
| 4 | Click the Main Layout tab to return to the Main layout .
<i>The output port appears in the subsystem.</i> |

Note: Use the same procedure to create input ports. Because this is a transmitter, you will not create input ports in this lesson.



Figure 4 Link tool in the subsystem

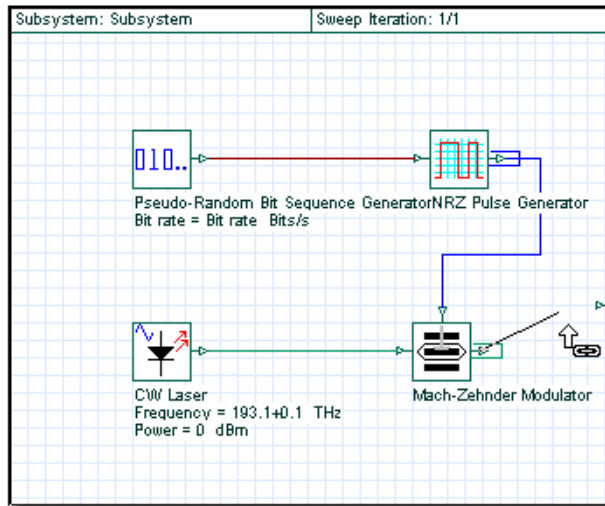
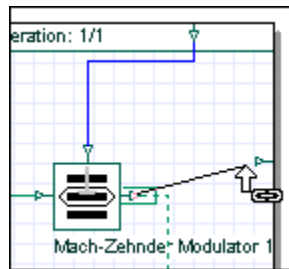


Figure 5 Linking output ports



Changing the port position and location

To change the port position and location, perform the following procedure.

Step Action

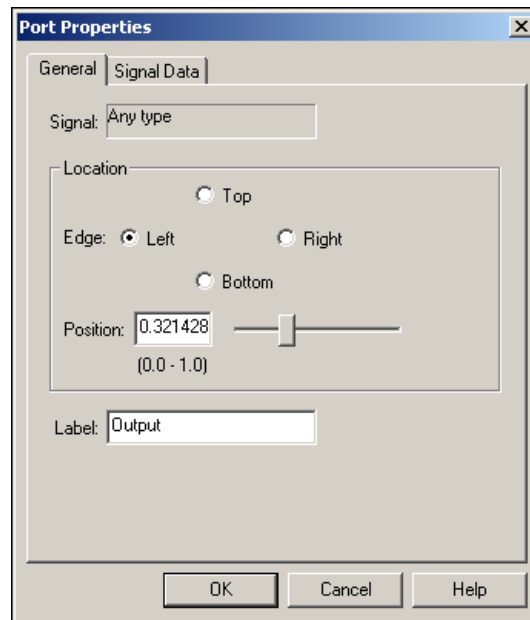
- 1** In the **Main layout**, double-click the subsystem port,
or

In the subsystem, double-click the port.
*The **Port Properties** dialog box appears (see [Figure 6](#)).*

- 2** Select the position in the subsystem that you want to move the port to:
 - Top
 - Bottom
 - Left
 - Right

- 3 Move the slider to specify the exact position that you want to select.
- 4 To return to the **Main layout**, click **OK**.

Figure 6 Port properties



Subsystem properties — name and icon

You can change the name and add an icon for a subsystem by changing the subsystem parameters.

Changing a subsystem name

To change the subsystem name, perform the following procedure.

Step Action

- 1 In the **Main layout**, double-click on the subsystem.
*The **Subsystem Properties** dialog box appears.*
- 2 In the **Label** field, type **External Modulated Transmitter** (see [Figure 7](#)).
- 3 To return to the **Main layout**, click **OK**.
The new subsystem name appears below the subsystem glass box and on the subsystem tab (see [Figure 8](#)).

Figure 7 Subsystem properties dialog

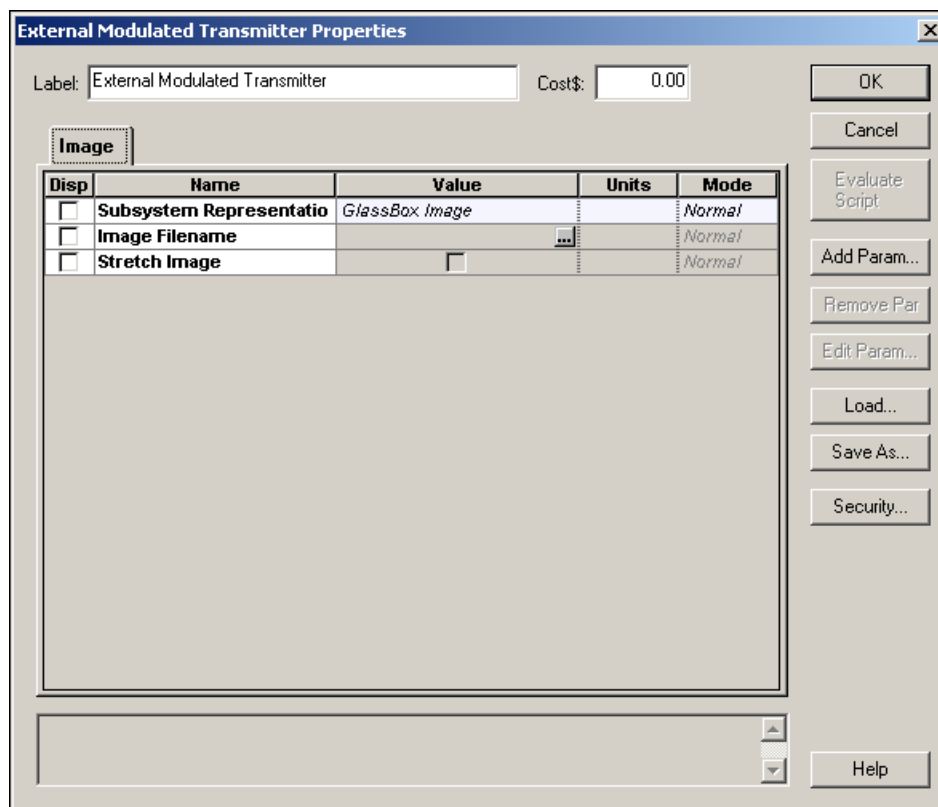
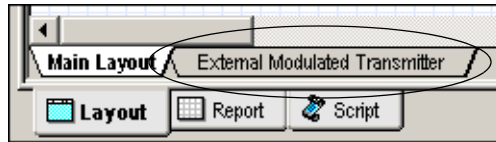


Figure 8 Subsystem modified tab name



Adding a subsystem icon

You can load a user-defined image from a file to use as the a subsystem icon.

To add a subsystem icon, perform the following procedure.

Step Action


- 1 In the **Main layout**, double-click the **External Modulated Transmitter** subsystem glass box.
*The **External Modulated Transmitter Properties** dialog box appears.*
- 2 Click the **Value** cell beside **Subsystem Representation**.
A drop-down list appears.
- 3 From the drop-down list, select **User Image**.
- 4 Click the **Value** cell beside **Image Filename**, and click .
*The **Open** dialog box appears (see [Figure 9](#)).*
- 5 Select the image that you want to use as the subsystem icon, and click **Open**.
Note: You can use the image **External1.bmp** in the samples directory.
*The image you selected appears as the **External Modulated Transmitter** subsystem icon (see [Figure 10](#)).*
- 6 To return to the **Main layout**, click **OK** in the **External Modulated Transmitter** properties dialog box.

Figure 9 Subsystem dialog - User icon import

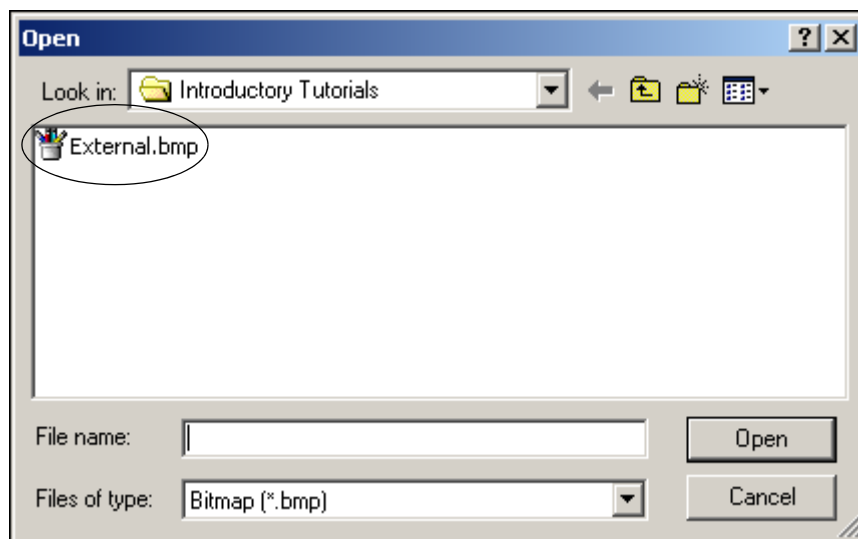
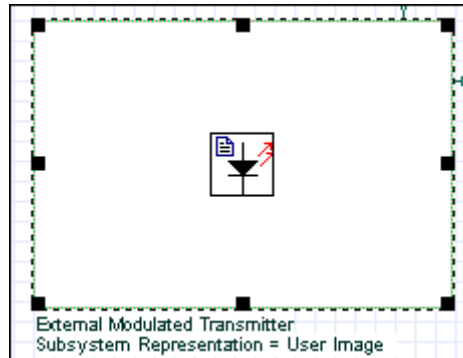


Figure 10 User imported subsystem icon in Project layout



Adding global parameters to a subsystem

In this example, use the **External Modulated Transmitter** subsystem that you created. To access any parameter inside of the **External Modulated Transmitter**, look inside of the subsystem.

To change the subsystem component parameter values without looking inside each time you need to access them, you can create global parameters (see [Figure 11](#)).

In the following procedures, you will export the **Power** and **Frequency** parameters so that you will be able to access them without looking inside the **External Modulated Transmitter** subsystem.

To add a global parameter to a subsystem, perform the following procedure.

Step Action

- 1 In the **Main layout**, double-click **External Modulated Transmitter** subsystem.
*The **External Modulated Transmitter Properties** dialog box appears.*
- 2 Click **Add Param**.
*The **Add Parameter** dialog box appears (see [Figure 11](#)).*
- 3 Type the following values:
 - **Name:** Frequency
 - **Type:** floating-point
 - **Category:** Main
 - **Minimum value:** 100
 - **Maximum value:** 200
 - **Current value:** 193.1
 - **Units:** THz
- 4 Click **Add**.
*The **Frequency** parameter is added to the **Main** category. A message appears in the message window at the bottom of the dialog box to advise you that the **Frequency** parameter has been added.*



- 5 Click **Add Param.**
*The **Add Parameter** dialog box appears.*
- 6 In the **Add Parameter** dialog box, type the following values:
 - **Name:** Power
 - **Type:** floating-point
 - **Category:** Main
 - **Minimum value:** -100
 - **Maximum value:** 30
 - **Current value:** 0
 - **Units:** dBm
- 7 Click **Add.**
*The **Power** parameter is added to the **Main** category. A message appears in the message window at the bottom of the dialog box to advise you that the **Power** parameter has been added.*

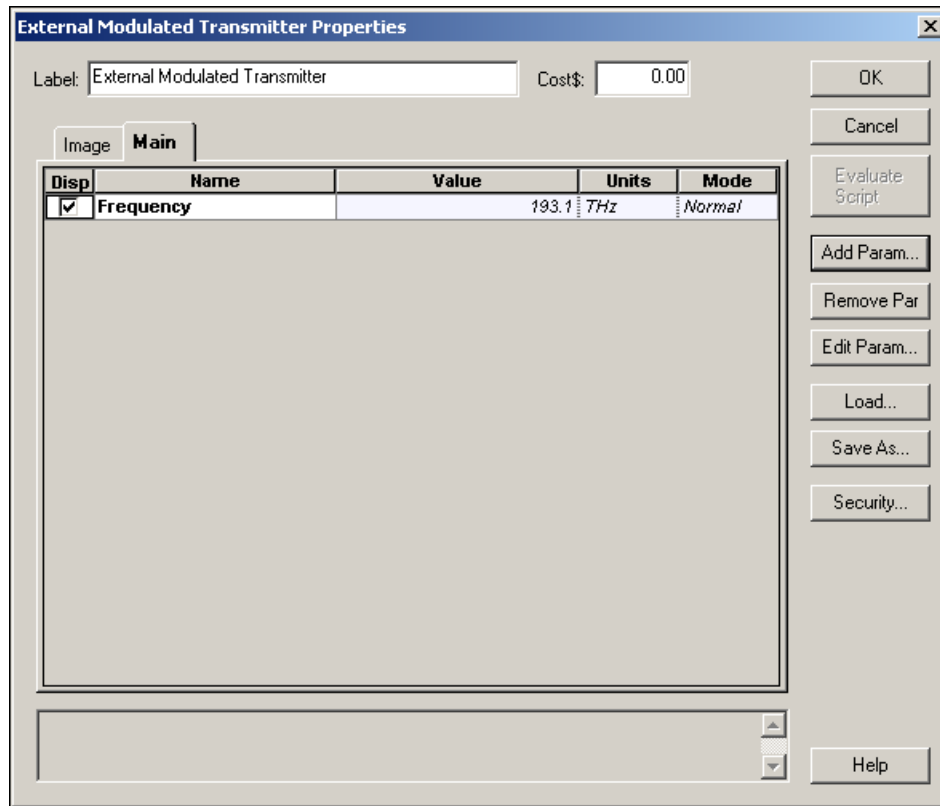
Figure 11 Add Parameter dialog

The screenshot shows the 'Add Parameter' dialog box with the following fields and values:

Field	Value
Name	Frequency
Type	floating-point
Category	Main
Minimum value	100
Maximum value	200
Current value	193.1
Units	THz
Choice list	

Buttons: Add, Cancel, Help

Figure 12 Frequency parameter added to External Modulated Transmitter



Accessing global parameters

In the last section, you added two parameters to the subsystem. Now you must create a reference between the components inside the subsystem and the global parameters (see Figure 13).

To access global parameters, perform the following procedure.

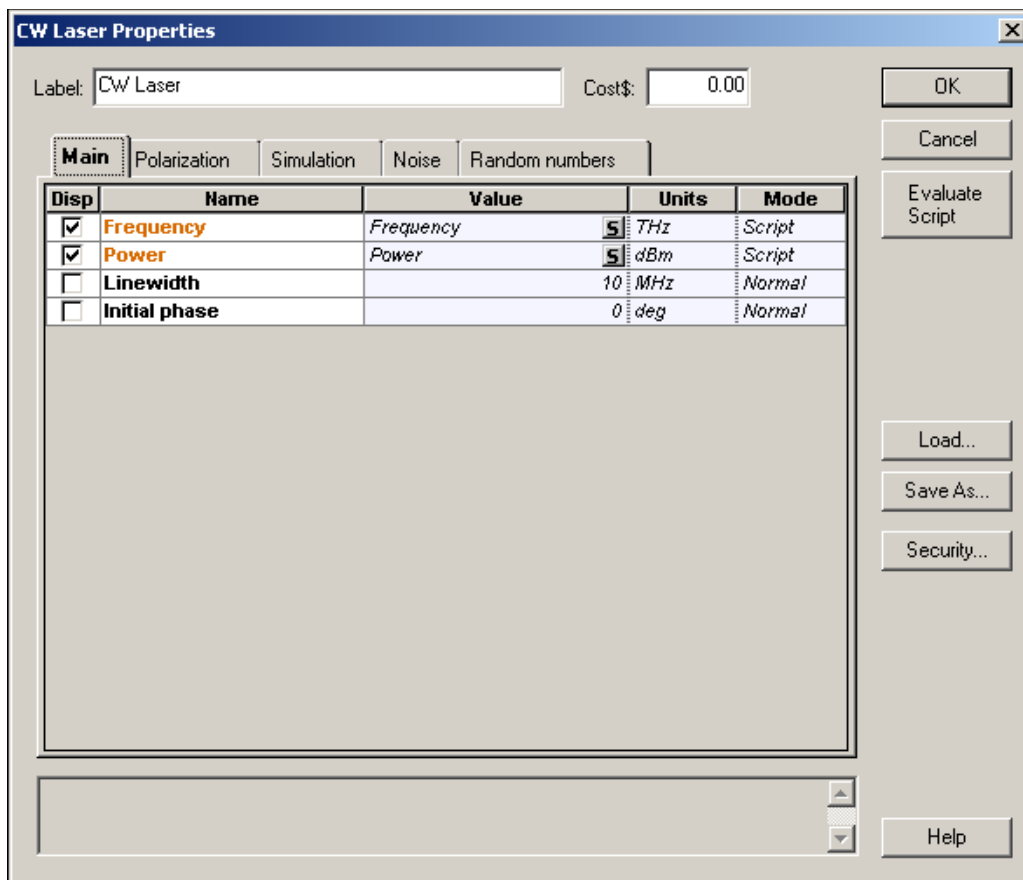
Step Action

- 1 Right-click on the **External Modulated Transmitter** subsystem.
A context menu appears.
- 2 Select **Look inside**.
The subsystem layout appears.
- 3 Double-click the **CW Laser**.
The CW Laser Properties dialog box appears.
- 4 Click in the **Mode** cell beside **Frequency** and select **Script** from the drop-down list.
- 5 Click in the **Units** cell beside **Frequency** and select **THz** from the drop-down list.
- 6 Click the button in the **Value** cell beside **Frequency**.
The Parameter Script Editor dialog box appears.



- 7 Type **Frequency** and click **OK**.
- 8 Click in the **Mode** cell beside **Power** and select **Script** from the drop-down list.
- 9 Click in the **Units** cell beside **Power** and select **dBm** from the drop-down list.
- 10 Click the button in the **Value** cell beside **Power**.
*The **Parameter Script Editor** dialog box appears.*
- 11 Type **Power** and click **OK**.
- 12 To return to the subsystem layout, click **OK**.
- 13 To return to the **Main layout**, click the **Main Layout** tab.

Figure 13 Using script parameters



For example, now when you change the value of the **Frequency** parameter for the **External Modulated Transmitter** subsystem, you will change the values for the **CW Laser** inside the **External Modulated Transmitter** subsystem.



Component Library

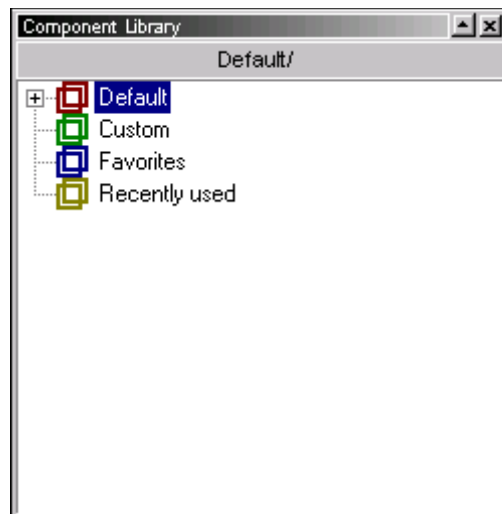
The OptiSystem Component Library consists of four sub-libraries (see [Figure 14](#)).

- **Default:** Read-only — you cannot add or change the parameters of a default component.
- **Custom:** Allows you to add new components and expand the Component Library.

Note: You can base new components on components from the **Default** library. However, the component name must be different from the name of the component in the **Default** library. You can also include subsystems in the **Custom** library.

- **Favorites:** Allows you to add components to a library that includes the ones you use most frequently.
- **Recently used:** Components that you have used in recent projects are added automatically to the **Recently used** library.

Figure 14 Component Library



Before you add the **External Modulated Transmitter** to the **Custom** library, you will create a new folder named **Transmitters** (see [Figure 15](#)).



Creating sub folders in the Custom Library

To create a sub folder in the **Custom** library, perform the following procedure.

- | Step | Action |
|------|--|
| 1 | From the Component Library, select Custom .
<i>The Custom library opens.</i> |
| 2 | Right-click in the background of the Custom library.
<i>A context menu appears (see Figure 15).</i> |
| 3 | Select Add Folder .
<i>The New Folder dialog box appears (see Figure 16).</i> |
| 4 | Type Transmitters and click OK .
<i>A new folder named Transmitters appears in the Custom library in the Component Library (see Figure 17).</i> |

Figure 15 Creating a custom folder menu

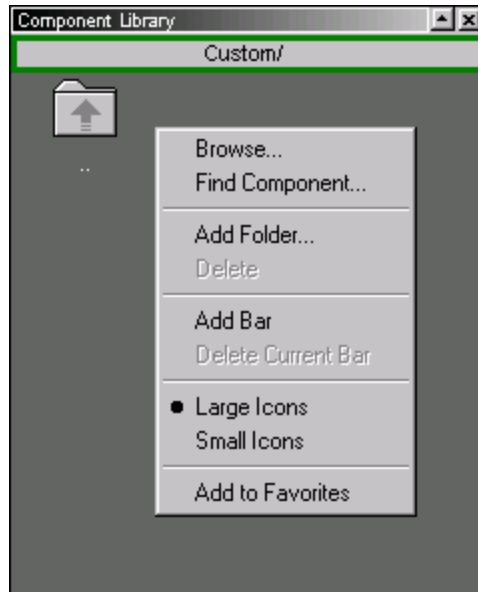
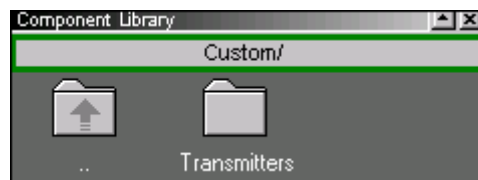


Figure 16 Add Folder dialog



Figure 17 Added folder in Custom library



Adding a component to the library

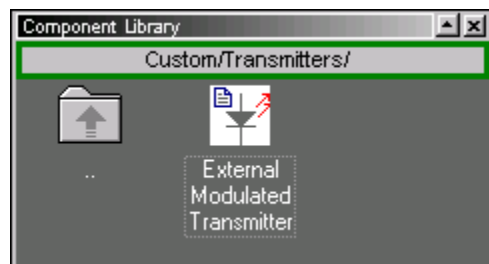
You can drag components to any folder in the **Custom** library. To add the **External Modulated Transmitter** to the **Custom** library in the **Transmitters** folder (see [Figure 18](#)), perform the following procedure.

Step	Action
-------------	---------------

- | | |
|----------|---|
| 1 | Double-click the Transmitters folder in the Custom library. |
| 2 | In the Main layout , select the External Modulated Transmitter component. |
| 3 | Drag the External Modulated Transmitter subsystem into the Transmitters folder. |

*The **External Modulated Transmitter** subsystem appears in the **Transmitters** folder.*

Figure 18 Added component in Custom Library



Creating additional ports

To access the electrical signal from the **NRZ Pulse Generator** inside the **External Modulated Transmitter** subsystem, create an additional output port in the subsystem (see [Figure 19](#)).

To create a port, perform the following procedure.

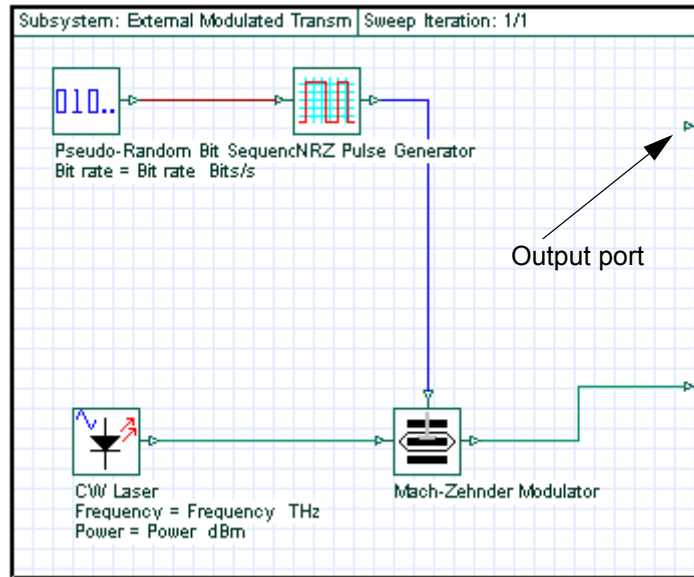
Step	Action
-------------	---------------

- | | |
|----------|--|
| 1 | In the Main layout , right-click on the External Modulated Transmitter .
<i>A context menu appears.</i> |
| 2 | Select Look Inside .
<i>The subsystem layout appears.</i> |



- 3 Create a new output port in the subsystem (see “Creating a subsystem output port” on page 40).

Figure 19 Adding a new output port to the subsystem



Adding a fork to the subsystem

You cannot connect the **NRZ Pulse Generator** output to this port because it's already connected to the **Mach-Zehnder Modulator** input. To solve this problem, you can duplicate the signal from the **NRZ Pulse Generator** and connect it to both the **Mach-Zehnder Modulator** input and subsystem output ports by adding a fork (see [Figure 20](#)).

To add a fork to the subsystem, perform the following procedure.

Note: To increase the size of the layout, press **Ctrl+Shift** and drag the cursor from the top left corner of the **Main layout** to the bottom right corner.

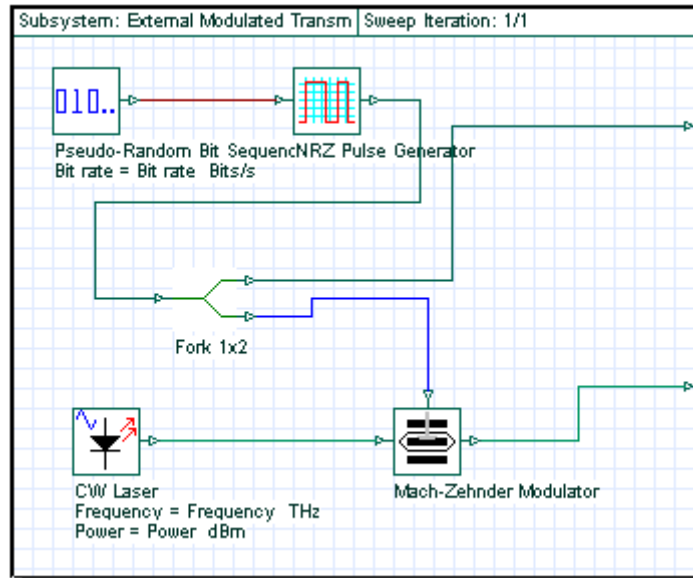
- | Step | Action |
|------|---|
| 1 | From the Component Library, select Default > Tools Library . |
| 2 | Drag the Fork 1x2 to the Main layout . |
| 3 | Select the blue connection between the NRZ Pulse Generator and the Mach-Zehnder Modulator .
<i>The selected connection changes to red.</i> |
| 4 | Press Delete to delete the connection. |
| 5 | Connect the NRZ Pulse Generator output to the Fork input. |



- 6 Connect the first **Fork** output to the subsystem output port.
- 7 Connect the second **Fork** output to the available **Mach-Zehnder Modulator** input port (see Figure 20).

Note: To delete the internal data monitors, from the **Layout Tools** toolbar select the **Monitor tool** and click the monitor in each component output port.

Figure 20 External Modulated Transmitter with two output ports



- 8 To return to the **Main layout**, click the **Main Layout** tab.

Saving the modified subsystem

To save the modified subsystem, perform the following procedure.

- | Step | Action |
|-------------|---|
| 1 | In the Transmitters folder in the Custom library, select and right-click the External Modulated Transmitter .
<i>A context menu appears.</i> |
| 2 | Select Delete . |
| 3 | Drag the modified External Modulated Transmitter from the Main layout into the Transmitters folder in the Custom library. |

Running the External Modulated Transmitter simulation

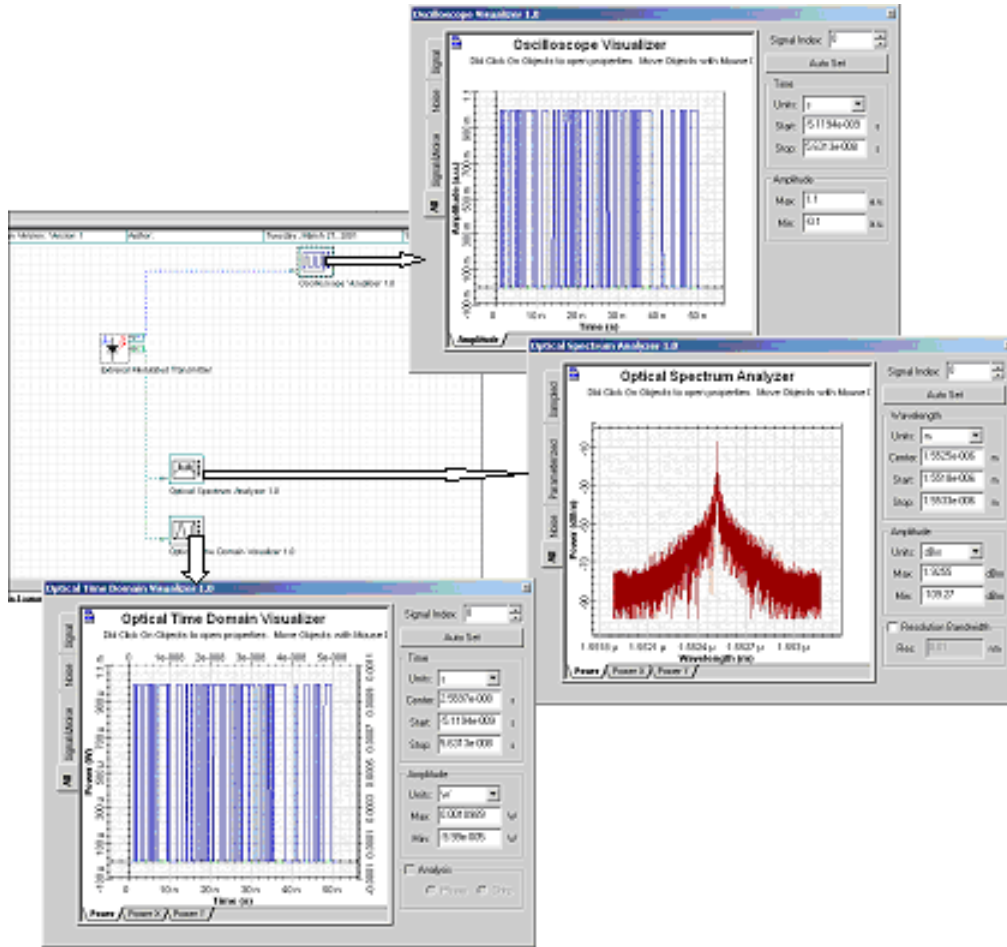
To run the simulation for the External Modulated Transmitter, perform the following procedure.

Note: Before you run the simulation, you must connect the External Modulated Transmitter output port to a component or visualizer.

- | Step | Action |
|-------------|--|
| 1 | Connect the first External Modulated Transmitter output to the Oscilloscope Visualizer . |
| 2 | Connect the second External Modulated Transmitter output port to the Optical Spectrum Analyzer and to the Optical Time Domain Visualizer . |
| 3 | Run the simulation: <ol style="list-style-type: none"> a. Click Calculate. b. Click the PLAY button.
<i>The results appear in the Calculation Output window.</i> |
| 4 | To view the graphs of the results, double-click each visualizer (see Figure 21).
Note: The center frequency and power of the transmitter are defined by the values of the Frequency and Power parameters. |



Figure 21 Results from External Modulated Transmitter



Notes:



Lesson 3: Optical Systems — WDM Design

This lesson describes how to simulate a WDM system with 8 channels. You will become familiar with the Component Library, parameter groups, and visualizers such as the **BER Analyzer**.

Global parameters

For this simulation we will use default parameters for the **Bit rate**, **Bit sequence length**, and **Sample rate**.

Viewing the global parameters

To view the default global parameters, perform the following procedure.

- | Step | Action |
|------|---|
| 1 | From the File menu, select New .
<i>A blank Main layout appears in the Project layout window.</i> |
| 2 | Double-click anywhere in the Main layout .
<i>The Layout 1 Parameters dialog box appears (see Figure 1).</i> |

The default global parameters used for this simulation are:

- **Bit rate:** 2500000000 Bits/s
- **Sequence length:** 128 bits



- Time window: 5.12e-008

Figure 1 Global parameters

The screenshot shows a software dialog box titled "Layout 1 Parameters". At the top, there is a "Label:" field containing "Layout 1". Below this are four tabs: "Simulation" (selected), "Signals", "Noise", and "Signal tracing". The main area contains a table with the following data:

Name	Value	Units	Mode
Simulation window	Set bit rate		Normal
Reference bit rate	<input checked="" type="checkbox"/>		Normal
Bit rate	2500000000	Bits/s	Normal
Time window	5.12e-008	s	Normal
Sample rate	160000000000	Hz	Normal
Sequence length	128	Bits	Normal
Samples per bit	64		Normal
Number of samples	8192		Normal
Iterations	1		Normal

On the right side of the dialog, there are buttons for "OK", "Cancel", "Add Param...", "Remove Par", "Edit Param...", and "Help". At the bottom, there are two small arrow buttons (up and down) and a "Help" button.



Transmitters

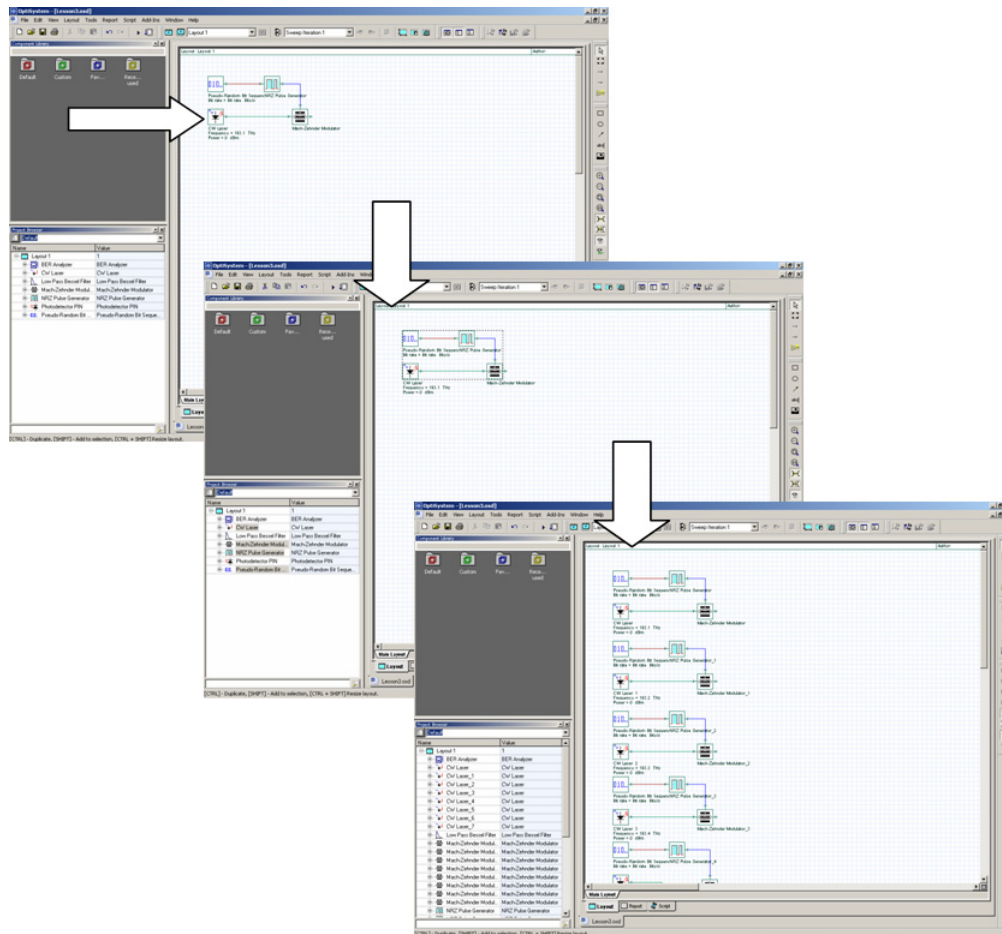
To create the transmitters for the **WDM** system with 8 channels, perform the following procedure.

Step Action

Note: To increase the size of the layout, press **Ctrl+Shift** and drag the cursor from the top left corner of the **Main layout** to the bottom right corner.

- 1 Create an external modulated laser using the procedures in [Lesson 1: Transmitter — External modulated laser](#).
- 2 Select the four external modulated laser components.
- 3 Copy and paste the selected components to create a total of eight transmitters (see [Figure 2](#)).

Figure 2 Creating eight external modulated lasers



LESSON 3: OPTICAL SYSTEMS — WDM DESIGN

- 4 From the Component Library, select **Default > WDM Multiplexers Library > Multiplexers**.
 - 5 Drag the **WDM Mux 8x1** to the **Main layout**.
 - 6 Connect the **Mach-Zehnder Modulator** outputs to the **WDM Mux 8x1** inputs.
- Note:** You can select the values for the Mux internal filters to be displayed in the layout by selecting the **Disp** option in the **Channels** tab of the **Mux properties** dialog box (see [Figure 3](#)).

Figure 3 Displaying multiplexer frequency channels

The screenshot shows the OptiSystem software interface. The main layout displays a circuit with four Mach-Zehnder Modulators connected to a WDM Mux 8x1 component. The 'WDM Mux 8x1 Properties' dialog box is open, showing the 'Channels' tab. The table below lists the channels and their properties.

Disp	Channels	Name	Value	Units	Mode
<input type="checkbox"/>	<input type="checkbox"/>	Frequency[0]	193.1 THz	Normal	
<input type="checkbox"/>	<input type="checkbox"/>	Frequency[1]	193.2 THz	Normal	
<input type="checkbox"/>	<input type="checkbox"/>	Frequency[2]	193.3 THz	Normal	
<input type="checkbox"/>	<input type="checkbox"/>	Frequency[3]	193.4 THz	Normal	
<input type="checkbox"/>	<input type="checkbox"/>	Frequency[4]	193.5 THz	Normal	
<input type="checkbox"/>	<input type="checkbox"/>	Frequency[5]	193.6 THz	Normal	
<input type="checkbox"/>	<input type="checkbox"/>	Frequency[6]	193.7 THz	Normal	
<input type="checkbox"/>	<input type="checkbox"/>	Frequency[7]	193.8 THz	Normal	

The 'Disp' column contains checkboxes for each channel. In the second instance of the dialog box shown in the screenshot, these checkboxes are checked, and the corresponding channels are displayed in the layout. The 'Channels' tab also includes 'Ripple', 'Simulation', and 'Noise' sub-tabs.



Parameter groups

To enter the frequency values for each channel, double-click each **CW Laser** and enter the frequency value. To simplify the process of entering parameter values for each component, use the **Parameter groups** feature.

Accessing the Parameter groups table

The **Parameter groups** table gives you access to relevant parameters for the **WDM** simulation (such as **Frequency** and **Power**) in one place. You can go into each component and enter the values to change the parameters.

To access a parameter group, perform the following procedure.

Step	Action
------	--------

- | | |
|---|---|
| 1 | From the Main toolbar, select Layout > Parameter groups .
<i>The Parameter Groups dialog box appears (see Figure 4).</i> |
|---|---|

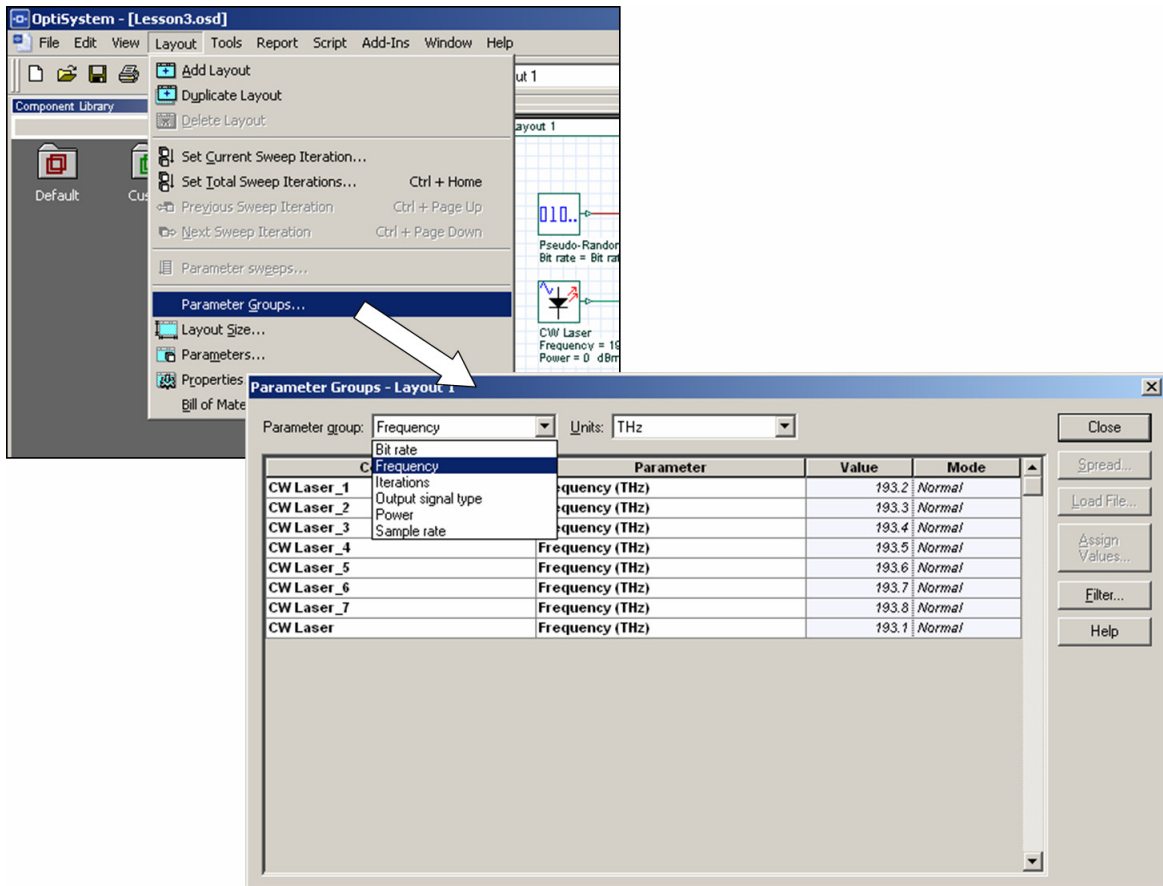
Note: Typically you can access the following parameters from the **Parameter Groups** table (see [Figure 4](#)):

- Bit rate
- Convert noise bins
- Frequency
- Iterations
- Output signal type
- Power
- Sample rate

You can also enter the units for the parameter group and edit the parameter values by typing new values or using the tools from the context menu.



Figure 4 Parameter groups



- 2 Click the **Value** column.
*The **Value** column is highlighted.*
- 3 In the **Parameters Groups** table, select the **Value** column and right-click.
A context menu appears.
- 4 Select **Spread**.
*The **Parameter Group Spread** dialog box appears (see [Figure 5](#)).*
- 5 In the **Start Value** text box, type:
193.1
- 6 In the **Increment** text box, type:
0.1
- 7 To return to the **Main layout**, click **Close**.
*The frequencies in the **Value** column are updated from 193.1 THz to 193.8 THz (see [Figure 5](#)).*



Figure 5 Entering frequency values



Testing the transmitter

To verify the system setting for this design, use an **Optical Spectrum Analyzer** and a **WDM** analyzer to obtain the signal spectrum and the total power for each channel.

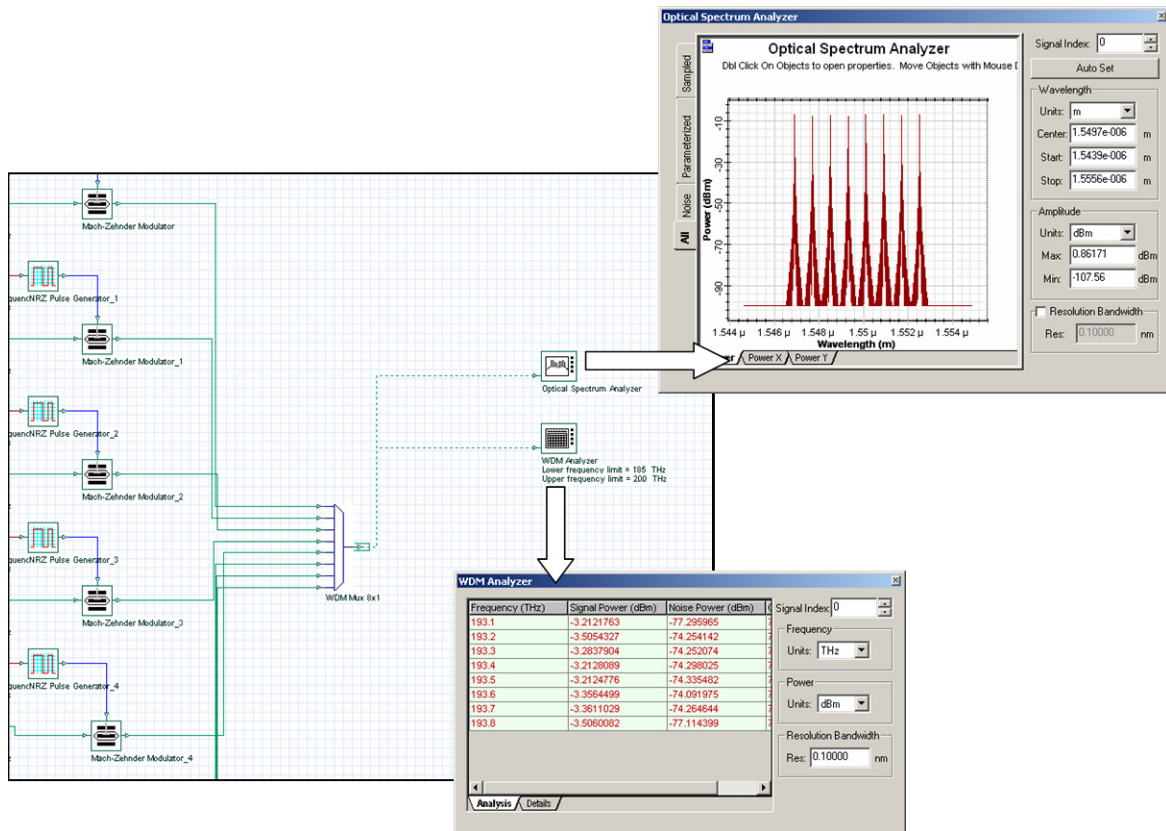
To test the transmitter, perform the following procedure.

Step Action

- 1 From the Component Library, select **Default > Visualizer Library > Optical**.
- 2 Drag the **Optical Spectrum Analyzer** to the **Main layout**.
- 3 Drag the **WDM Analyzer** to the **Main layout**.
- 4 Connect the **Optical Spectrum Analyzer** input and the **WDM Analyzer** input to the **WDM Mux** output.
- 5 Run the simulation:
 - a. Click **Calculate**.
*The **Calculation** dialog box appears.*
 - b. Click the **Run** button.
*The calculation progress appears in the **Calculation** dialog box.*
- 6 Double-click **Optical Spectrum Analyzer** to view the graph (see Figure 6).
The graph appears and displays eight equally-spaced channels.
- 7 Double-click the **WDM Analyzer** to view the numerical results (see Figure 6).
*The **WDM Analyzer** dialog box appears and displays the numerical results for the eight channels.*

Note: The average signal power for each channel is approximately -3.3 dBm for a resolution bandwidth of 0.1 nm.

Figure 6 Simulation results from the visualizers



Fiber + EDFA spans

To create a fiber connected to an EDFA, perform the following procedure.

Step Action

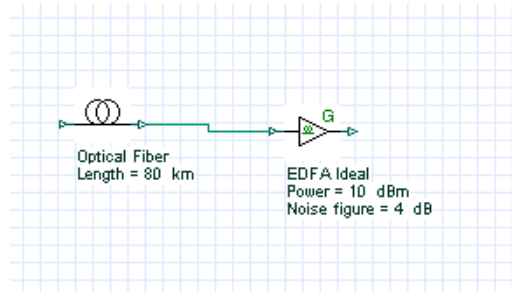
- 1 From the Component Library, select **Default > Optical Fibers Library**.
- 2 Drag the **Optical Fiber** to the **Main layout**.
- 3 Double-click the **Optical Fiber**.
*The **Optical Fiber Properties** dialog box appears.*
- 4 To change the parameter length to 80 km, click the **Value** cell beside **Length**, and type:

80



- 5 To return to the **Main layout**, click **OK**.
- 6 From the Component Library, select **Default > Amplifiers Library > Optical**.
- 7 Drag the **Optical Amplifier** to the **Main layout**.
- 8 In the EDFA Parameters, change **Operation Mode** to **Power Control**.
- 9 Connect the **Optical Fiber** output to the **EDFA Ideal** input (see [Figure 7](#)).

Figure 7 Fiber + EDFA span



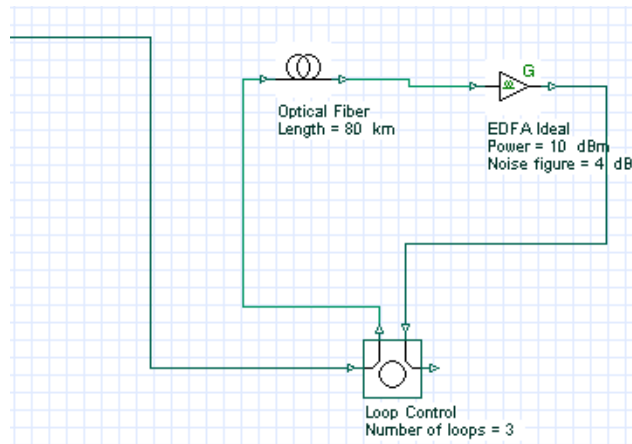
Connecting the Loop Control

The **Loop Control** allows you to set the number of times the signal propagates in the components that are connected between the **Loop Control** input and output ports. Use the **Loop Control** to calculate the system performance based on the number of fibers and EDFA spans.

To connect the **Loop Control**, perform the following procedure.

- | Step | Action |
|-------------|---|
| 1 | From the Component Library, select Default > Tools Library . |
| 2 | Drag the Loop Control to the Main layout . |
| 3 | Connect the output port of the WDM Mux 8x1 to the first Loop Control input port. |
| 4 | Connect the first Loop Control output port to the Optical Fiber input port (see Figure 8). |
| 5 | Connect the output port of the second EDFA Ideal (connected to the Optical Fiber) to the Loop Control input port (see Figure 8). |

Figure 8 Loop control



Setting the number of signal round trips

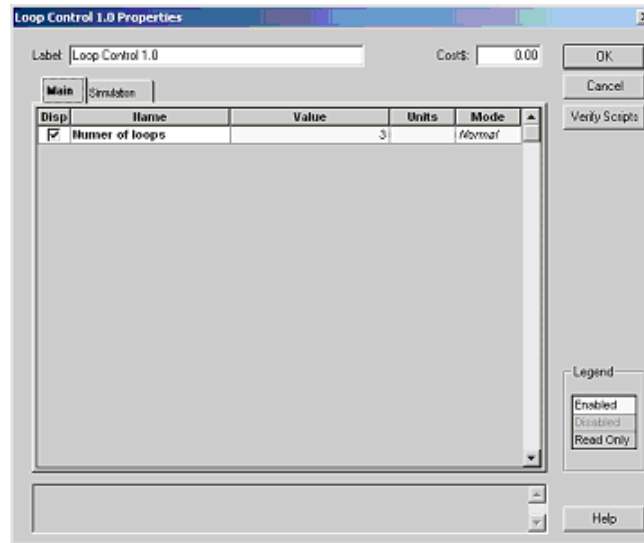
You define the number of round trips that the signal makes through the loop by setting the Number of loops parameter in the Loop Control.

To set the number of signal round trips, perform the following procedure.

- | Step | Action |
|-------------|--|
| 1 | Double-click the Loop Control .
<i>The Loop Control Properties dialog box appears (see Figure 9).</i> |



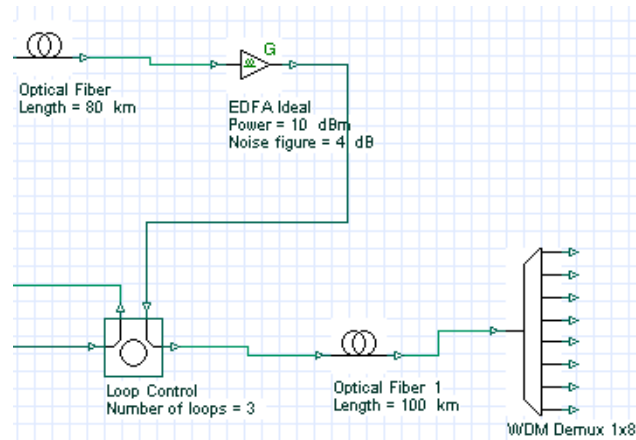
Figure 9 Loop control parameters



- 2 To set the number of signal round trips, click the **Value** cell beside **number of loops**, highlight the 0, and type:
3
Note: The signal will propagate $3 \times 80\text{km} = 240 \text{ km}$.
- 3 To return to the **Main layout**, click **OK**.
- 4 From the Component Library, select **Default > Optical Fibers Library**.
- 5 Drag the **Optical Fiber** to the **Main layout**.
- 6 Set the fiber **Length** to 100km.
- 7 From the Component Library, select **Default > WDM Multiplexers Library > Demultiplexers**.
- 8 Drag the **WDM Demux 1x8** to the **Main layout**.
- 9 Connect the second **Loop Control** output port to the **Optical Fiber** and then to the **WDM Demux 1x8** input port (see [Figure 10](#)).
- 10 To view the WDM Demux 1x8 frequencies:
 - a. Double-click the **WDM Demux 1x8**
*The **WDM Demux 1x8 Properties** dialog box appears.*
 - b. Select the **Channels** tab.
 - c. In the **Disp** column, select the check box beside each frequency.
 - d. To return to the **Main layout**, click **OK**.
*The eight frequencies and their values appear below the **WDM Demux 1x8** (see [Figure 10](#)).*



Figure 10 Demultiplexer



Obtaining results after the demultiplexer

To verify the system setting for this design, you will use an **Optical Spectrum Analyzer**, a **WDM Analyzer**, and an **Optical Time Domain Visualizer**.

To obtain the signal results in time and frequency and the total power for each channel after the signal passes through the **WDM Demux 1x8**, perform the following procedure.

- | Step | Action |
|------|---|
| 1 | From the Component Library, select Default > Visualizer Library > Optical . |
| 2 | Drag the Optical Time Domain Visualizer to the Main layout . |
| 3 | Drag the Optical Spectrum Analyzer to the Main layout . |
| 4 | Drag the WDM Analyzer to the Main layout . |
| 5 | Connect each of the visualizers to the first output port on the WDM Demux 1x8 (193.1 THz). |
| 6 | Run the simulation: <ol style="list-style-type: none"> Click Calculate. Click the Run button. |

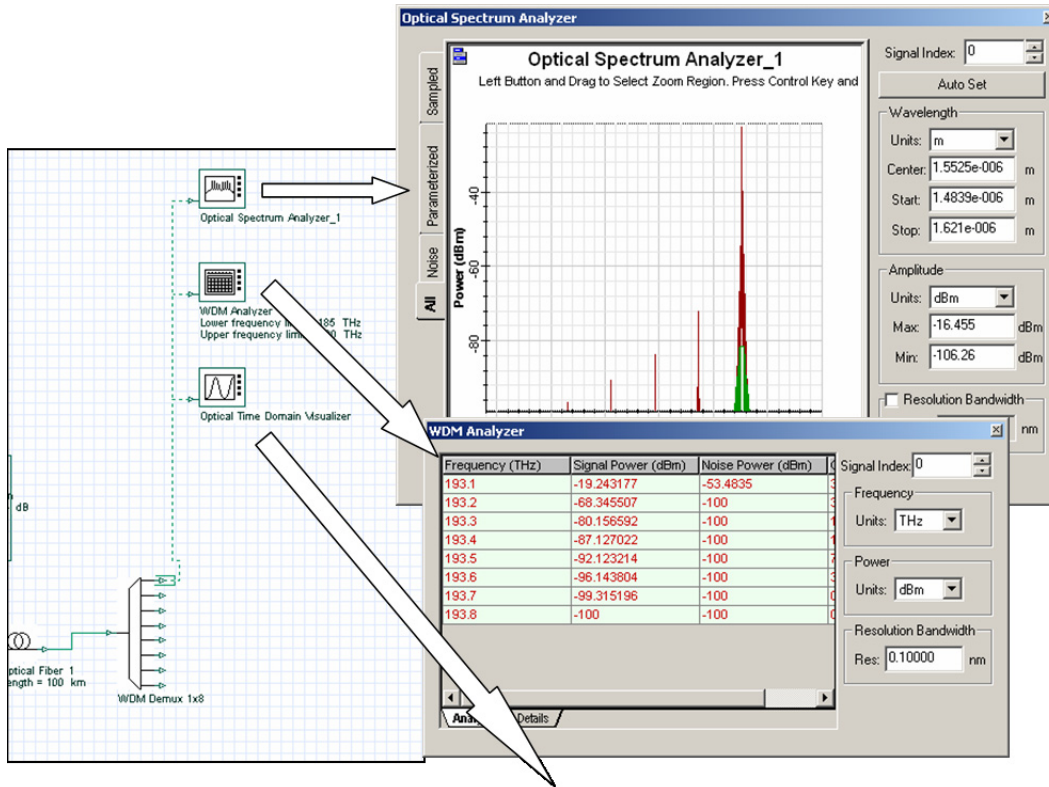
*The calculation progress appears in the **Calculation** dialog box.*

Note: This simulation will take some time.



- 7 Double-click the visualizers to view the results and graphs (see Figure 11).

Figure 11 Simulation results from the visualizers

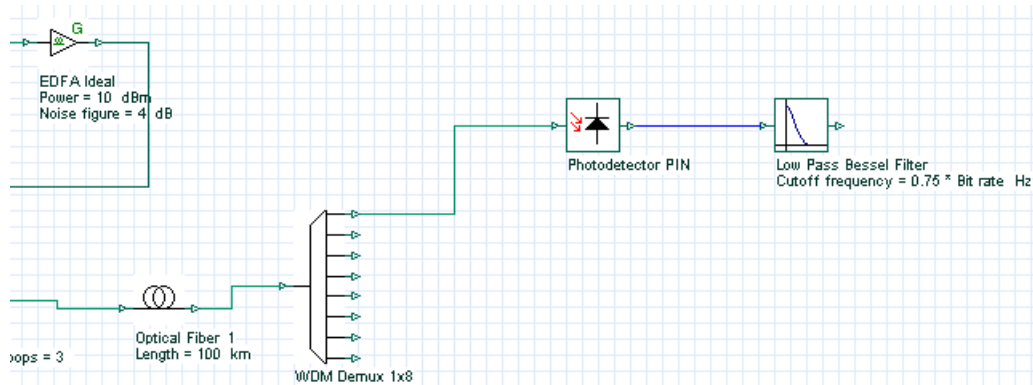


Adding a receiver

To add a photodetector, an electrical amplifier, and a Bessel filter to the design, perform the following procedure.

- | Step | Action |
|------|--|
| 1 | From the Component Library, select Default > Receivers Library > Photodetectors . |
| 2 | Drag the Photodetector PIN to the Main layout . |
| 3 | Set the Thermal Noise parameter to 10^{-21} . |
| 4 | From the Component Library, select Default > Filters Library > Electrical . |
| 5 | Drag the Low Pass Bessel Filter to the Main layout . |
| 6 | Connect the first output port on the WDM Demux 1x8 to the Photodetector PIN input port. |
| 7 | Connect the Photodetector PIN output port to the Low Pass Bessel Filter input port (see Figure 12). |

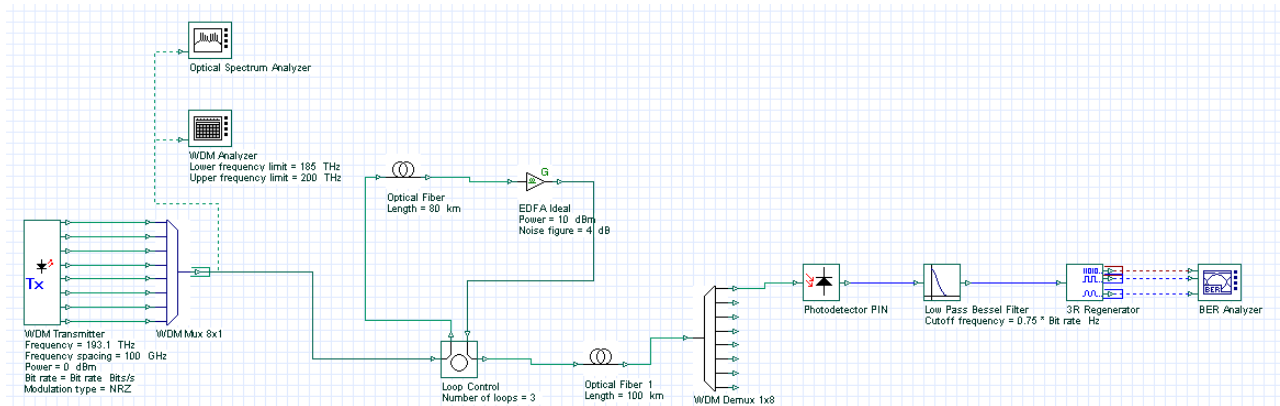
Figure 12 Adding the receiver to the WDM system



Note: This design could be made using the **WDM Transmitter** component from the **Optical Transmitters** library. Fewer steps are required if you use this component. When using the **WDM Transmitter**, the receiver will require the **3R Regenerator** component from the Receivers library (see sample file **Lesson3B.osd**).



Figure 13 WDM Transmitter layout design (Lesson 3B.osd)



BER Analyzer

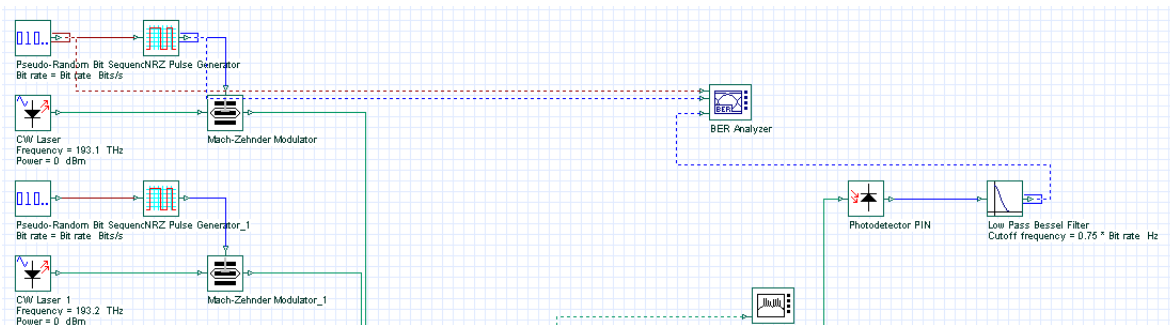
The **BER Analyzer** calculates system performance. It can predict the BER, Q-factor, threshold, and Eye aperture of a system. You can view the BER patterns and the BER value in each point of the Eye diagram using 3D graphs.

Adding the BER Analyzer

To add the **BER Analyzer** to the system, perform the following procedure.

- | Step | Action |
|------|---|
| 1 | From the Component Library, select Default > Visualizer Library > Electrical . |
| 2 | Drag the BER Analyzer to the Main layout .
Note: The first BER Analyzer input port receives the binary signal. |
| 3 | Connect the output port of the first Pseudo-Random Bit Sequence Generator to the first BER Analyzer input port.
Note: The second BER Analyzer input port receives the original sampled signal. It compensates the delay between the signals that are transmitted and received. |
| 4 | Connect the output of the first NRZ Pulse Generator to the second BER Analyzer input port. |
| 5 | Connect the Low Pass Bessel Filter output port to third BER Analyzer input port (see Figure 14). |

Figure 14 Connecting the BER Analyzer



- 6 Run the simulation:
- Click **Calculate**.
 - Click the **Run** button.
The calculation progress appears in the Calculation dialog box.
- Note:** This simulation will take some time.



- 7 To display the results and graphs, double-click the **BER Analyzer**.

Viewing the BER Analyzer graphs and results

To view the BER Analyzer graphs and results, perform the following procedure.

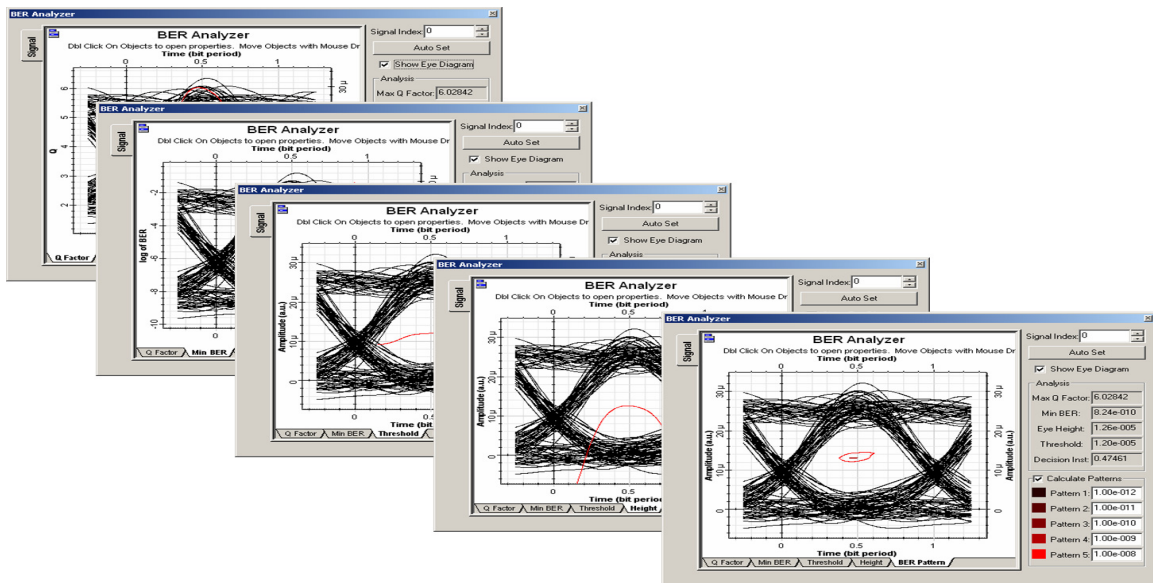
Step Action

- 1 Double-click the **BER Analyzer**.
The BER Analyzer window appears and displays a graph.
- 2 Select **Show Eye Diagram**.
The graph is redrawn and displays the Eye Diagram.

Note: When you open the **BER Analyzer**, the following graphs appear together with the Eye Diagram (see [Figure 15](#)):

- **Q-Factor:** Maximum value for the Q-Factor versus Decision Instant.
- **Min BER:** Minimum value for the BER versus Decision Instant.
- **Threshold:** Threshold value versus Decision Instant that gives the maximum Q-Factor and the Min BER.
- **Height:** Eye height versus Decision Instant.
- **BER Pattern:** When Calculate Patterns is selected, displays the regions where the BER value is less than the user-defined values.

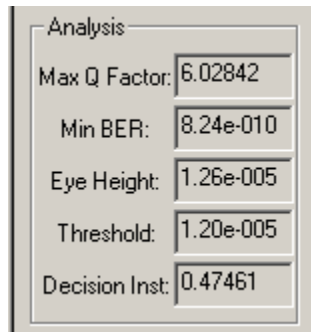
Figure 15 BER Analyzer graphs



The **Analysis** group box displays the following values (see [Figure 16](#)):

- Maximum Q-Factor
- Minimum BER
- Maximum eye aperture
- Threshold
- Decision Instant at the Max Q-Factor/ Min BER

Figure 16 BER Analyzer results



Calculating BER patterns

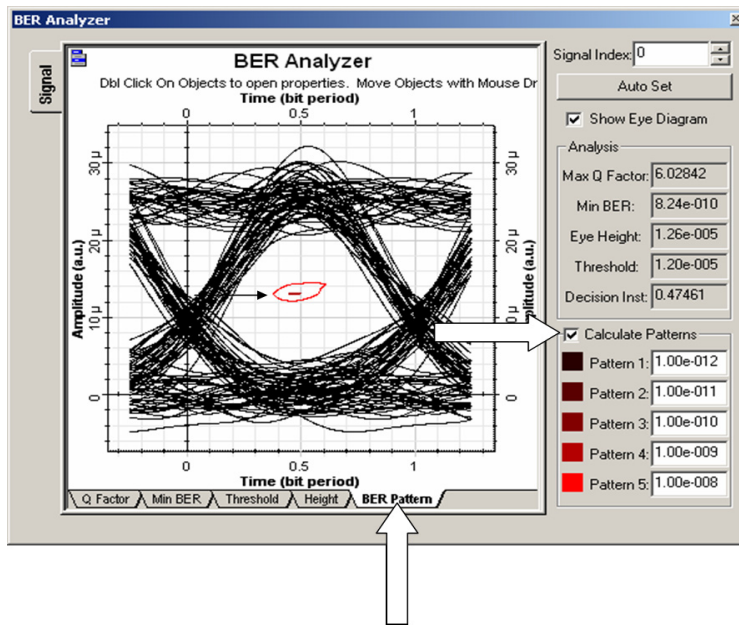
To calculate the BER patterns, perform the following procedure.

Step Action

Note: The **Calculate Patterns** check box is only available when you select the **BER Pattern** tab (see [Figure 17](#)).

- 1 In the **BER Analyzer** window, select the **BER Pattern** tab.
*The graph changes to display the **BER Patterns** graph.*
- 2 Select the **Calculate Patterns** check box.
The visualizer recalculates the results and displays a new graph.

Figure 17 Calculating BER patterns



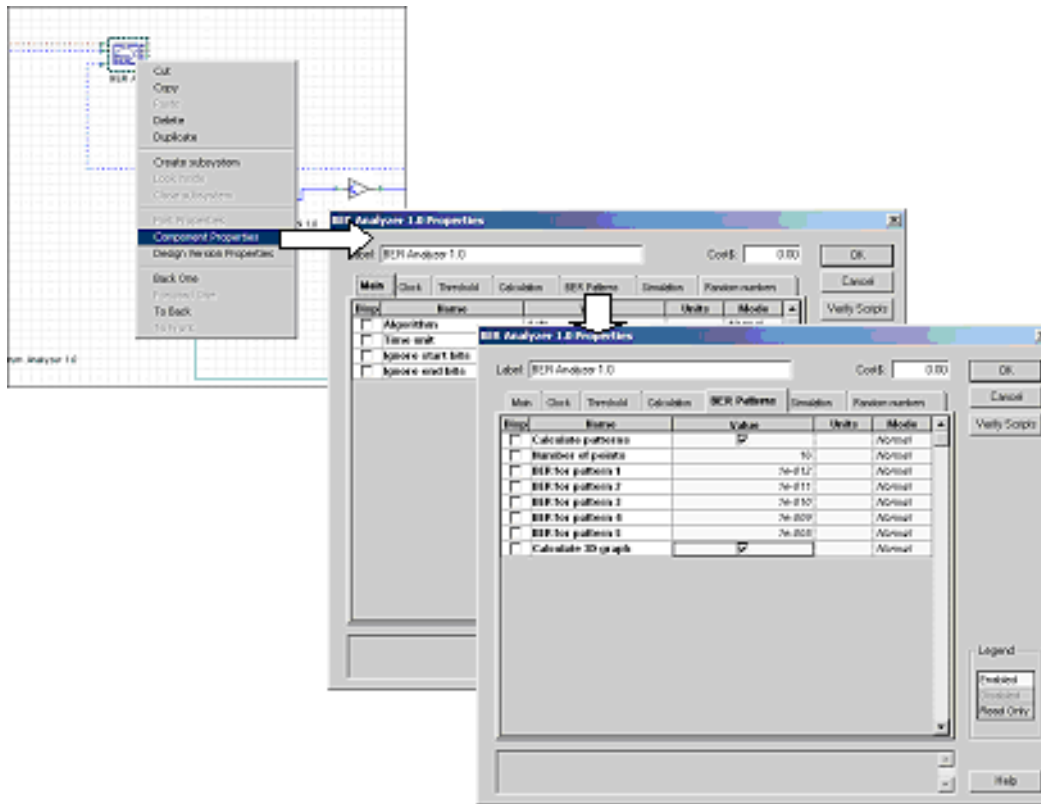
Calculating the 3D BER graph

To calculate the 3D BER, perform the following procedure.

Step	Action
	Note: You must enable the calculation of the BER patterns and the 3D graph.
1	Right-click the BER Analyzer . <i>A context menu appears.</i>
2	Select Component Properties . <i>The BER Analyzer Properties dialog box appears.</i>
3	Select the BER Patterns tab (see Figure 18).
4	Select the Value cell beside Calculate Patterns .
5	Select the Value cell beside Calculate 3D graph .
6	In the Disp column, select Calculate Patterns and Calculate 3D graph .
7	To recalculate the results and return to the Main layout , click OK . <i>The visualizer recalculates the graphs and results.</i>



Figure 18 Calculating 3D graph



Viewing the 3D graph

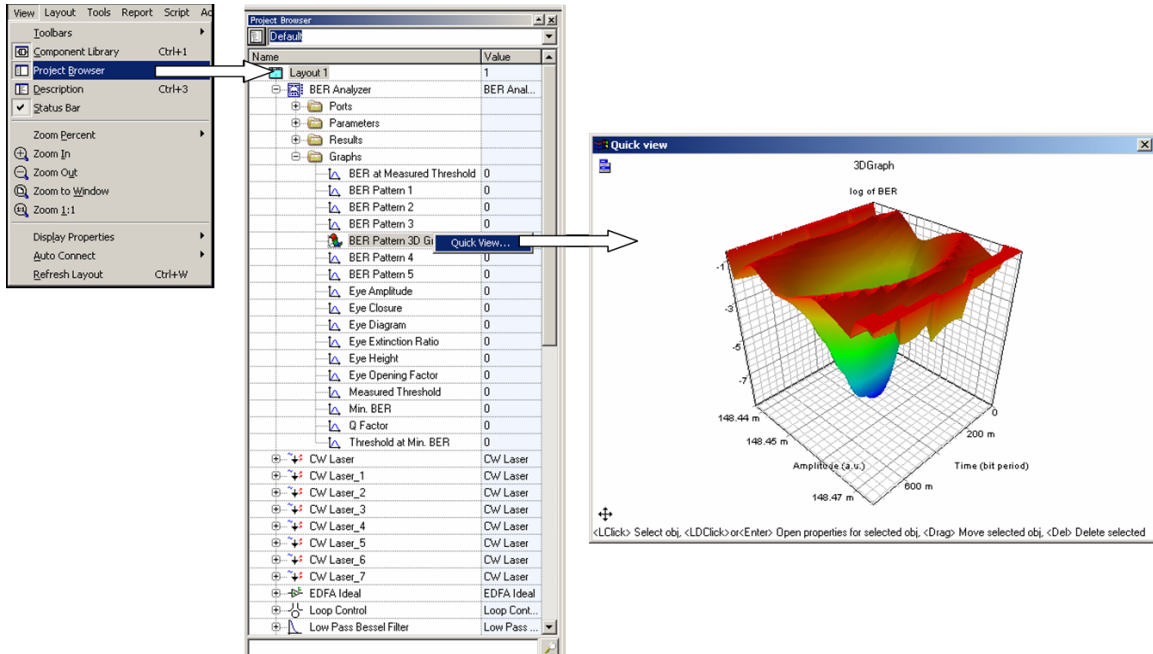
To view the 3D graph, perform the following procedure.

Step Action

- 1 If the **Project Browser** is closed:
 - From the **Main** toolbar, select **View > Project Browser**,
 - or
 - Press **Ctrl+2**.
- 2 In the **Project Browser**, expand the **BER Analyzer** component. *A list of available options appears.*
- 3 Expand the **Graphs** item in the list. *A list of **BER Analyzer** available graphs appears.*
- 4 From the list of **BER Analyzer** graph items, right-click the **BER Pattern 3D Graph**.
- 5 Select **Quick View**. *The 3D graph appears (see [Figure 19](#)).*



Figure 19 3D graphs from the project browser



Lesson 4: Parameter Sweeps — BER x Input power

This lesson describes how to combine the results from the **BER/Eye Analyzer** with the signal input power using parameter sweeps. You will become familiar with parameter sweeps, graph builder, results, graphs, and views.

The first three procedures describe how to iterate the **Power** parameter for the **CW Laser** component.

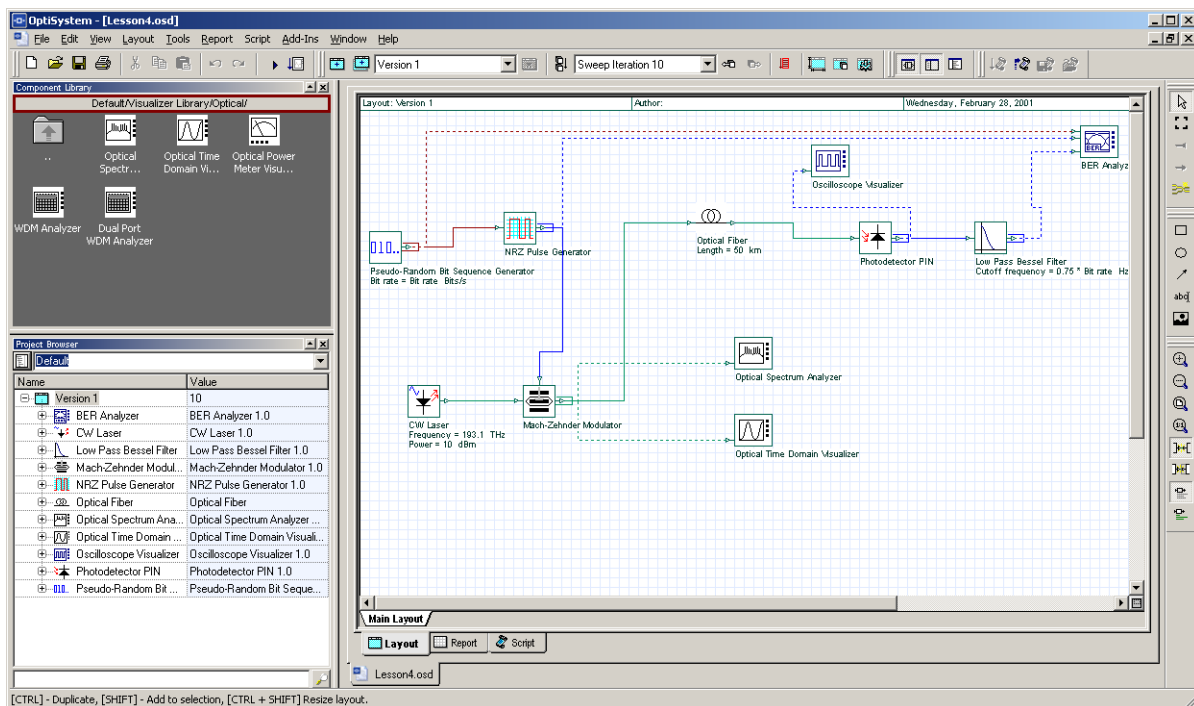
Opening the sample file

Use the sample file **Lesson4 Parameter Sweeps - BER x Input power.osd**. To open the file, perform the following procedure.

Step Action

- 1 From the **Main** toolbar, select **File > Open**.
*The **Open** dialog box appears.*
- 2 From your C: drive, select **Program Files > Optiwave Software > OptiSystem 9 > samples > Introductory tutorials > "Lesson4 Parameter Sweeps - BER x Input power.osd"**.
The sample file appears (see [Figure 1](#)).

Figure 1 Sample file that calculates BER x Input power



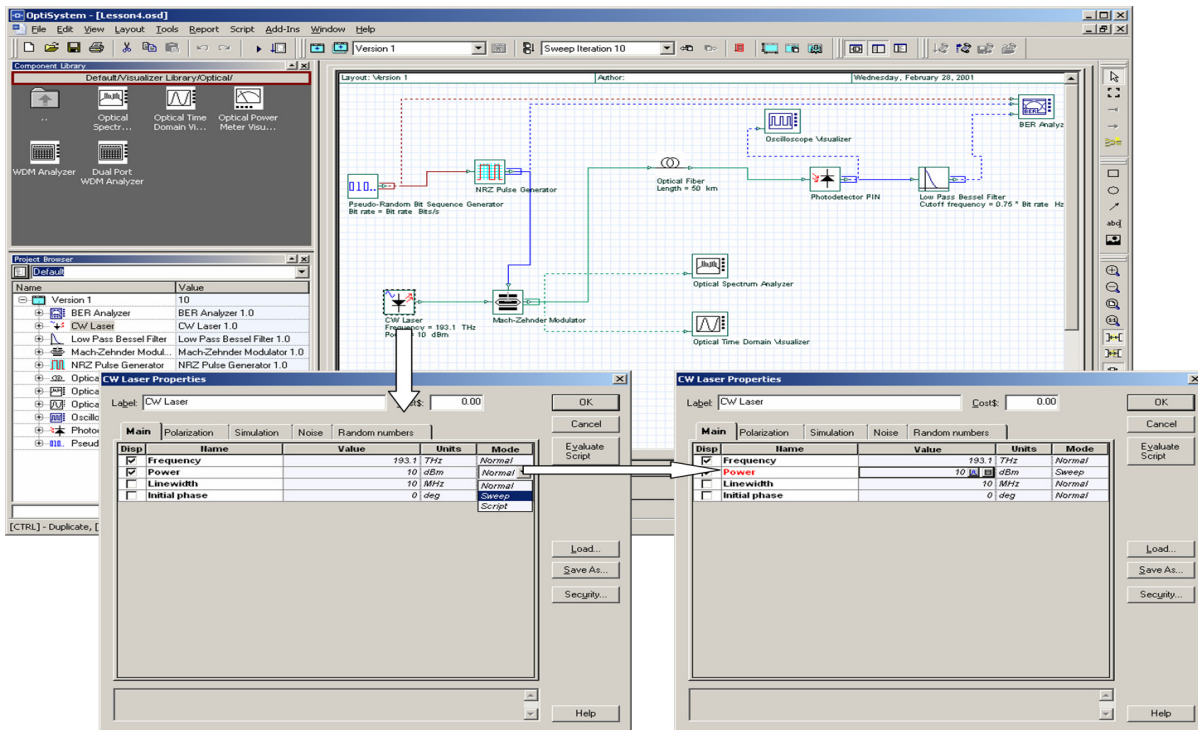
Selecting the parameter to iterate using the Sweep mode

To select the parameter to iterate using the Sweep mode, perform the following procedure.

Step Action

- 1 Double-click the **CW Laser**.
The CW Laser Properties dialog box appears (see Figure 2).
- 2 In the **Power** row, click the **Mode** cell.
A drop-down list appears.
- 3 Select **Sweep**.

Figure 2 Changing the parameter mode to Sweep



- 4 To save the settings, click **OK**.
The CW Laser Properties dialog box closes.



Changing the number of sweep iterations

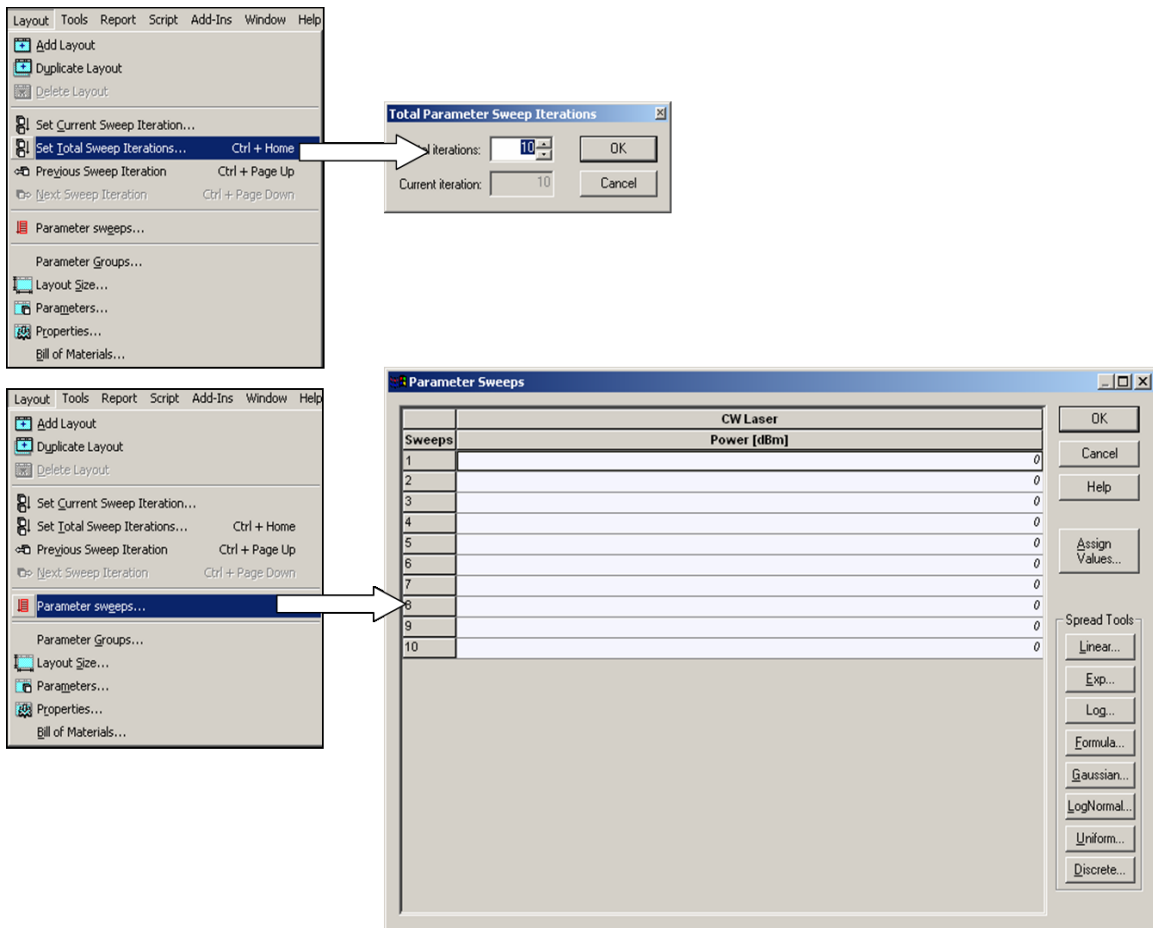
We want to be able to calculate the design using different values of the input power.

To change the number of sweep iterations, perform the following procedure.

Step Action

- 1 From the **Layout** menu, select **Set Total Sweep Iterations**.
*The **Total Parameter Sweep Iterations** dialog box appears (see [Figure 3](#)).*
- 2 In the **Total Iterations**, type:
10
- 3 To set the value and return to the **Main layout**, click **OK**.

Figure 3 Adding iterations

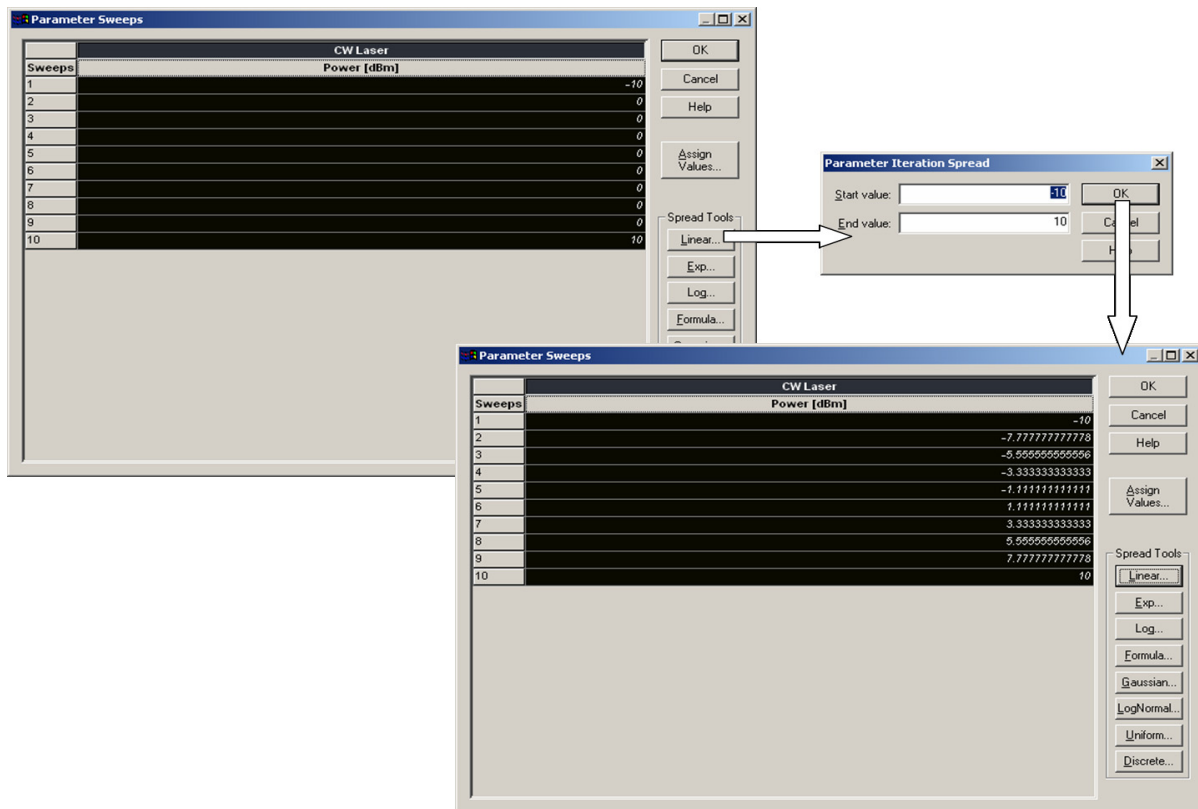


Changing the values of sweep iterations

To change the values of sweep iterations, perform the following procedure.

- | Step | Action |
|------|--|
| 1 | From the Layout menu, select Parameter Sweeps .
<i>The Parameter Sweeps dialog box appears.</i> |
| 2 | Select the Power column (all cells).
<i>The selected cells are highlighted.</i> |
| 3 | Under Spread Tools , click Linear .
<i>The Parameter Iteration Spread dialog box appears (see Figure 4).</i> |
| 4 | In Start Value , type:
-10 |
| 5 | In End Value , type:
10 |
| 6 | To set the values, click OK .
<i>The Parameter Iteration Spread dialog box closes.</i> |
| 7 | To return to the Main layout , click OK .
<i>The Parameter Sweeps dialog box closes.</i> |

Figure 4 Entering values for all iterations



Running the simulation

To run the simulation, perform the following procedure.

Step	Action
-------------	---------------

- | | |
|----------|---|
| 1 | Click Calculate . |
| 2 | Click the Run button.
<i>The calculation progress appears in the Calculations dialog box window.</i> |

Obtaining results using the Report Page

To create a graph showing the BER versus the power, you must obtain the results generated by the **BER Analyzer**.

To obtain results using the **Report Page**, perform the following procedure.

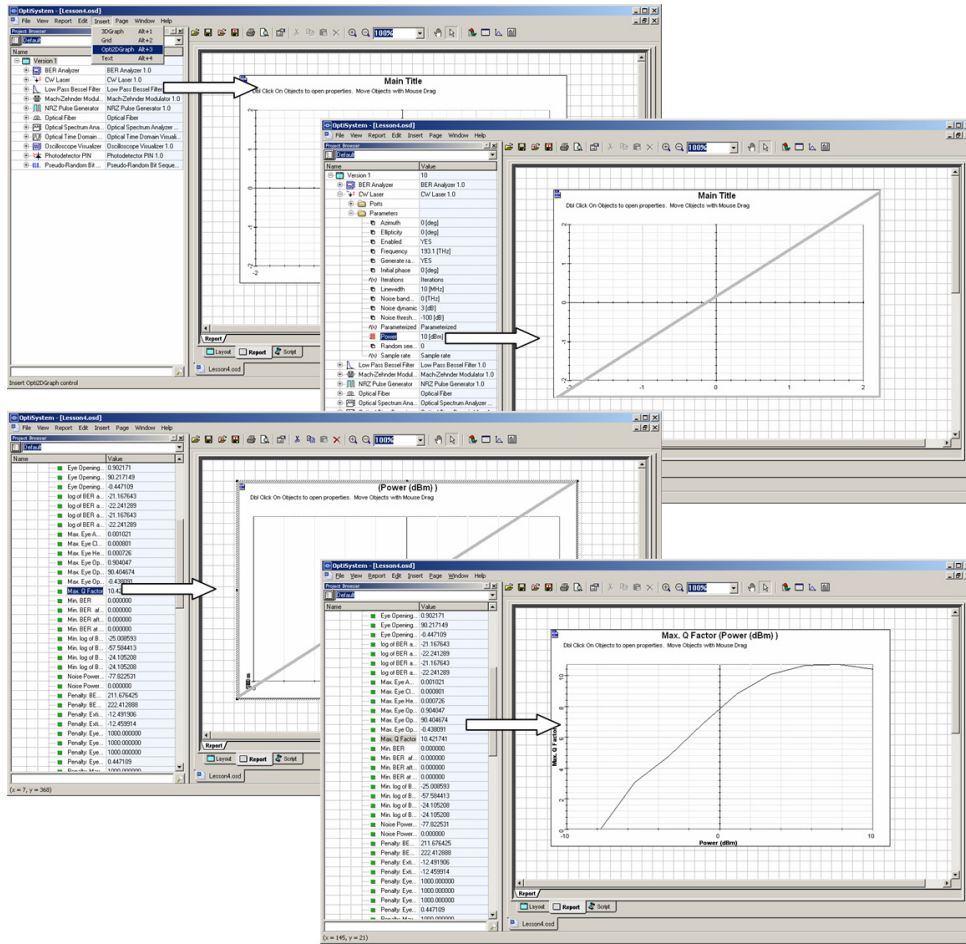
Step	Action
-------------	---------------

- | | |
|----------|---|
| 1 | Select the Report tab in the Project layout window.
<i>The Report window appears.</i> |
| 2 | Click the Opti2DGraph button on the Report toolbar and draw a 2D graph in the report window.
<i>The 2D graph appears in the Report window.</i> |
| 3 | In the Project Browser , select the parameter Power from the CW Laser , and drag it to the 2D graph X-axis (a grey triangle appears in the graph in the bottom right corner).
<i>The graph appears in the Report window.</i> |
| 4 | In the Project Browser, select the Min. log of BER result from the BER Analyzer component, and drag it into the Y-axis of the graph (the grey triangle appears in the top left corner).
<i>The result is plotted in the graph.</i> |



LESSON 4: PARAMETER SWEEPS — BER x INPUT POWER

Figure 5 Building the graph



Browsing parameter sweep iterations

You can open the **BER Visualizer** and obtain the Eye diagrams and graphs, such as Q-factor and BER, for different iterations changing the current sweep iteration.

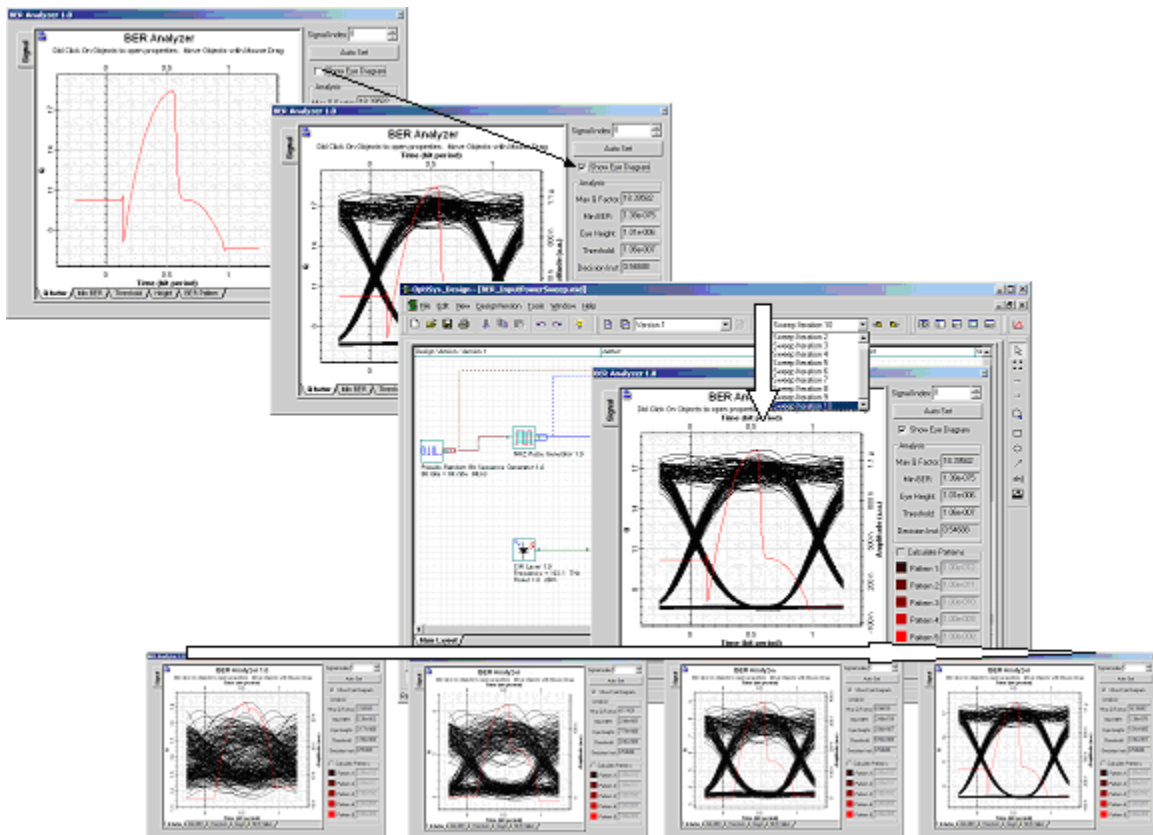
To browse parameter sweep iterations, perform the following procedure.

Step Action

- 1 In the **Main layout**, double-click the **BER Analyzer**.
*The **BER Analyzer** graph opens.*
- 2 Select the **Show Eye Diagram** check box.
*The graph changes to display the **Eye Diagram**.*
- 3 In the toolbar, change the Sweep iteration (see [Figure 6](#)).

Note: You obtain different results for each iteration, one for each input power value.

Figure 6 Browsing through the parameter sweep iterations, the Eye diagram, and Q-Factor for each value of laser power



Combining graphs from sweep iterations

Visualizers can generate graphs for all sweep iterations.

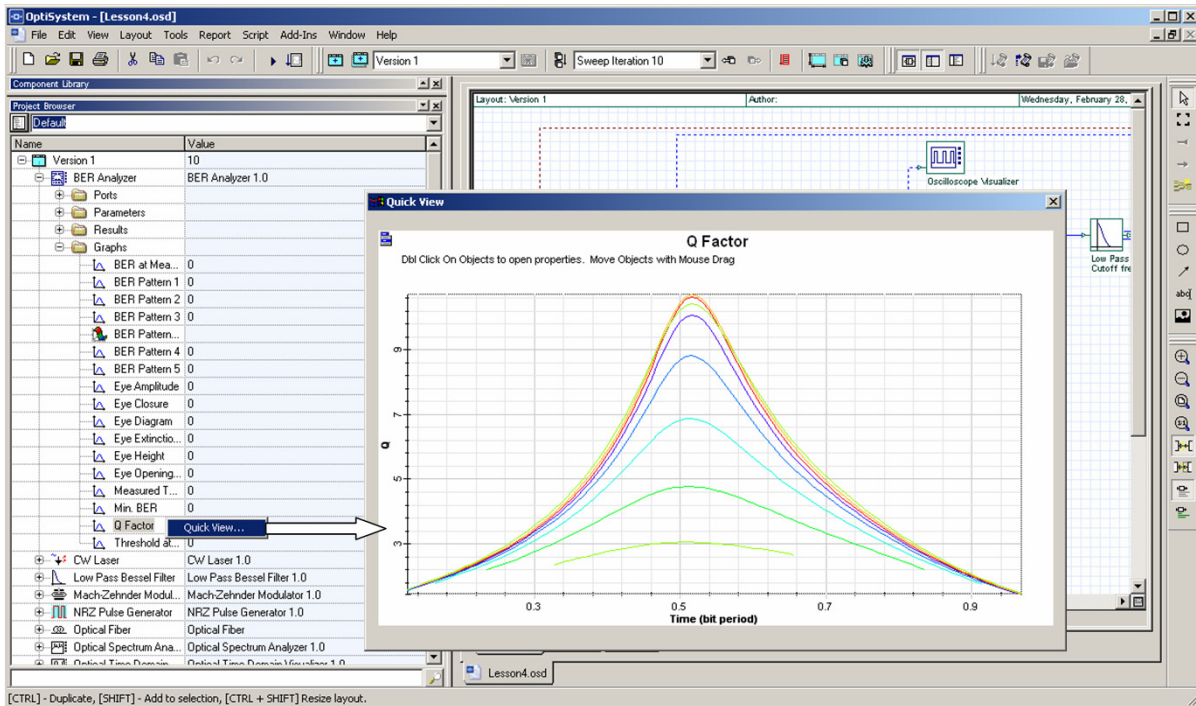
You can access the graphs for all sweep iterations by using the **Graph** subfolder in the **Project Browser**.

Note: A visualizer will display only the current sweep iteration when you access the graphs by double-clicking on the graph and browsing for different iterations.

To combine graphs from sweep iterations, perform the following procedure.

- | Step | Action |
|------|--|
| 1 | If the Project Browser is closed: <ul style="list-style-type: none"> • From the Main toolbar, select View > Project Browser or <ul style="list-style-type: none"> • press Ctrl+2. |
| 2 | Expand the Graphs subfolder for the BER Analyzer .
<i>The list of available graphs appears.</i> |
| 3 | Right-click Q Factor , and select Quick view ,
or |
| 4 | Drag and drop the graph into the Report Page .
<i>A graph appears and displays the iterations for each input power value (see Figure 7).</i> |

Figure 7 Multiple iterations graph



Lesson 5: Bidirectional Simulation — Working with multiple iterations

Besides the typical global simulation parameters of OptiSystem, such as Bit rate, Samples per bit and Sequence length, there are two additional parameters that are very important when designing bidirectional systems or systems with closed loops, such as ring networks.

These parameters are

- Iterations
- Initial delay

Iterations force a component to calculate again until it generates, at the output ports, the Iterations number of signals.

Initial delay forces a component to generate a null signal for each output port.

Because the majority of OptiSystem components will not calculate if a signal is not available at the input ports, the Iterations and Initial delay parameter will help the user to avoid a situation in which the simulation will not perform because a component cannot find signals at the input port - and the system will be deadlocked.

The concept of multiple iterations and delays is very confusing, because it does not have a physical meaning. It is a simulation technique and requires an understanding of the calculation scheduler of OptiSystem.

A basic bidirectional simulation

System setup

- Using the default global parameters, we can start adding the components to design the basic bidirectional system.

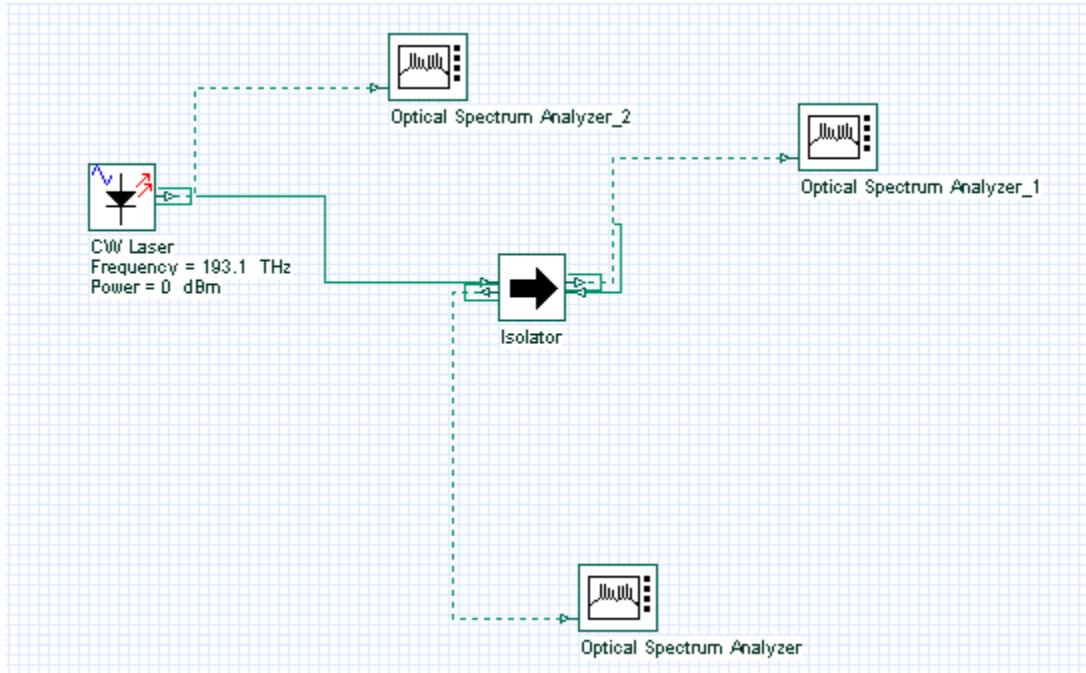
From the component library, drag and drop the following component in to the layout:

- From "Default/Transmitters Library/Optical Sources", drag and drop the "CW Laser" into the layout.
- Change the laser parameter *Linewidth* to 0 MHz.
- From "Default/Passives/Optical/Isolators", drag and drop the "Isolator" components into the layout.
- Change the Isolator parameter *Insertion loss* to 3 dB.
- From "Default/Visualizers Library/Optical", drag and drop 3 of the "Optical Spectrum Analyzer" into the layout.



The next step is to connect the components according to the Figure 1. Do not forget to connect the Isolator Input 2 at the Output 2.

Figure 1 Basic bidirectional system



Running the simulation

We can run this simulation and analyze the results:

- To run the simulation you can go to the File menu and select Calculate. You can also press Control+F5 or use the calculate button in the toolbar. After you select Calculate, the calculation dialog box should appear.
- In the calculation dialog box, press the Play button. The calculation should perform without errors.

Viewing the results - system deadlock

In order to see the results, double click at each of the OSA visualizers.

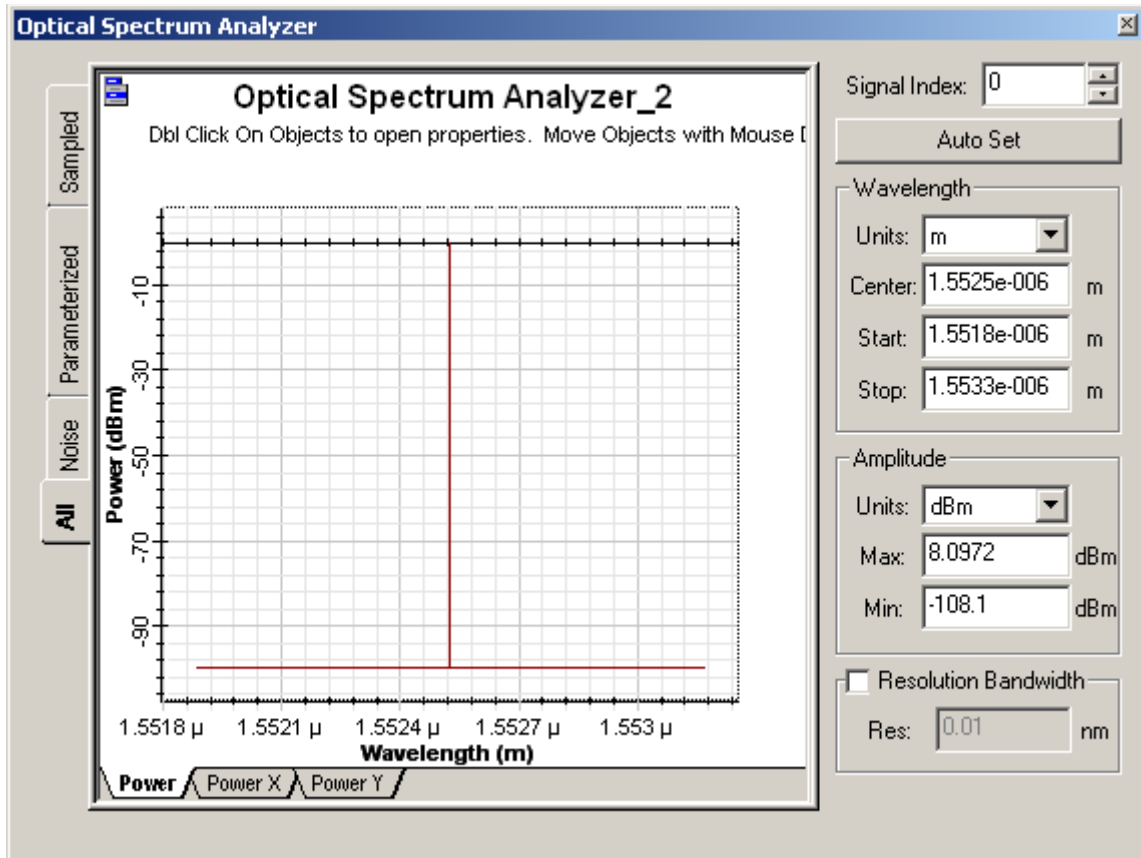
The OSA connected at the CW Laser output will have a signal as presented in Figure 2.

As expected, we have a CW signal centered at 193.1 THz with peak power around 0 dBm. However, the other two OSAs have no signals, and this is also expected.

Why is this expected? It is expected because the Isolator does not have a signal at each input port. The laser will send a signal to the first input port of the isolator. However, the isolator will not calculate if a signal is not available at the second port - the system is deadlocked.



Figure 2 CW Laser output



Using delays

What if we add a delay at the second input port of the Isolator?

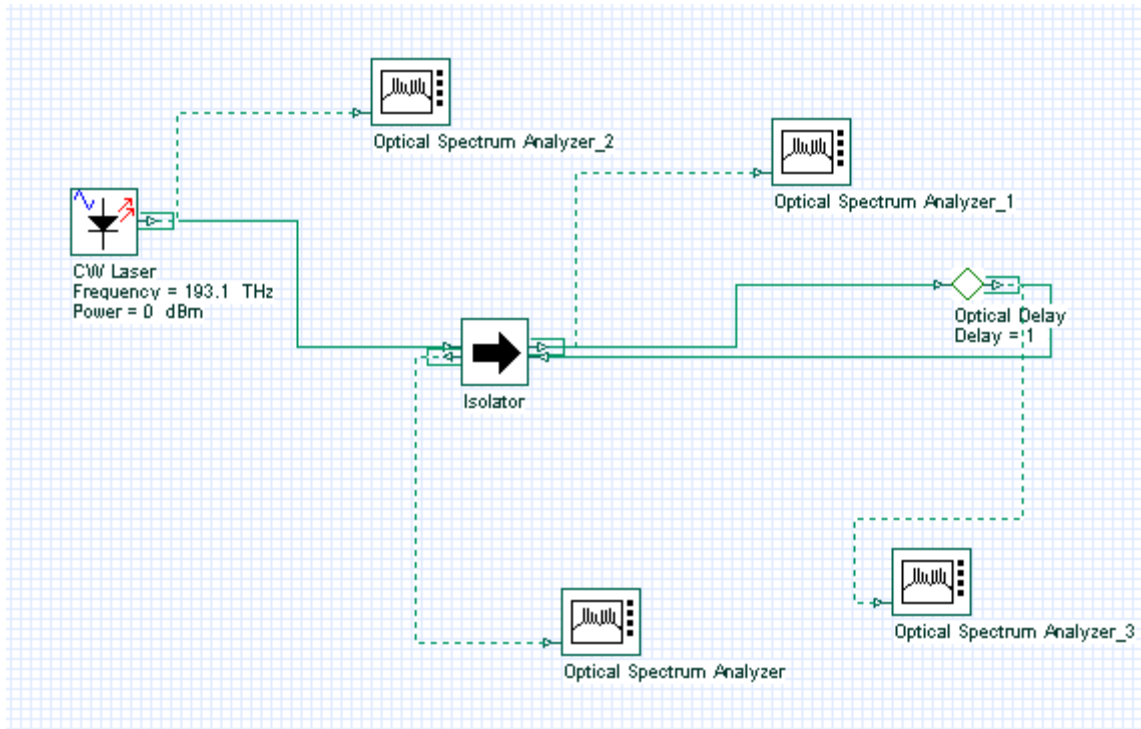
- From "Default/Tool library", drag and drop the "Optical Delay" into the layout.
- Add another OSA to the system.
- Connect the component according to Figure 3.

The delay component is important because, unlike the Isolator, it will calculate without having a signal at the input port. If the laser and the delay calculate, the Isolator will have signals available at both input ports.

Run the simulation again and open the visualizers.



Figure 3 Basic bidirectional system using a delay



Viewing the results

Let's take a look at the signal at the second port of the Isolator. As expected, the input signal was attenuated by the isolator insertion loss and the peak power is -3 dBm. The OSA at the first isolator output shows the value of the return loss of the device and the peak power is -60 dBm.

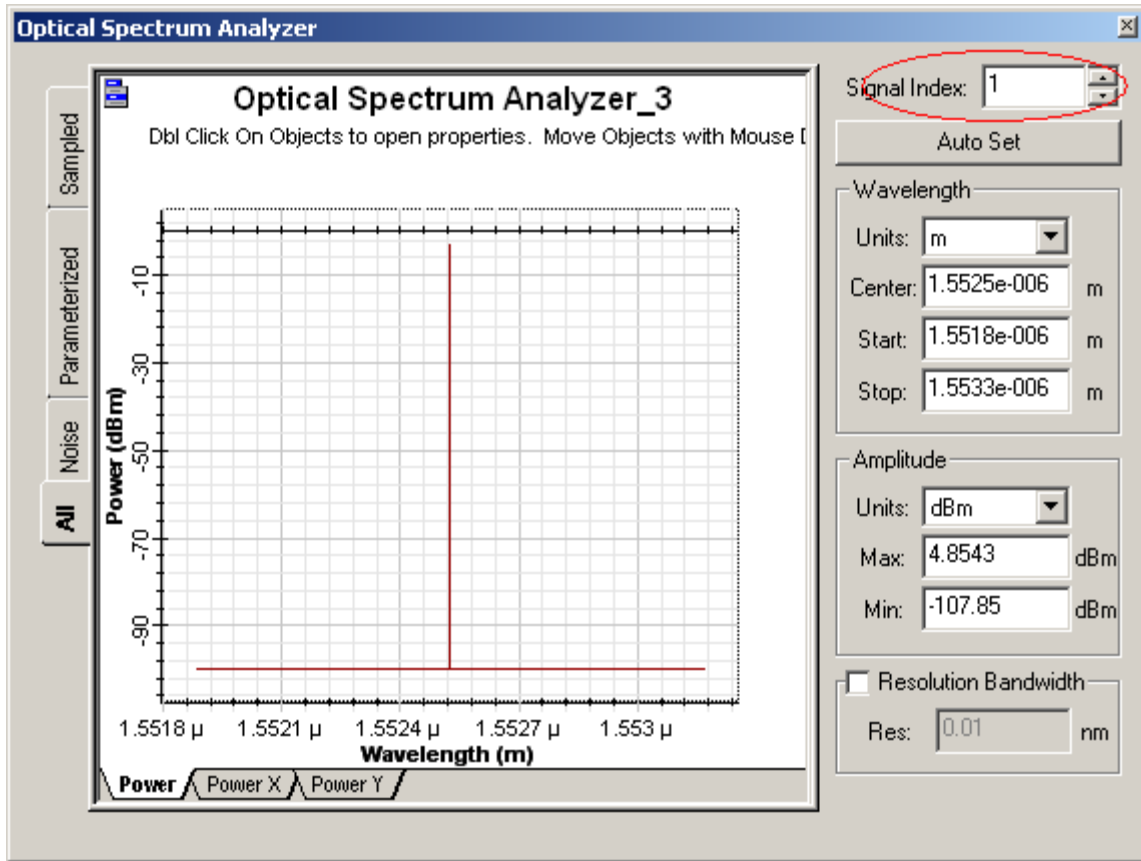
Now, let's see the signal at the Delay output. If we open the OSA at the delay output there is no signal - it is empty. This is the result of the first calculation of the delay. There was no input signal and the delay generated a null signal. This null signal and the laser signal were sent to the Isolator inputs. The isolator calculated with these two signals and generated a signal per output port.

One of the signals went back to the delay, which will then transfer the signal to its output port. This means there are two signals at the delay output: the null signal and the signal from the isolator.

In order to browse through the signals at the delay output you should use the *Signal index* parameter from the OSA. Set the signal index to 1 and you will see the signal from the isolator, with peak power of -3 dBm (Figure 4).



Figure 4 Second signal at the delay output



This means we could calculate using the isolator, however the results are not exactly what we expect in this type of bidirectional system. There is a feedback between the second isolator input and output port that did not cause any effect in our results. We should force the signal to “circulate” between the laser, isolator and delay in order to see the signal evolution in the system.

In order to force the signal to circulate multiple times in the system, we can force the laser to generate the same signal multiple times, forcing the component to calculate multiple times - because there will be signals available at the input ports of the isolator and delay.

Using *Iterations* parameter

Change the global parameter *Iterations* to 10, run the simulation again.

Viewing the results

Let's see the results from the OSAs. The laser generated 10 signals. You can change the signal index parameter at the laser output and see that the signal power does not change. This is expected, because we have 10 copies of the same signal.

We expect the second isolator output signal to have -3dBm of power. However we don't know the power at the first isolator output port. This port will have the laser input power reflected by the return loss, plus the feedback signal attenuated by the isolator insertion loss.

Look at the OSA at the isolator outputs in order to evaluate the results. Do not forget to change the signal index!

By changing the signal index you can see how the power changes per iteration. We are using 10 iterations but 2 or 3 should be enough to give a “steady-state” value for the output power of the OSAs.

Cascading devices

Let's make the system more complicated:

- Duplicate the Isolator.
- Connect the component according to Figure 5.

From what we learned until now, it seems obvious that this system will be deadlocked because the two Isolators will not have signals at the input ports.

You can run the simulation and you will see that the 10 signals from the laser will not reach all the visualizers. In order to make the system work, we should add more delays:

- Duplicate the delay twice.
- Connect the component according to Figure 6.

If you run the simulation again, the system will run properly. You should increase the OSA signals index in order to visualize the proper results.



Figure 5 Basic bidirectional system cascading two isolators

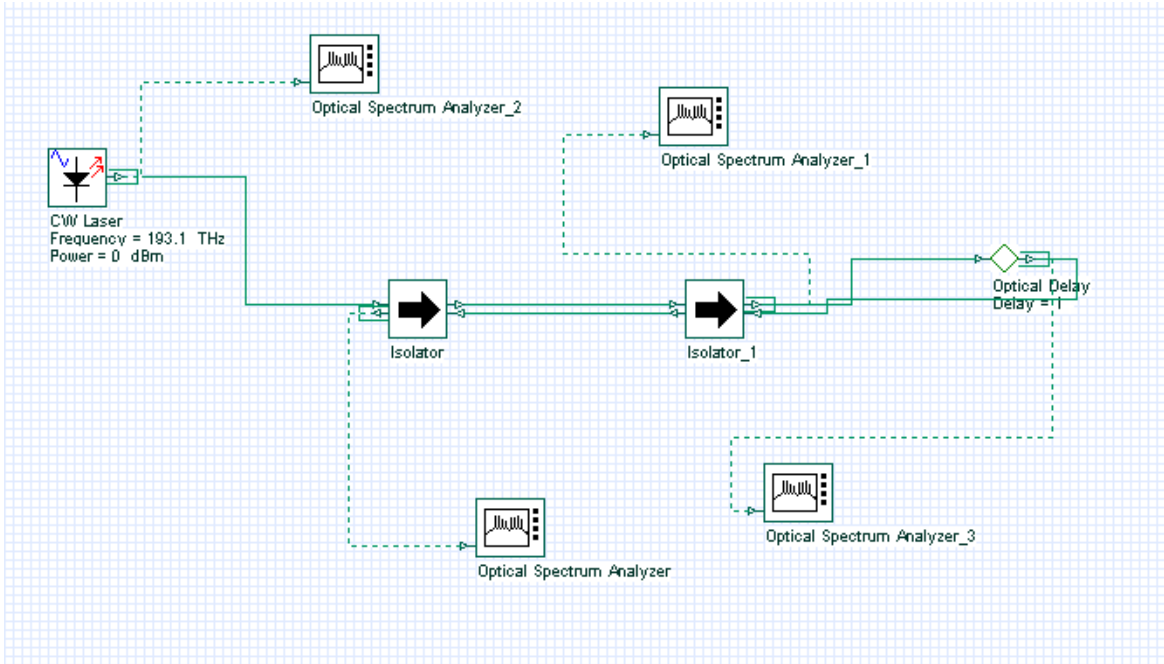
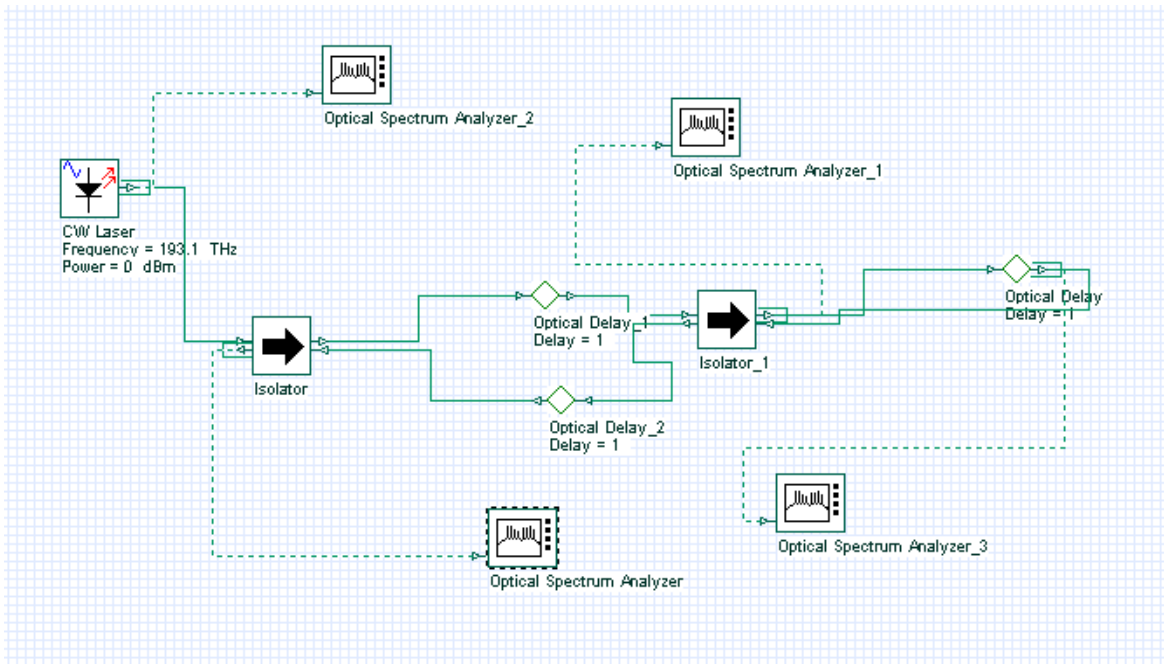


Figure 6 Basic bidirectional system cascading two isolators and delays



Using *Initial delay* parameter

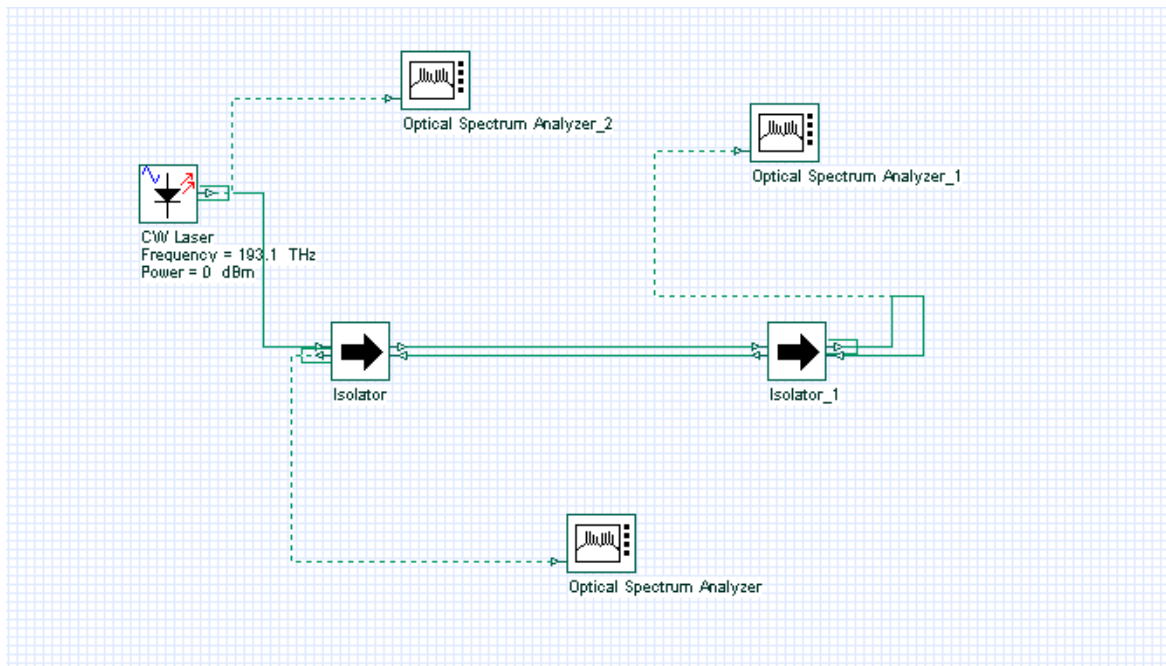
The main problem with cascading these devices is that the user should add delays per each input port, and this can be cumbersome for some designs. This is why OptiSystem allows the user to include a delay by default for each input port:

- Remove the delays.
- Connect the component according to Figure 7.
- In the global parameters, enable parameter *Initial delay*.
- Run the simulation.

This time we have a clean design, without multiple delays. The final results are the same as adding the delays.

There is one main difference between the designs of Figure 6 and 7. By enabling the global parameter initial delay, all components will generate a null signal, including the laser component. This is not true for Figure 6, in which only the delays generated the null signals.

Figure 7 Basic bidirectional system with two isolators in cascade



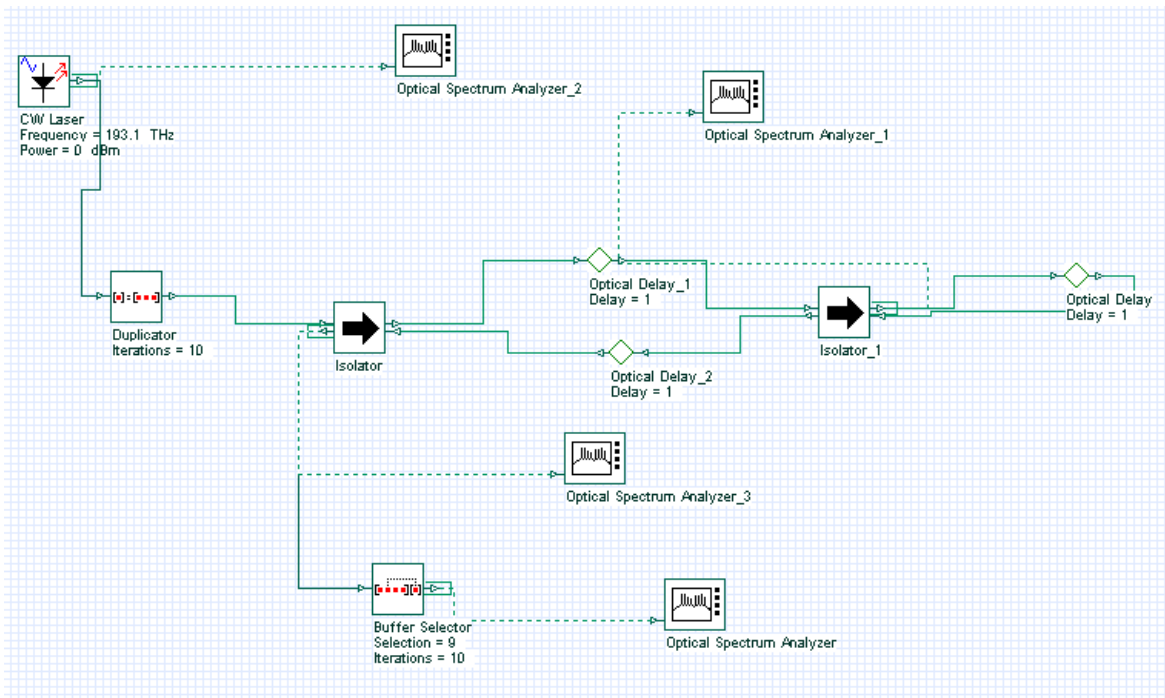
Tools library

OptiSystem Tools Library has components that can manipulate signals and signal buffers.

Signals can be duplicated, selected, initialized, etc. For example, Figure 8 presents a system that is equivalent to Figure 6, without using global parameters: Iterations = 1 and Initial delay = disabled.

- Repeat the steps to generate Figure 6 layout.
- Set the global iterations parameter to 1.
- Set the *Initial delay* parameter to disabled status.
- From "Default/Tool library", drag and drop the "Duplicator" into the layout.
- Change the parameter *Iterations* mode to Normal. Set the value to 10.
- From "Default/Tool library", drag and drop the "Buffer selector" into the layout.
- Change the parameter *Iterations* and Selection mode to Normal. Set the value of Iterations to 10 and Selection to 9.
- Connect the component according to Figure 8.
- Run the simulation.

Figure 8 Bidirectional system using duplicator, delay and selector



This time the duplicator component will do the job of generating multiple signals. It will generate 10 copies of the signal from the laser. The selector will select the last signal from the input buffer and sent it to the output port.

LESSON 5: BIDIRECTIONAL SIMULATION — WORKING WITH MULTIPLE ITERATIONS

This means the signal index parameter of the OSA connected to the Selector will have only one value:0. This is because the other signals were filtered by the selector component.







Lesson 6: Time-Driven Simulation — Working with individual samples

One of the main features of OptiSystem is the mixed signal representation.

Individual sample is a signal type that allows the users to simulate time-driven systems in the electrical and optical domain. By using time-driven simulation, users can create designs that have closed loops and feedbacks.

Additionally, the optical individual sample allow for simulation of WDM systems by using multiple bands. Each band shares the same sample rate and time window, but they can have different center frequencies.

To manipulate individual samples, there are four new components available in the component library:

Component name	Folder	Icon
Convert To Electrical Individual Samples	Signal Processing Library/Tools/Electrical/	
Convert From Electrical Individual Samples	Signal Processing Library/Tools/Electrical/	
Convert To Optical Individual Samples	Signal Processing Library/Tools/Optical/	
Convert From Optical Individual Samples	Signal Processing Library/Tools/Optical/	

To verify if a component is compatible with individual samples, you should check OptiSystem component library documentation.



A Time-Driven Simulation

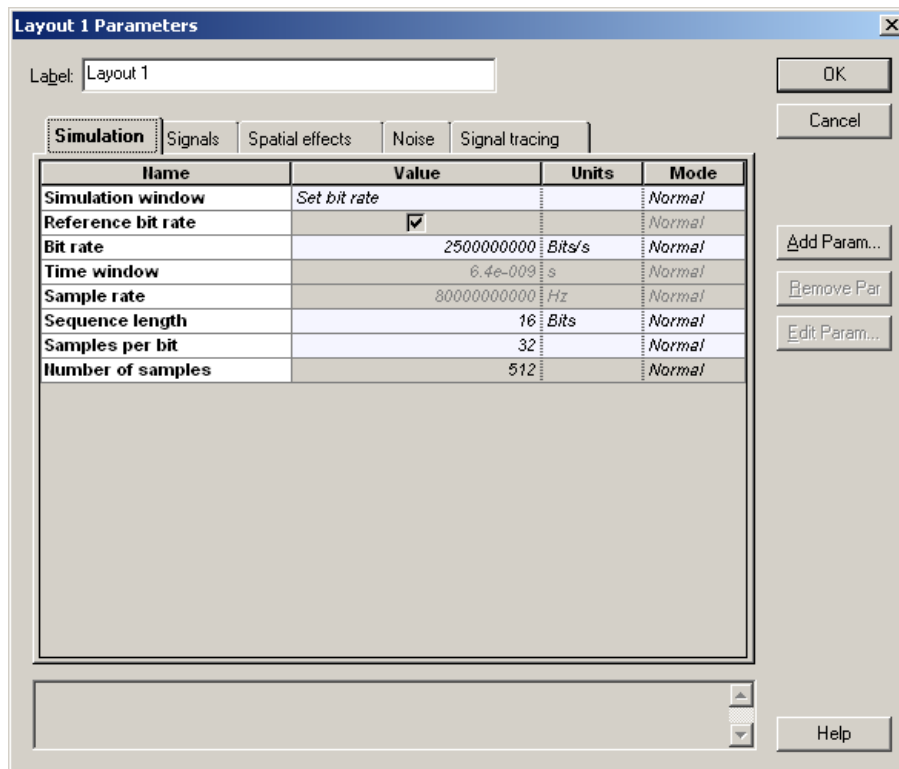
Global parameters

To reduce the calculation time, we should reduce the number of samples of the signals that will be calculated during the simulation.

To reduce the number of samples, we can reduce the sequence length and the number of samples per bit.

- In the global parameters dialog box, change the parameter *Sequence length* to 16, and the *Samples per bit* to 32. The *Number of samples* parameter should be 512 (Figure 1).

Figure 1 Global parameters



This means that we will have 512 individual samples at the output of the Convert To Electrical Individual Samples component, and 512 individual samples per WDM channel at the output of the Convert To Optical Individual Samples component.



System setup

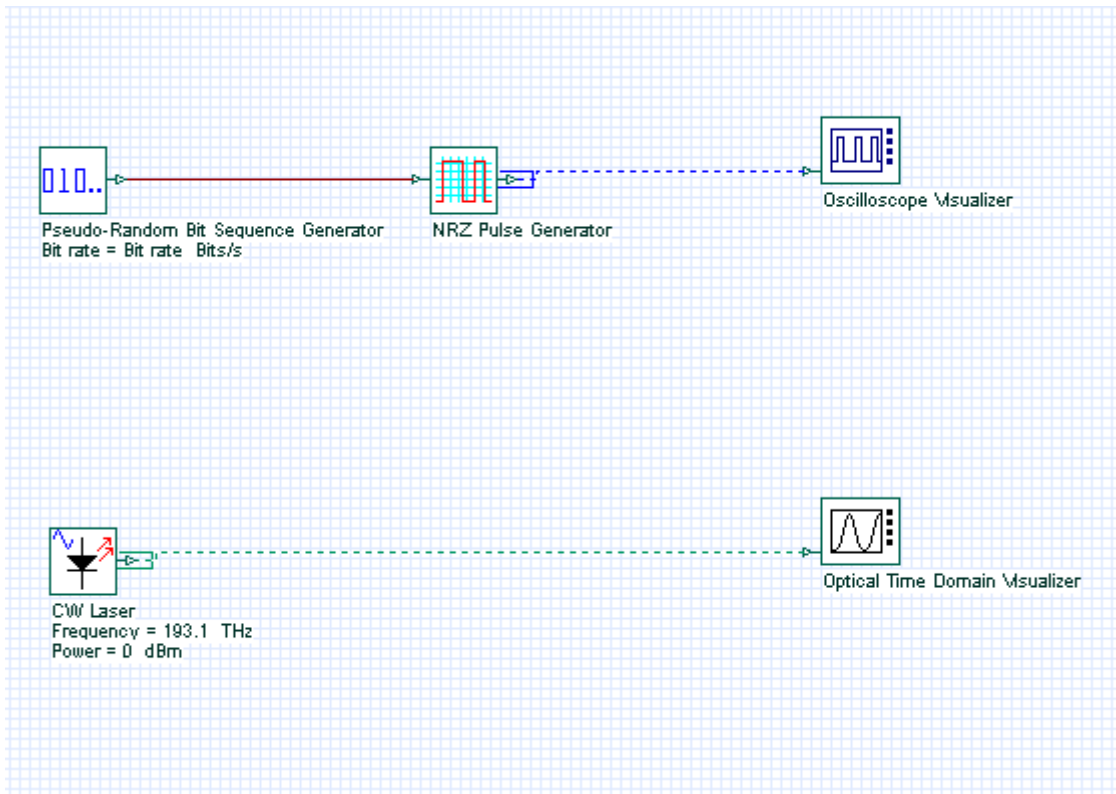
We can start adding the components to design the basic time-driven system.

From the component library, drag and drop the following component in to the layout:

- From "Default/Transmitters Library/Optical Sources", drag and drop the "CW Laser" into the layout.
- From "Default/Transmitters Library/Bit Sequence Generators", drag and drop the "Pseudo-Random Bit Sequence Generator" into the layout.
- From "Default/Transmitters Library/Pulse Generators/Electrical", drag and drop the "NRZ Pulse Generator" into the layout.
- From "Default/Visualizers Library/Optical", drag and drop the "Optical Time Domain Visualizer" into the layout.
- From "Default/Visualizers Library/Electrical", drag and drop the "Oscilloscope Visualizer" into the layout.

The next step is to connect the components according to the Figure 2.

Figure 2 First layout



Running the simulation

We can run this simulation and analyze the results:

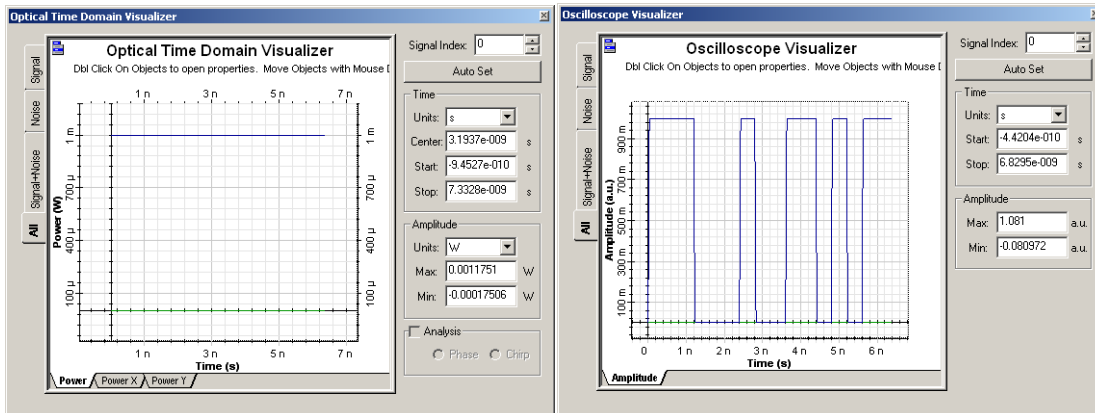
- To run the simulation, you can go to the File menu and select Calculate. You can also press Control+F5 or use the calculate button in the toolbar. After you select Calculate, the calculation dialog box should appear.
- In the calculation dialog box, press the Play button.

Viewing the results

To see the results, double click at each of the time domain visualizers. The optical and electrical signals are available in Figure 3.

These are the graphs of the sampled signals. To convert these signals into individual samples, we can use the components from Table 1.

Figure 3 CW Laser output and NRZ pulse generator output

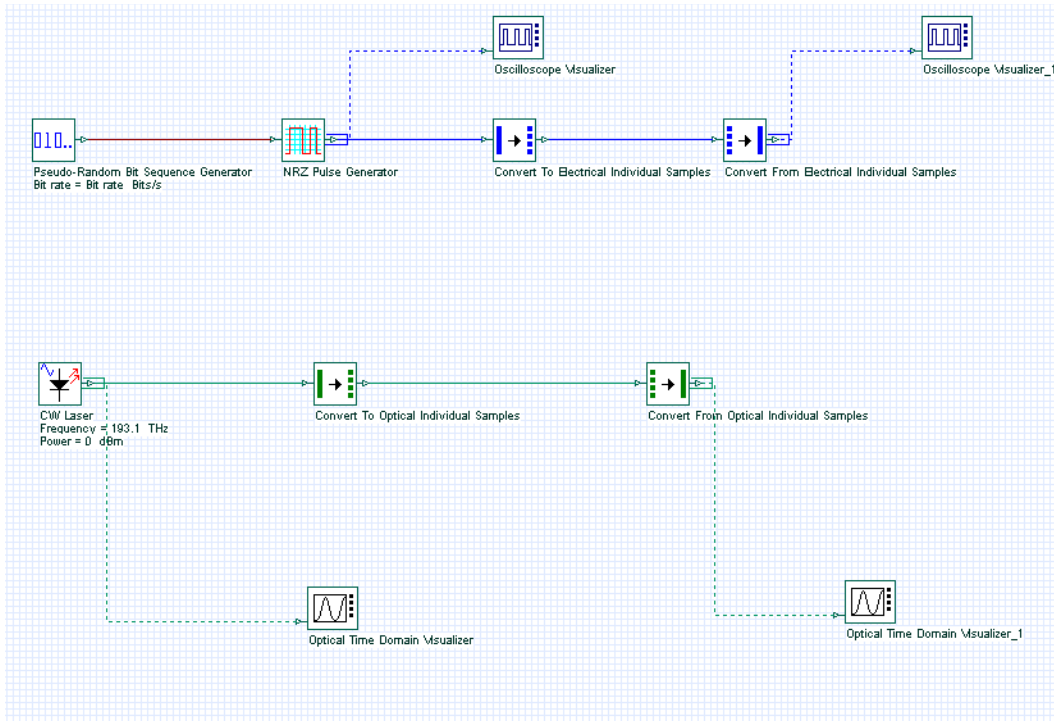


System setup - Individual Samples

- Drag and drop the components from Table 1 into the layout.
- Duplicate the time domain visualizers
- Connect the components according to the Figure 4.
- Run the simulation



Figure 4 Layout using the individual sample tools



Viewing the results

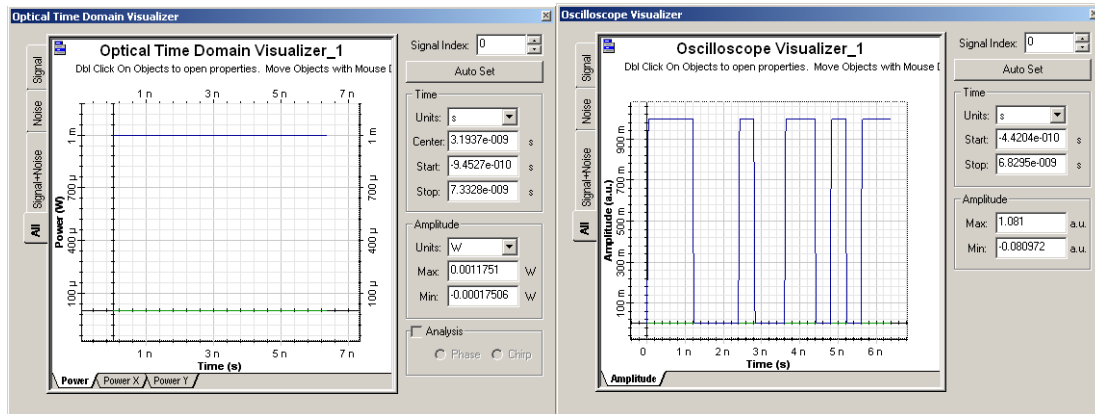
To see the results, double click at each of the time domain visualizers.

The optical and electrical signals are available in the Figure 5. You will notice that the signals are the same as in Figure 3.

The CW laser will send a sample signal to the 'Convert to...' component. This component will generate 512 samples and send each one of them to the 'Convert from...' component. The 512 samples will be combined into a signal that should be the same as the original signal from the laser. The same process applies to the electrical signal.



Figure 5 'Convert from...' components outputs



Important remarks

- Visualizers cannot display individual samples. You should use a 'Convert From...' component before a visualizer.
- Check if a component is compatible with individual samples before creating a project.
- The project scope that requires simulation using individual samples must be confined between the 'Convert To...' and 'Convert From...' components.
- For each individual sample a component will run one calculation. This means the simulation will be slower when using individual samples.
- Faster simulation can be obtained by disabling calculation scheduler messages.

Filtering using individual samples

There are two type of filtering techniques used in OptiSystem when working with individual samples.

If a filter has the parameters *Digital filter* and *Digital filter order* available, the user can enable the FIR filter and enter the number of coefficients for the filter.

Be aware of the FIR filter latency because the filter will introduce a delay equal to one half of the number of coefficients.

If the filter does not have these parameters or if the 'Digital filter' parameter is disabled, the individual samples will be attenuated by the value at the filter center frequency

In this case, we are assuming that the filter bandwidth is much wider than the sample rate. For optical signals, there attenuation will be wavelength dependent.

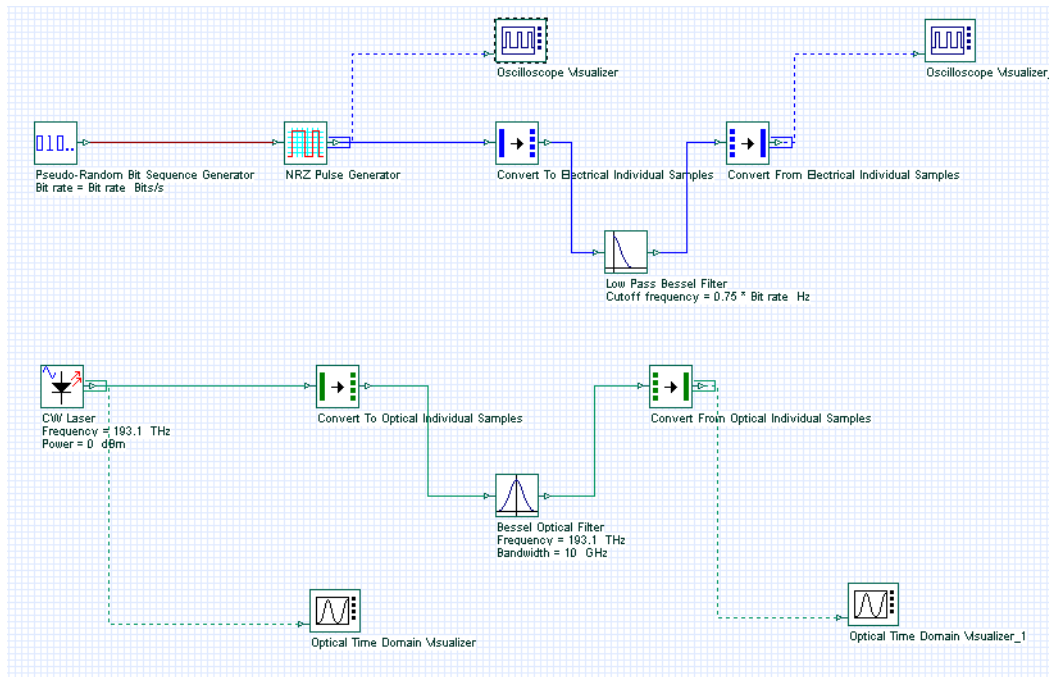
This means that every individual sample will have a different value of attenuation depending on the filter shape.

- From "Default/Filters Library/Electrical", drag and drop the "Low Pass Bessel Filter" into the layout.
- From "Default/Filters Library/Optical", drag and drop the "Bessel Optical Filter" into the layout.



- For both components, enable the parameter *Digital filter*.
- Connect the components according to the Figure 6.
- Run the simulation.

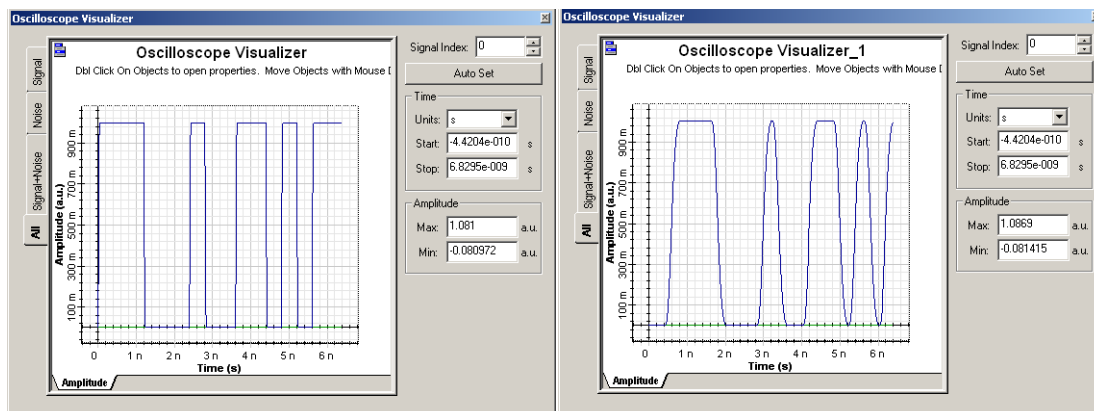
Figure 6 Filtering individual samples



Viewing the results

Compare the signals from the two Oscilloscopes (Figure 7). You will see that the second signal is delayed and filtered. This is related to the FIR filter used in the low pass filter. The same applies to the optical signal.

Figure 7 Electrical signal before and after FIR filter



Notes:



Lesson 7: Optical Amplifiers — Designing optical fiber amplifiers and fiber lasers

OptiSystem allows the design and simulation of optical fiber amplifiers and fiber lasers.

The projects presented here are available under OptiSystem installation folder samples\Optical amplifiers.

This tutorial will describe part of the library of optical amplifiers.











There are four categories of components in the library:

- EDFA
- Raman
- SOA
- Waveguide amplifiers

The EDFA folder contains the Erbium-doped fiber model and other models to allow the simulation of EDFAs in the steady-state condition. Furthermore, the folder includes models to simulate erbium-ytterbium codoped fiber, ytterbium-doped fiber and dynamic models for the erbium and ytterbium fibers.





Table 1 Doped fiber models

Component name	Folder	Icon
Erbium Doped Fiber	Amplifiers Library/Optical/EDFA	
EDFA	Amplifiers Library/Optical/EDFA	
EDFA Black Box	Amplifiers Library/Optical/EDFA	
Optical Amplifier	Amplifiers Library/Optical/EDFA	
EDFA Measured	Amplifiers Library/Optical/EDFA	
EDF Dynamic	Amplifiers Library/Optical/EDFA	
EDF Dynamic Analytical	Amplifiers Library/Optical/EDFA	
Er-Yb Codoped Fiber	Amplifiers Library/Optical/EDFA	
Yb Doped Fiber	Amplifiers Library/Optical/EDFA	
Yb Doped Fiber Dynamic	Amplifiers Library/Optical/EDFA	

The second category is the Raman models. These components allow the simulation of Raman fiber amplifiers. The Raman folder includes the Raman models for steady state and dynamic conditions.













Table 2 Raman models




Component name	Folder	Icon
Raman Amplifier - Average Power Model	Amplifiers Library/Optical/Raman	
Raman Amplifier - Dynamic Model	Amplifiers Library/Optical/Raman	

The tutorial will use passive components that allow the design of amplifiers and lasers. These components are spread in different folders at the OptiSystem component library and some of them are described at Table 3. These are the equivalent components to those found in OptiAmplifier.



Table 3 Passive components

Component name	Folder	Icon
Isolator Bidirectional	Passives Library/Optical/Isolators	
Reflector Bidirectional	Passives Library/Optical/Reflectors	
Attenuator Bidirectional	Passives Library/Optical/Attenuators	
Pump Coupler Bidirectional	Passives Library/Optical/Couplers	
Coupler Bidirectional	Passives Library/Optical/Couplers	
Tap Bidirectional	Passives Library/Optical/Taps	
Circulator Bidirectional	Passives Library/Optical/Circulators	
Polarization Combiner Bidirectional	Passives Library/Optical/Polarization	
Reflective Filter Bidirectional	Filters Library/Optical/	
3 Port Filter Bidirectional	Filters Library/Optical/	

Component name	Folder	Icon
Transmission Filter Bidirectional	Filters Library/Optical/	
Nx1 Mux Bidirectional	WDM Multiplexers Library/Multiplexers	
2x2 Switch Bidirectional	Network Library/Optical Switches	

Optical Amplifiers

Global parameters

The first step when using OptiSystem is setting the global parameters.

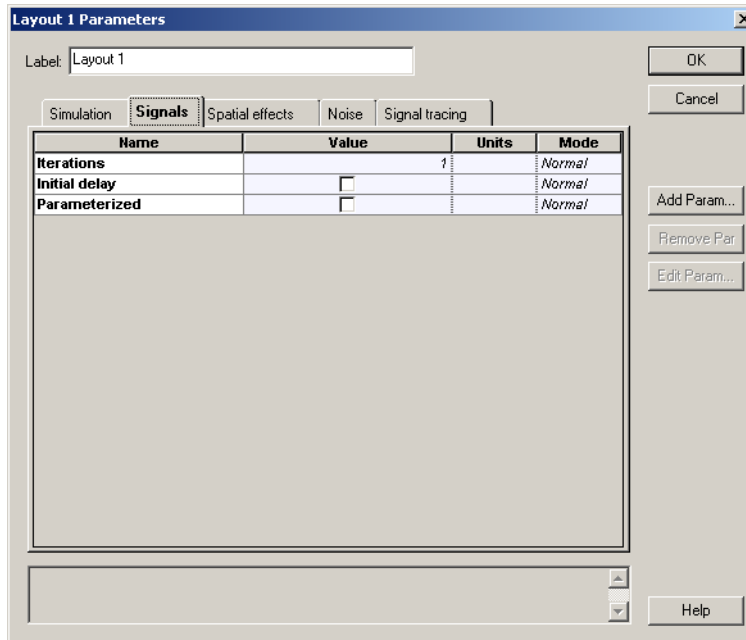
As we already know, one of the main parameters is the time window, calculated from the bit rate and sequence length.

For the amplifier and laser design, there are other important parameters that will define the number of iterations in the simulations and introduce an initial delay in the in the simulation.



These parameters are the *Iterations* and *Initial delay*, and are available in the global parameters window (Figure 1).

Figure 1 Global parameters: Signals tab

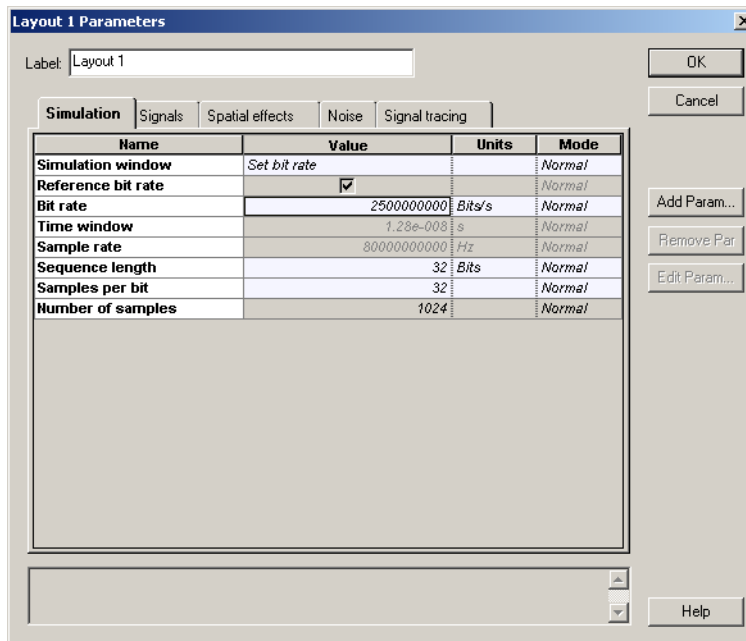


For this tutorial we will use the default parameters, except for some global parameters.

- In the global parameters dialog box, change the parameter *Bit rate* to 2.5e9, *Sequence length* to 32, and *Samples per bit* to 32. The *Time window* parameter should be 1.28e-8 s (Figure 2).



Figure 2 Global parameters: Simulation parameters tab



System setup

After setting the global parameters, we can start adding the components to design the basic EDFA design.

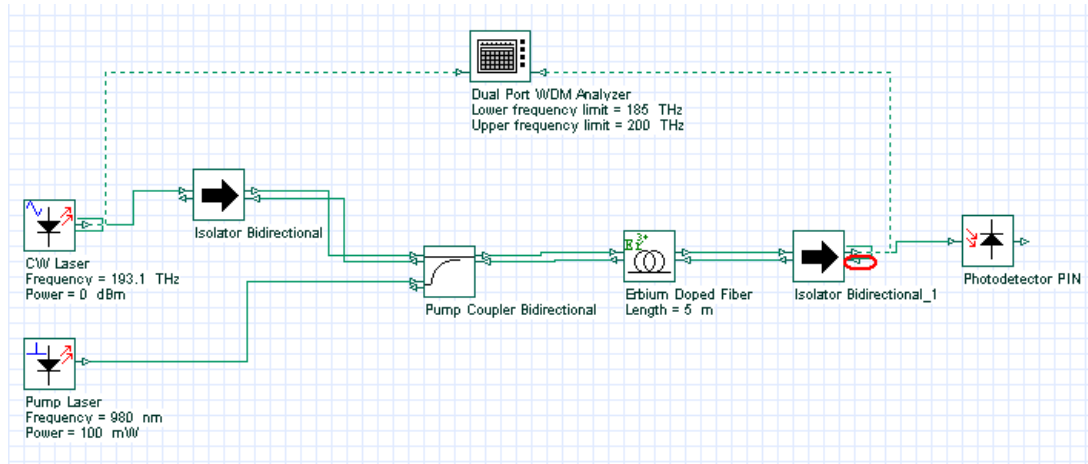
From the component library, drag and drop the following component in to the layout:

- From "Default/Transmitters Library/Optical Sources", drag and drop the "CW Laser" and the "Pump Laser" into the layout.
- From "Default/Amplifiers Library/Optical/Doped Fibers", drag and drop the "Erbium Doped Fiber" component into the layout.
- From "Default/Passives Library/Optical/Isolators", drag and drop two "Isolator Bidirectional" components and a "Pump Coupler Bidirectional" component into the layout.
- From "Default/Receivers Library/Photodetectors", drag and drop a "Photodetector PIN" component into the layout.
- From "Default/Visualizers Library/Optical", drag and drop the "Dual Port WDM Analyzer" into the layout.

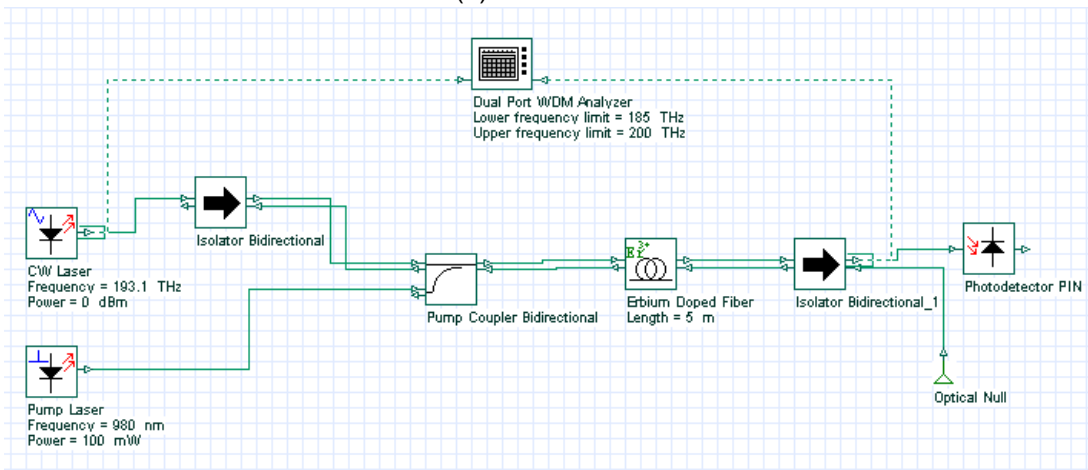
The next step is to connect the components according to Figure 3. As we can see, there is one input port in the isolator opened, so to be able to run the simulation it is necessary to include an optical null in the design (Figure 3 (b)).



Figure 3 EDFA layout



(a)



(b)

Signals tab

Although all components are in the layout and correctly connected, we are not able yet to run the simulation properly.

First, because we are considering the signals propagating in both directions, we will need more than one global iteration for the results in the system to converge.

Secondly, in the first iteration there are no backward signals in the left input port of the bidirectional components, such as the isolator and pump coupler. This will make the simulation stop.

To resolve the first condition, you just have to increase the number of iterations.

To resolve the second condition, there are two possible solutions: We can enable in the Signals Tab the *Initial delay* parameter (Figure 4) or we can introduce in the layout the component "Optical Delay" (Figure 5).



Figure 4 Global parameters - Increasing the number of iterations and enabling *Initial Delay*

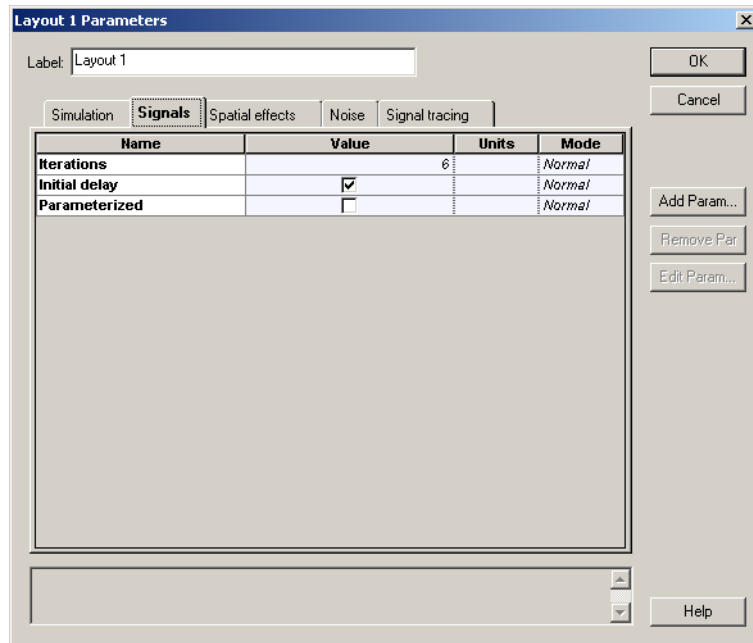
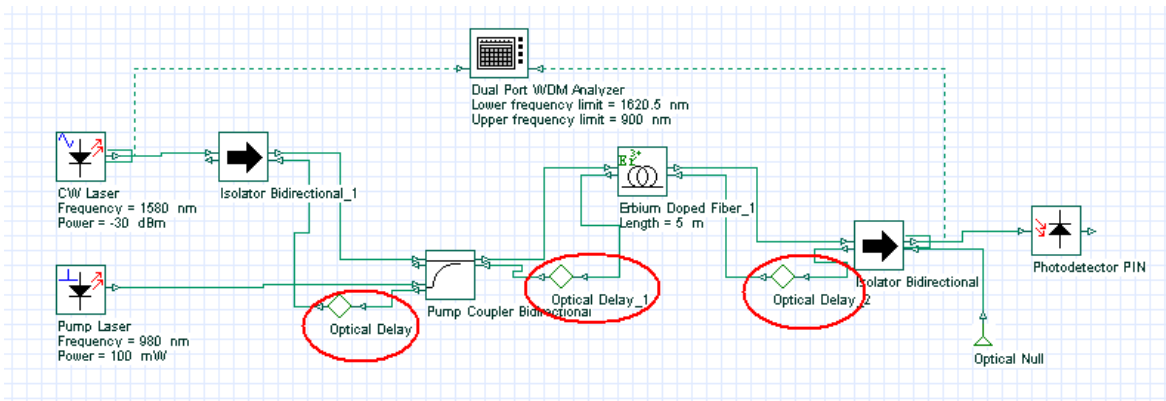


Figure 5 Including the "Optical Delay" components in the layout



Running the simulation

We can run this simulation for the system shown in Figure 3 and analyze the results:

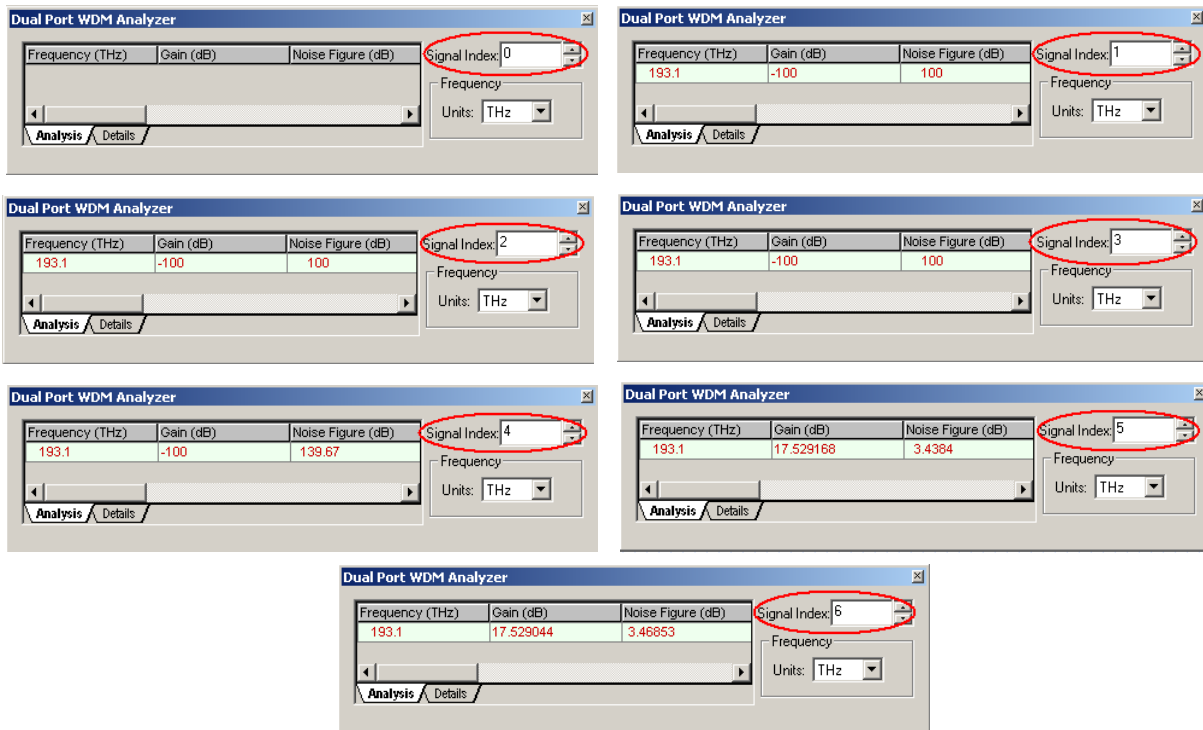
- To run the simulation, you can go to the File menu and select Calculate. You can also press Control+F5 or use the calculate button in the toolbar. After you select Calculate, the calculation dialog box should appear.
- In the calculation dialog box, press the Play button. The calculation should perform without errors.

Viewing the results

To see the results, double click the Dual Port WDM Analyzer. You can see the results of each iteration by browsing in the *Signal Index* parameter.

Figure 6 shows the results in the WDM analyzer for the 6 iterations and initial delay.

Figure 6 Dual Port WDM Analyzer views for different iterations



Running the simulation

To compare the results in the two different designs, Figure 3 and Figure 5, we can run the same simulation for the system showed in Figure 5 and analyze the results:

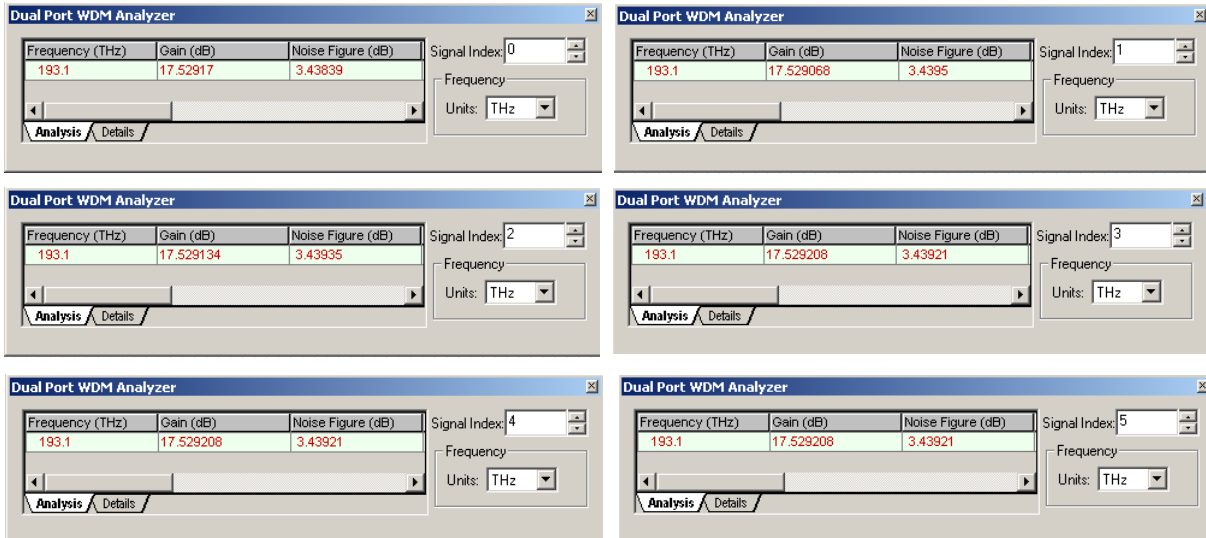
- To run the simulation, you can go to the File menu and select Calculate. You can also press Control+F5 or use the calculate button in the toolbar. After you select Calculate, the calculation dialog box should appear.
- In the calculation dialog box, press the Play button. The calculation should perform without errors.

Viewing the results

To see the results, double click the Dual Port WDM Analyzer. You can see the results of each iteration by browsing in the *Signal Index* parameter.

Figure 7 shows the results in the WDM analyzer for the 6 iterations.

Figure 7 Dual Port WDM Analyzer views for different iterations



As we can see, the second design converges faster than the design with 'Initial Delay'. The design from Figure 3 takes more iteration because of the 'Initial delay'.

Viewing the gain versus wavelength

We will build the graph gain as a function of the signal wavelength.

Starting using the layout from Figure 5:

- Double-click on the CW Laser component. The component properties will appear. See Figure 8 (a).
- Change the unit of the *Frequency* parameter to nm.
- Select the parameter that you want put in sweep mode. In this case the parameter is *Frequency*. Change the mode of the *Frequency* parameter in the CW Laser component from Normal to Sweep. See Figure 8 (b).
- Select the Parameter Sweeps tab at the CW laser properties, or at the menu tool bar, or at the Layout menu. See Figure 9.

Figure 8 CW Laser properties (a) Frequency parameter in Normal and (b) Sweep mode

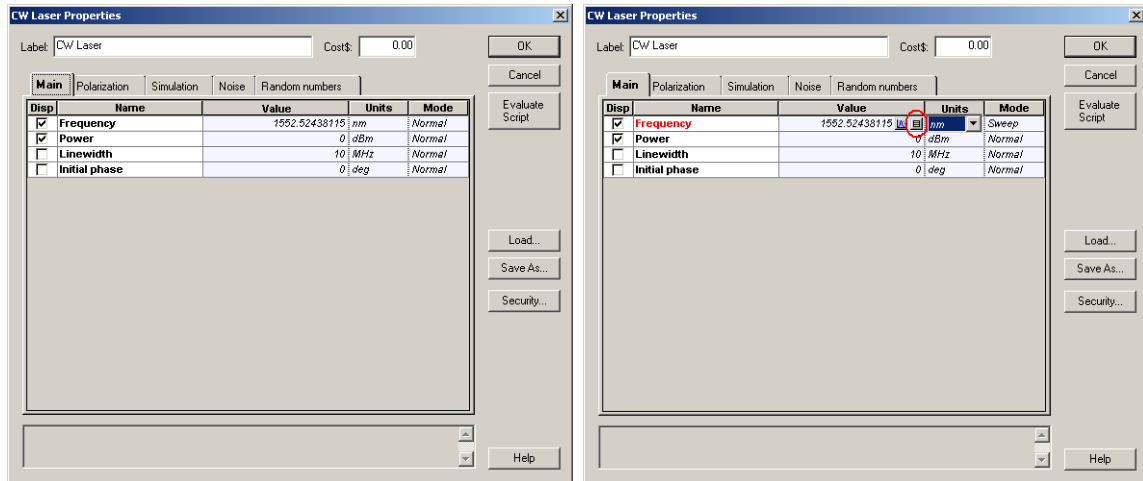
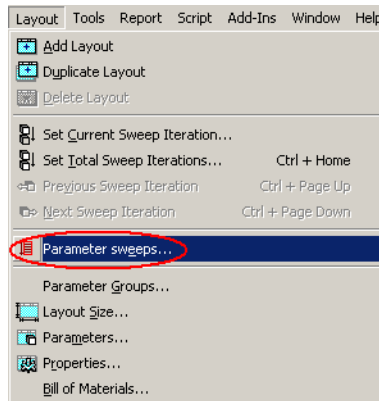


Figure 9 Parameter Sweeps at (a) the menu tool bar and (b) at the Layout menu

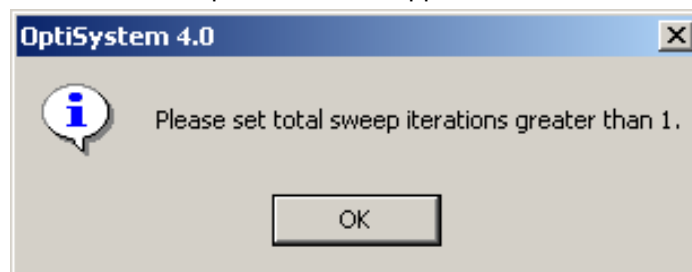


(a)

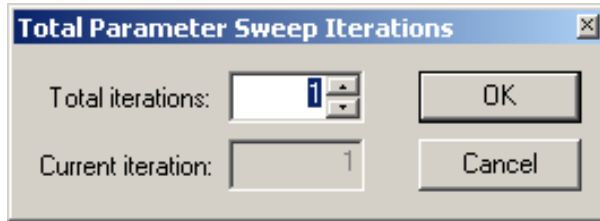


(b)

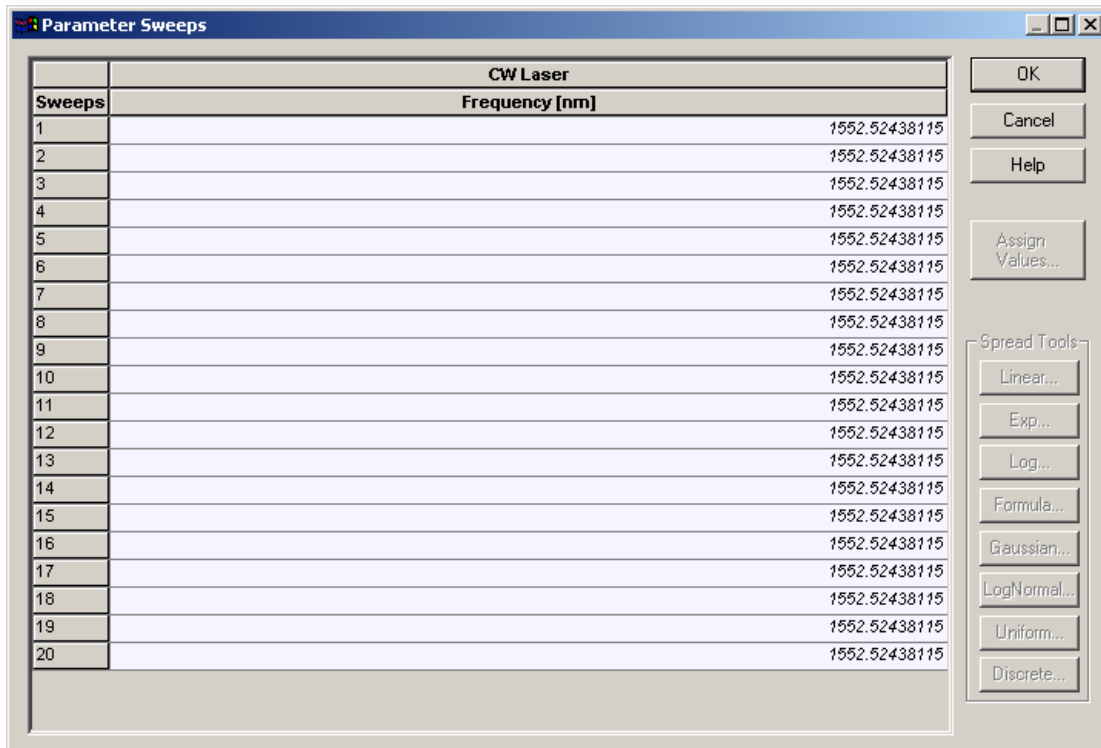
- After you click in the Parameter Sweeps tab, a window asking you to set the total number of sweep iterations will appear.



- Click on OK and the windows to enter the number of sweep iterations will appear.



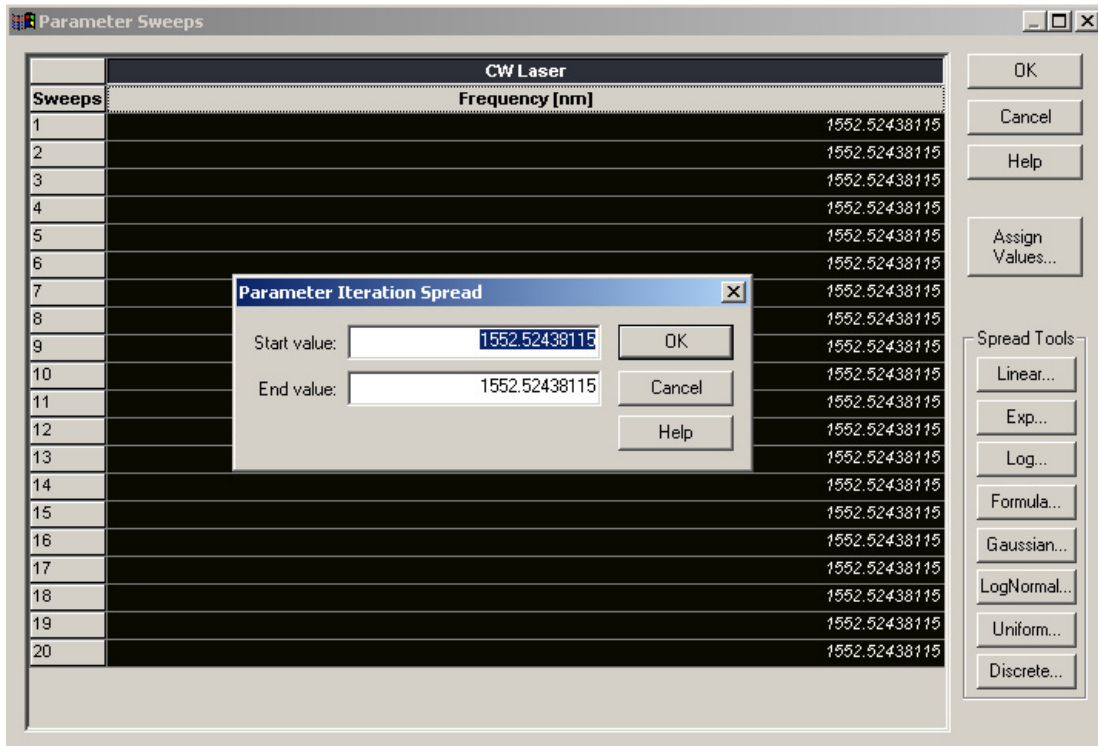
- Choose the number of sweep iterations. In this case we select 20 sweeps. After you click the OK button, the Parameter Sweeps tab will appear.



- Now we can enter the value of each sweep iteration, or we can select all the sweeps and click in one of the buttons (Linear, Exp, Log, Formula, Gaussian, etc.) to spread the values in a range. In our case we clicked in the Linear button. A new



window will open to select the wavelength range in which the signal will be distributed.



- We chose the range from 1530 nm to 1560 nm, and after we clicked on OK, the values were distributed.



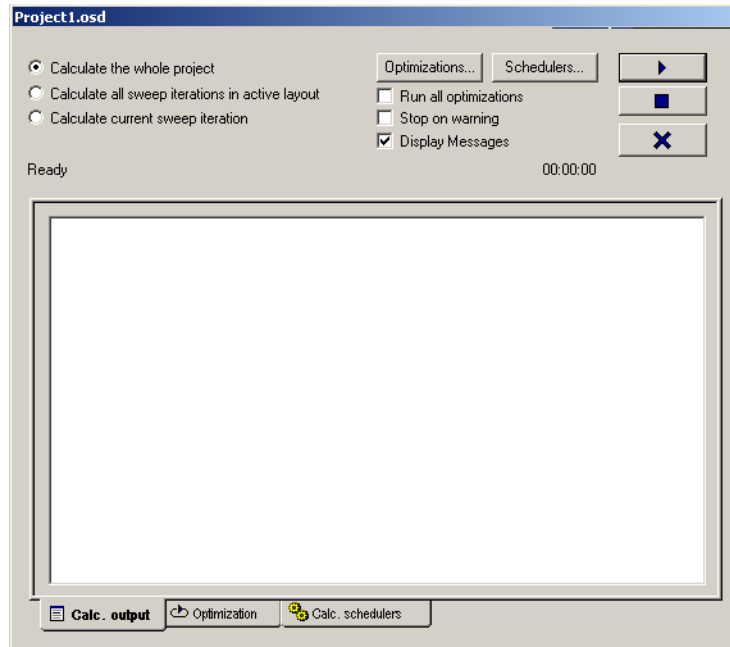


Running the simulation

- To run the simulation, you can go to the File menu and select Calculate. You can also press Control+F5 or use the calculate button in the toolbar. After you select Calculate, the calculation dialog box should appear.
- In this case, we could select to simulate all sweep iterations, or choose a specific iteration and select to calculate the current iteration. For this topic, we selected to calculate all iterations. See Figure 10.
- In the calculation dialog box, press the Play button. The calculation should perform without errors.



Figure 10 Calculating all sweep iterations



Viewing the results

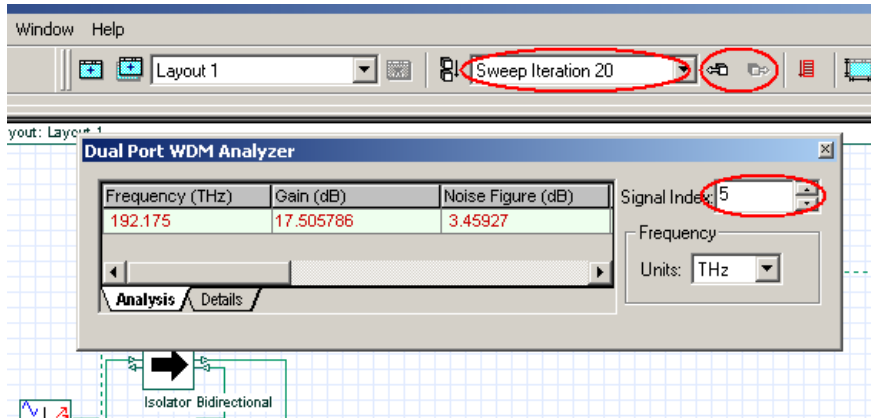
To see the results, double click the Dual Port WDM Analyzer. You can see the results of each iteration (global iteration) in one specific sweep iteration (wavelength) by browsing in the *Signal Index* parameter.

In our case, we kept the Signal Index in the last (global) iteration (5), and we can browse in the results for all (wavelengths) sweep iterations by clicking in the icons "Next Sweep Iteration" and "Previous Sweep Iteration".

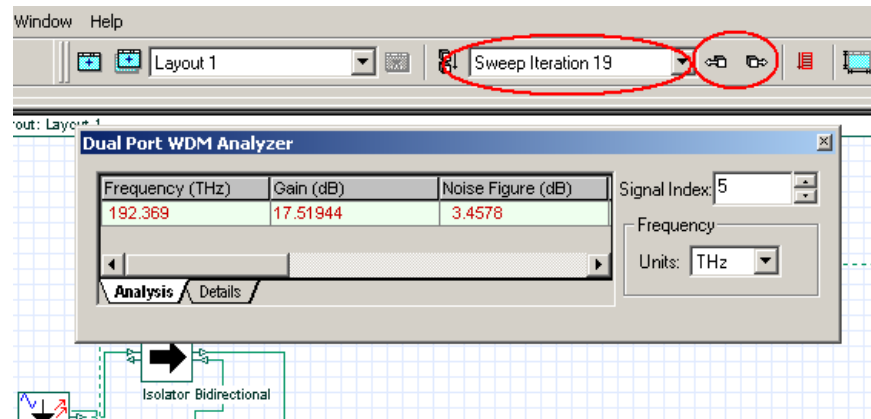
Figure 11 shows the results in the WDM analyzer for the sweep iteration number 20 and 19.



Figure 11 Results for the sweep iteration (a) 20 and (b) 19



(a)



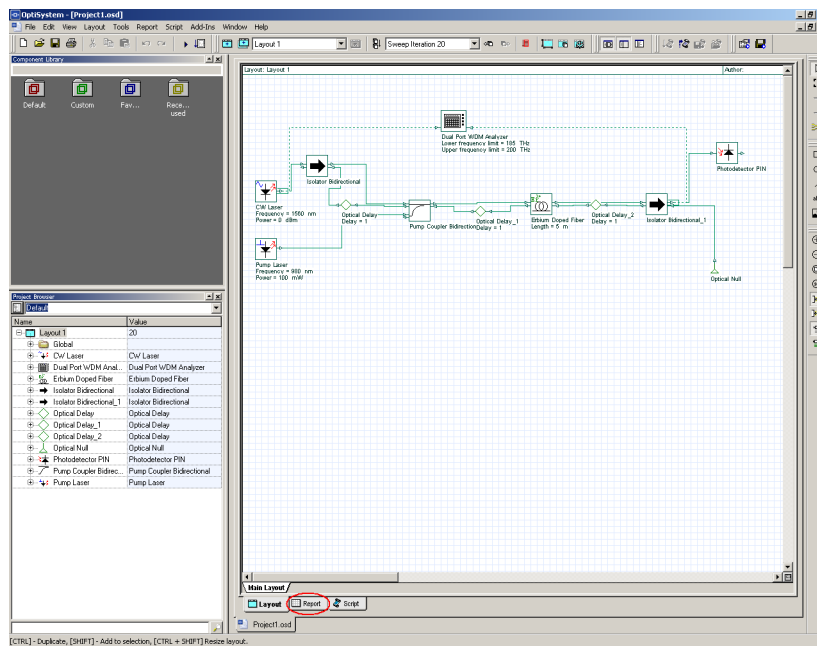
(b)

Building the graph Gain x Wavelength and Noise Figure x Wavelength

- The next step would be to visualize gain and noise figure for all wavelengths simulated. Therefore, we need to build graphs with the results found in the simulation.
- For this task, we will use the Report page.
- Go to the Report tab and click on it. See Figure 12.



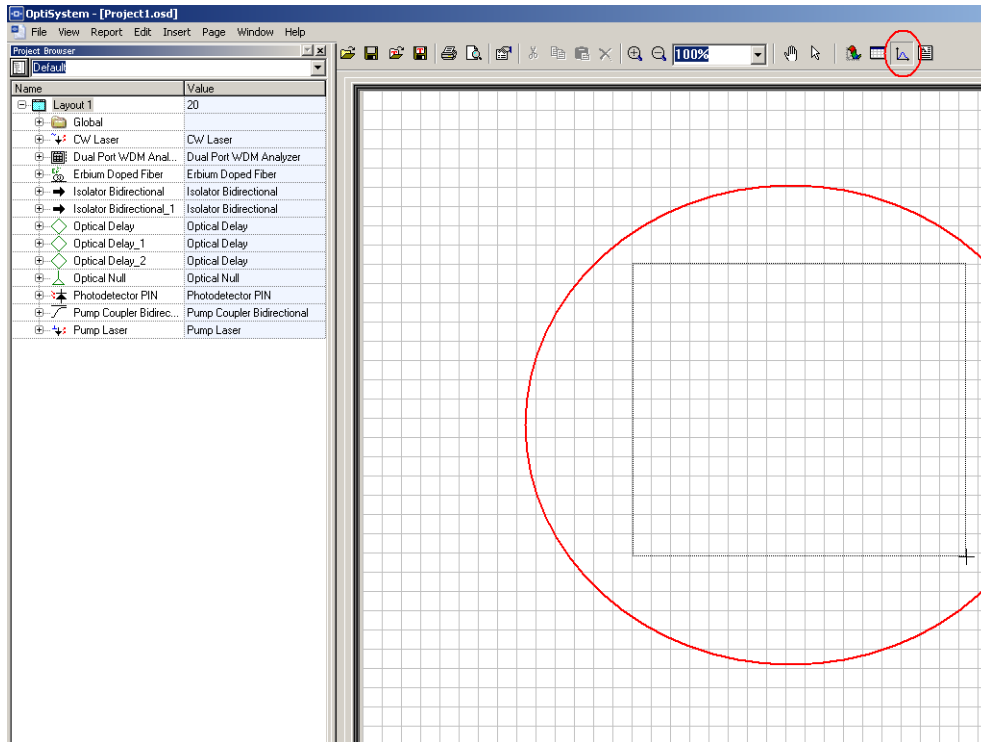
Figure 12 View of the GUI with the system layout



- The report page will appear. After that, select the Opti2Dgraph (graph icon highlighted) and open in the report page a 2D graph. See Figure 13.



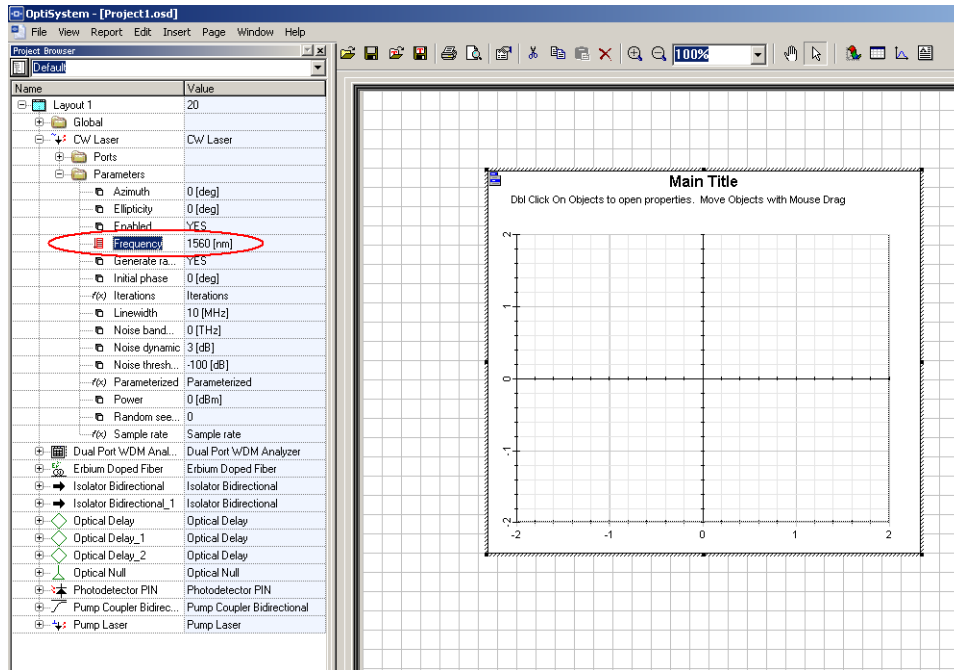
Figure 13 View of the report page



- After opening the 2D graph, go to the project browser window and open the parameter folder of the component that has the information you want. In this case we need the wavelength of the CW Laser, so we selected the *Frequency* parameter. See Figure 14.

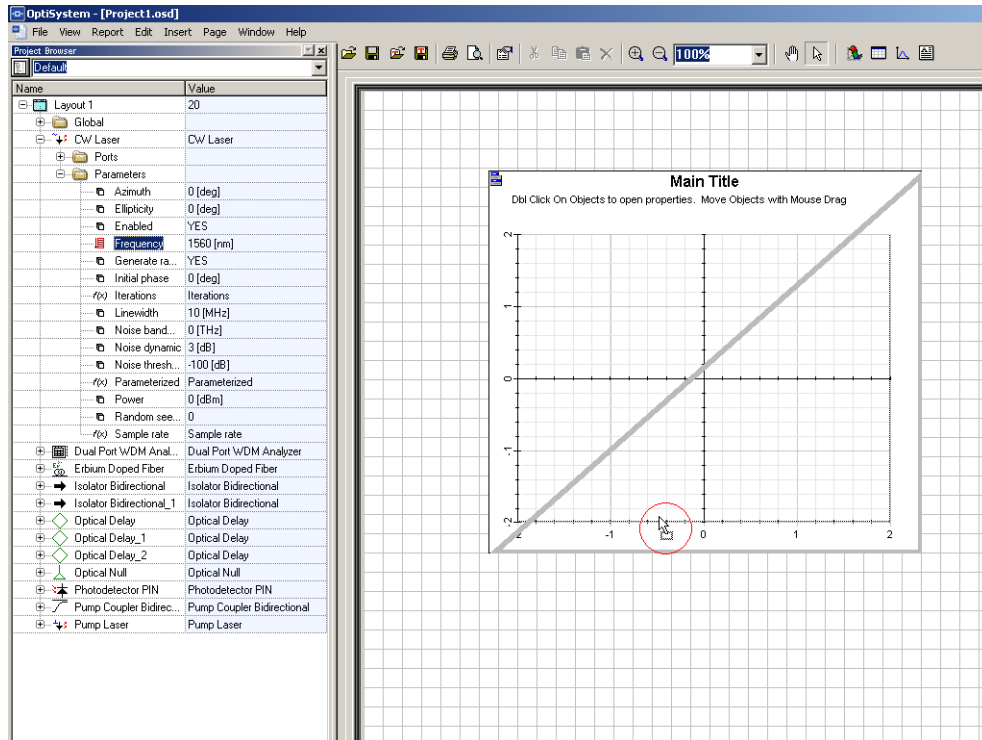


Figure 14 View of the report page with the graph opened and the *Frequency* parameter selected



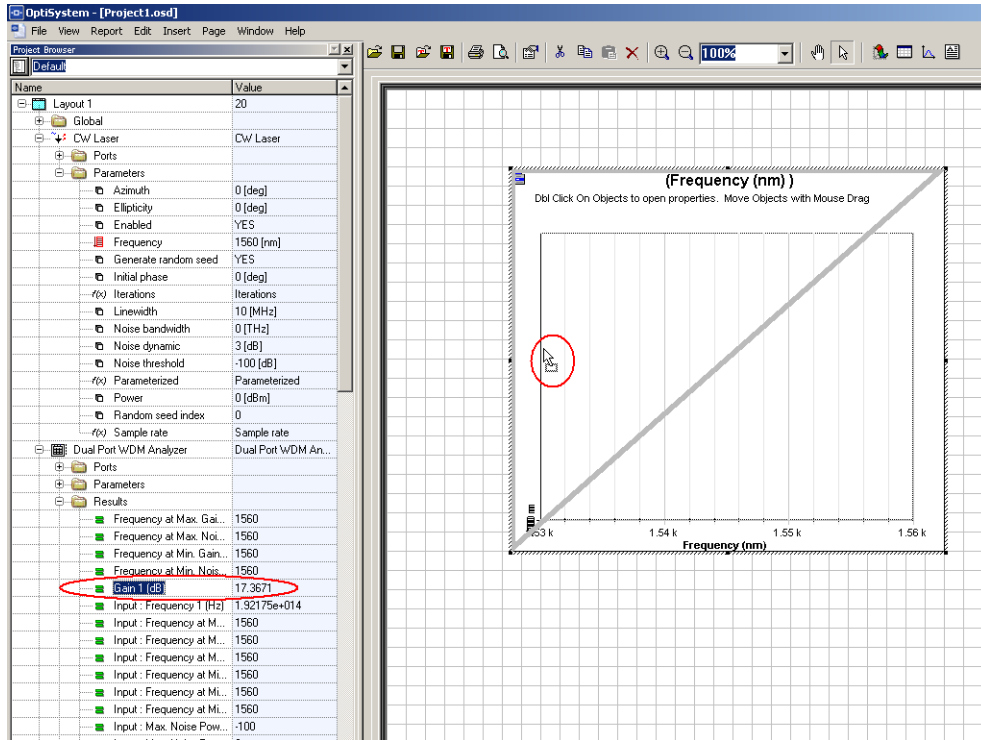
- The Frequency parameter was then dragged to the 2D graph and dropped at the X-axis. See Figure 15.

Figure 15 View of the report page with *Frequency* parameter being dropped at the graph



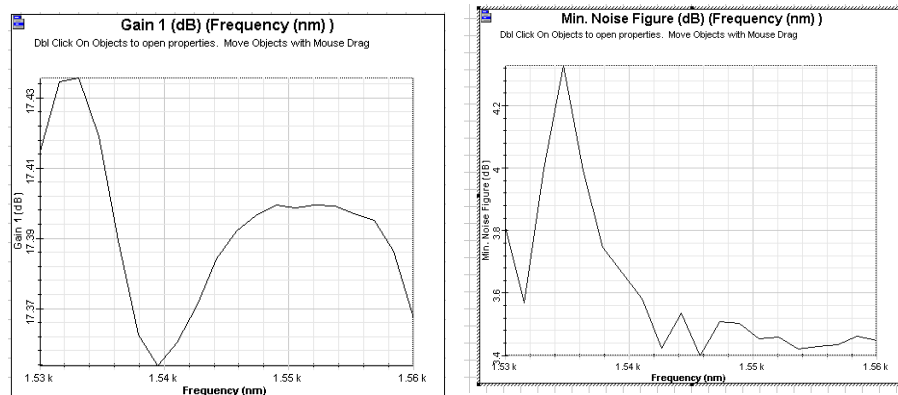
- After dropping the *Frequency* parameter, we have to get the next variable of the graph. So go to the Dual Port WDM Analyzer and open the Results folder.
- Choose one of the results that you want plot in your graph. In this case, we choose the *Gain* parameter. Now, drag and drop it in the y-axis in the graph. See Figure 16.

Figure 16 View of the report page with the Gain result being dropped at the graph



- After that, the graph Gain x Wavelength is ready. See Figure 17(a). The same procedure was used to build the graph NF x Wavelength.

Figure 17 Graphs (a) Gain x Wavelength and (b) NF x Wavelength



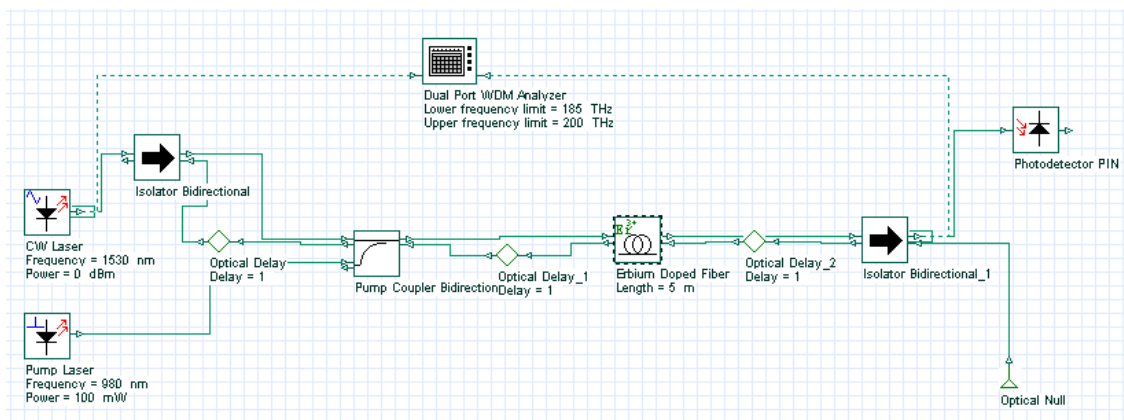
Optimizing the Fiber length

In addition to using the sweep iterations to analyze the amplifier characteristics (such as gain versus signal wavelength and gain versus doped fiber length), the user can use the optimization tool to find the best parameters for the amplifier design.

In this example, we will use the optimization tool to find the best doped fiber length to get the maximum signal gain.

- Load the same layout presented at Figure 5. Remember to return the Total Number of Sweep Iterations to 1 if you had changed it.
- Run the simulation. This step is necessary to setup the Dual Port WDM Analyzer for the optimization.

Figure 18 Layout of the system to be optimized



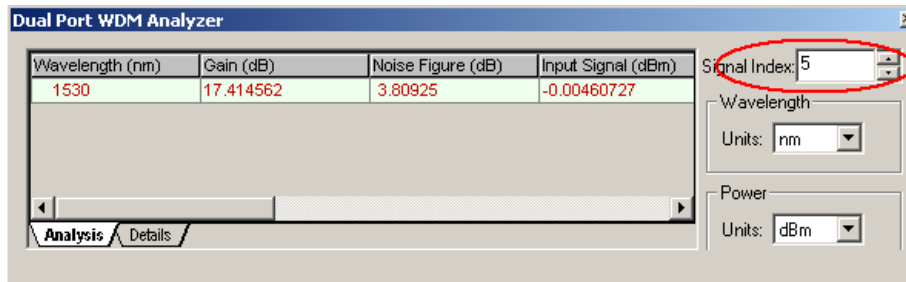
Viewing the results

After you run the system, double click in the Dual Port WDM Analyzer to see the results.

To see the results in different iterations and set the Signal Index to the last iteration, see Figure 19.

To guarantee the optimization will work correctly, you need to set the visualizer Signal Index to the last iteration.

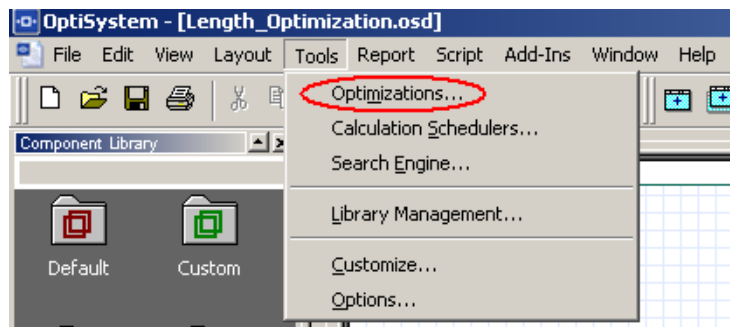
Figure 19 Dual Port WDM Analyzer set with the results of the last iteration



Setting up the Optimization

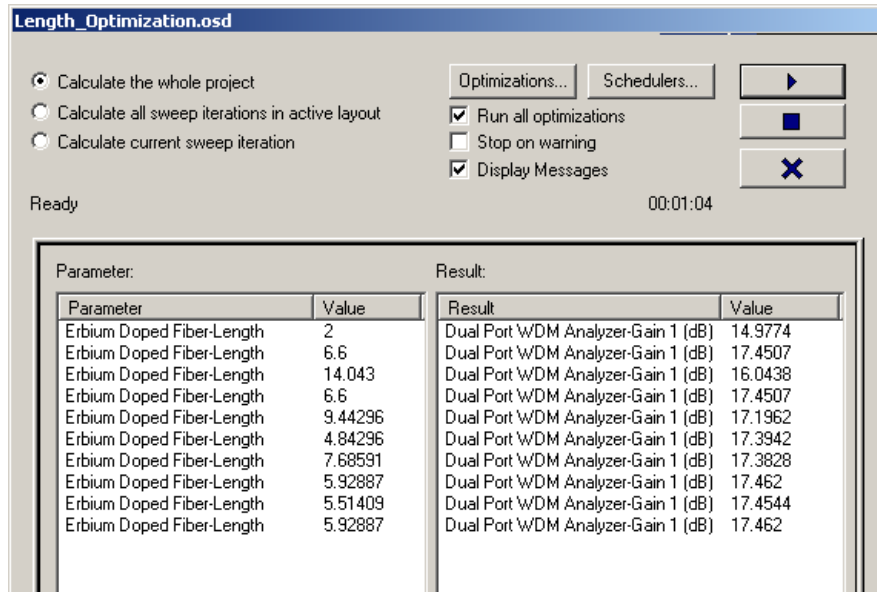
- Go to the tools menu and click in Optimizations. See Figure 20.

Figure 20 Optimizations option at the Tools menu



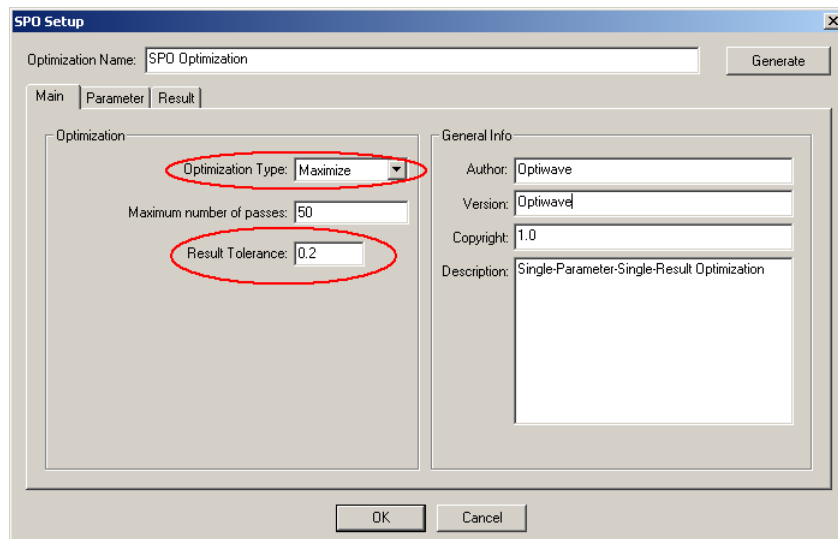
- The Optimizations window will appear.
- There are two options for the optimization: the Single Parameter Optimization (SPO) and the Multiple Parameter Optimization (MPO). In our case, we want only to optimize the Fiber length as a function of the gain, so we can select the SPO and insert it on the project. For example, Figure 21 displays the calculation window and the optimization tab.

Figure 21 Optimizations tab during calculation



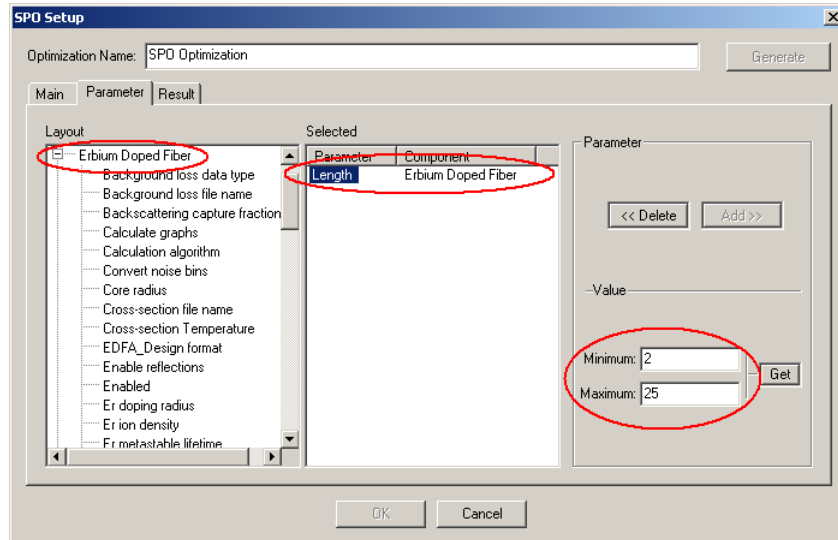
- When we insert the SPO optimization, the SPO Setup window will open. See Figure 22.
- In the Main tab, select the Optimization Type you want. In this case we selected to maximize the result.
- Setup the number of passes and the result tolerance.

Figure 22 SPO Setup window at the Main tab



- Go to the Parameter tab in the SPO Setup and select the parameter that you want to optimize. In this case, Length at the Erbium-doped fiber component. See Figure 23.
- After you select the parameter, specify the range of values you want to analyze. Be careful in choosing these values, they have to match the unit chosen in the *Component* parameter. For this case there is only one unit, meter. But in some cases, such as for power, the unit could be W, mW or dBm.

Figure 23 SPO Setup window at the Parameter tab



- In the Result tab, select the result in the Dual Port WDM Analyzer that you want to maximize (Gain). See Figure 24.
- Because we want to maximize the gain, the Target value will not be enabled.
- Click on ok in the SPO Setup.
- To run the simulation you can go to the File menu and select Calculate. See Figure 25.
- Before you press the Play button, click in the box to run all optimizations. Then, press the Play button.



Figure 24 SPO Setup window at the Result tab

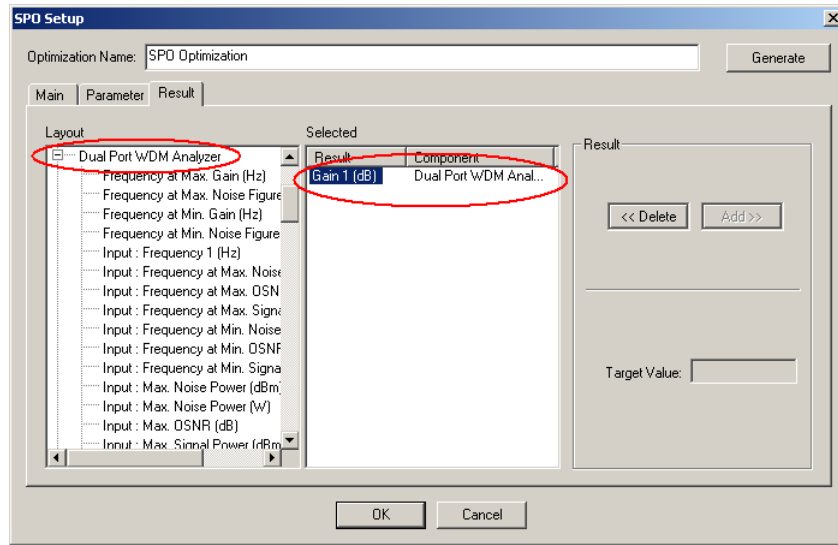
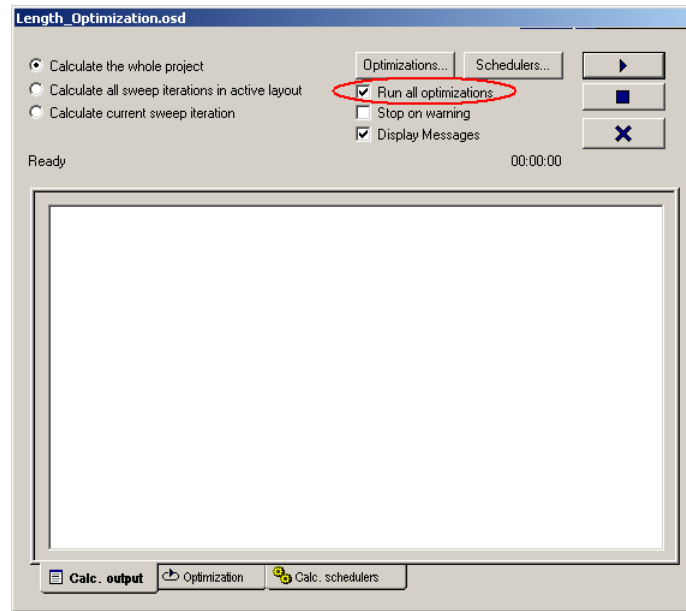


Figure 25 Calculate window

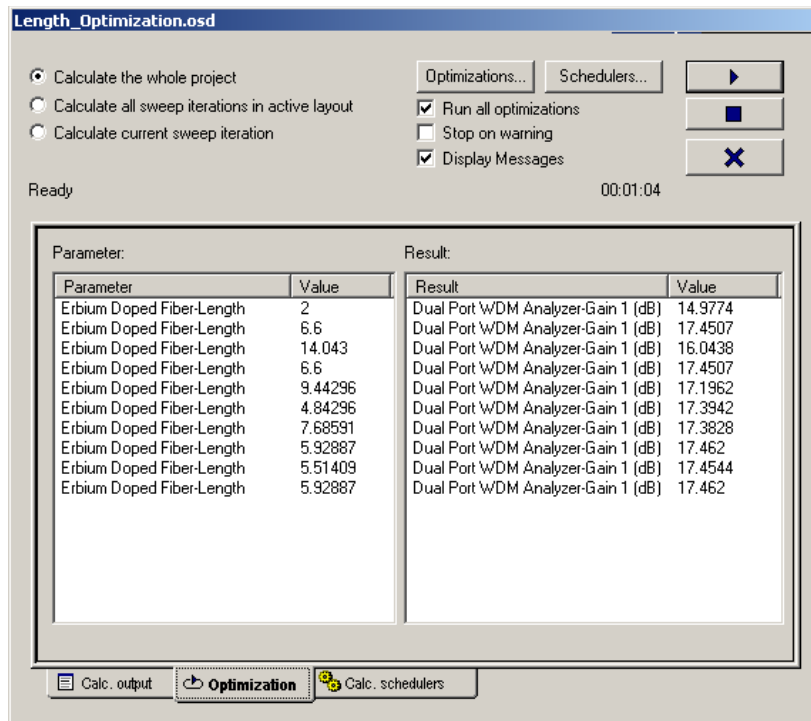


During the calculation, the parameters and results of each pass will appear in the Optimization Tab. See Figure 26(a).

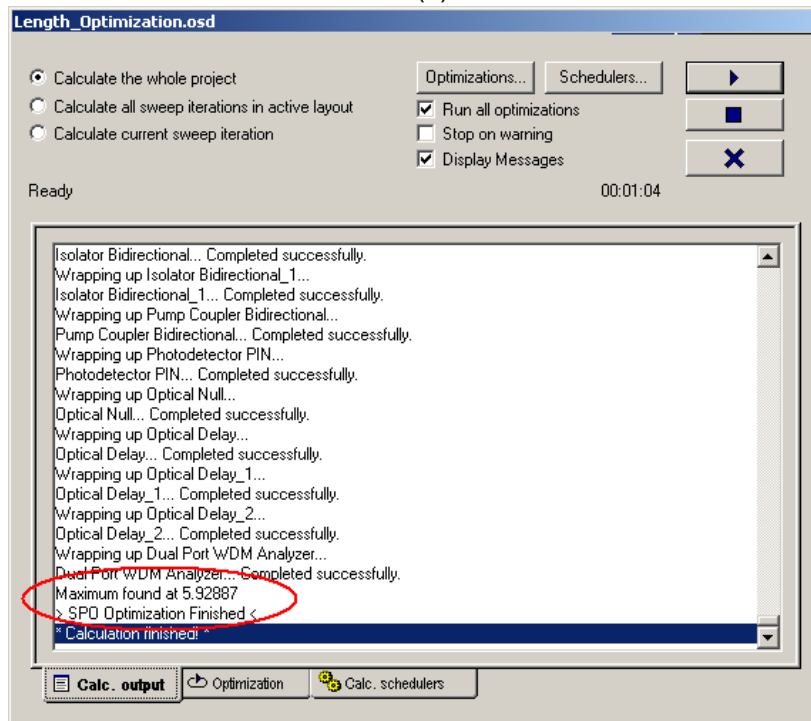
In the calculation output tab will appear the optimum fiber length where we got the maximum gain for the signal at 1530 nm.



Figure 26 (a) Optimization tab and (b) Calc. Output tab



(a)



(b)



Lesson 8: Optical Systems — Working with multimode components

The multimode component library of OptiSystem allows for simulation of links with multimode signals.

The projects presented here are available under OptiSystem installation folder Samples/Multimode.






The first part of this tutorial will describe the library of spatial optical sources and transmitters. These components have the ability to include transverse mode profiles in the optical signals.




There are two categories of components in the library:

- spatial optical sources/transmitters
- transverse mode generators

The spatial transmitters are “ready-to-use” components that encapsulate an optical source or transmitter and a transverse mode generator.






Table 1 Spatial optical sources, transmitters and pulse generators

Component name	Folder	Icon
Spatial CW Laser	Transmitters Library/Optical Sources/	
Spatial Laser Rate Equations	Transmitters Library/Optical Sources/	
Spatial LED	Transmitters Library/Optical Sources/	
Spatial Optical Gaussian Pulse Generator	Transmitters Library/Pulse Generators/Optical/	
Spatial Optical Impulse Generator	Transmitters Library/Pulse Generators/Optical/	

Component name	Folder	Icon
Spatial Optical Sech Pulse Generator	Transmitters Library/Pulse Generators/Optical/	
Spatial Optical Transmitter	Transmitters Library/Optical Transmitters/	
Spatial VCSEL	Transmitters Library/Optical Sources/	

The second category is the transverse mode generators. These components allow for adding a transverse mode profile in the optical signal.

Table 2 Transverse mode generators

Component name	Folder	Icon
Multimode Generator	Transmitters Library/Multimode/	
Laguerre Transverse Mode Generator	Transmitters Library/Multimode/	
Hermite Transverse Mode Generator	Transmitters Library/Multimode/	
Donut Transverse Mode Generator	Transmitters Library/Multimode/	
Measured Transverse Mode Generator	Transmitters Library/Multimode/	

This means you can directly use one of the components from Table 1 or you can combine any optical source, transmitter or pulse generators with one of the transverse mode generators from Table 2 and build a new subsystem.

The second part of the tutorial presents the passive components that allow for translation and rotation of the optical field, as well as focusing an optical beam or applying an aperture in order to limit the optical beam.

Finally, a description of the receiver and photodetector components is presented.



Multimode Transmitters

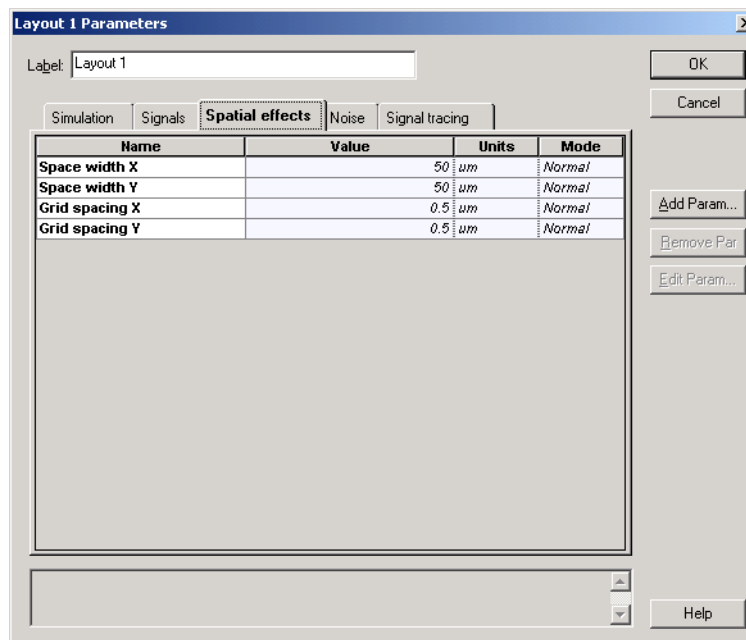
Global parameters

The first step when using OptiSystem is setting the global parameters.

As we already know, one of the main parameters is the time window, which is calculated from the bit rate and sequence length.

For multimode simulations, there are additional parameters that will define the space window for the transverse mode profiles. These parameters are the *Space width X* and *Y* and the *Grid spacing X* and *Y*, and are available in the global parameters window (Figure 1).

Figure 1 Global parameters: Spatial effects tab

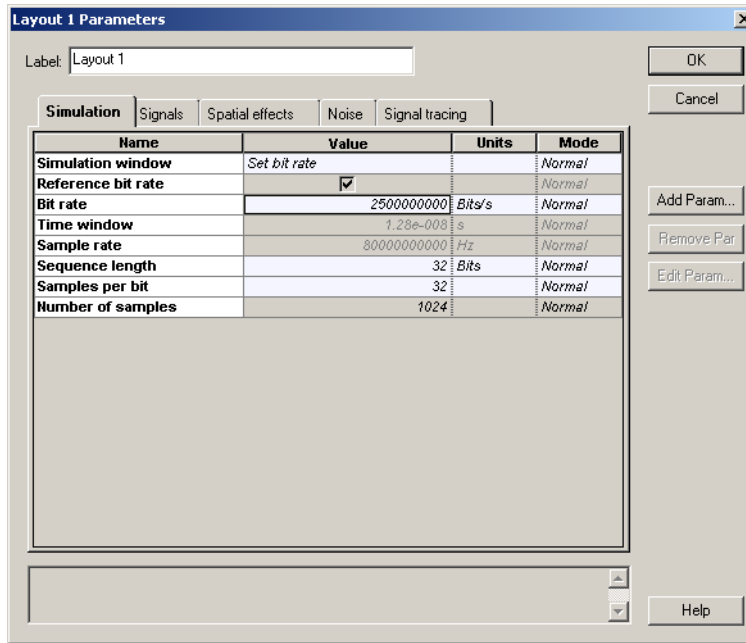


For this tutorial we will use the default parameters, except for the bit rate, that should be changed to 10 GB/s

- In the global parameters dialog box, change the parameter *Bit rate* to 10e9. The *Time window* parameter should be 1.28e-8 s (Figure 2).



Figure 2 Global parameters: Simulation parameters tab



System setup

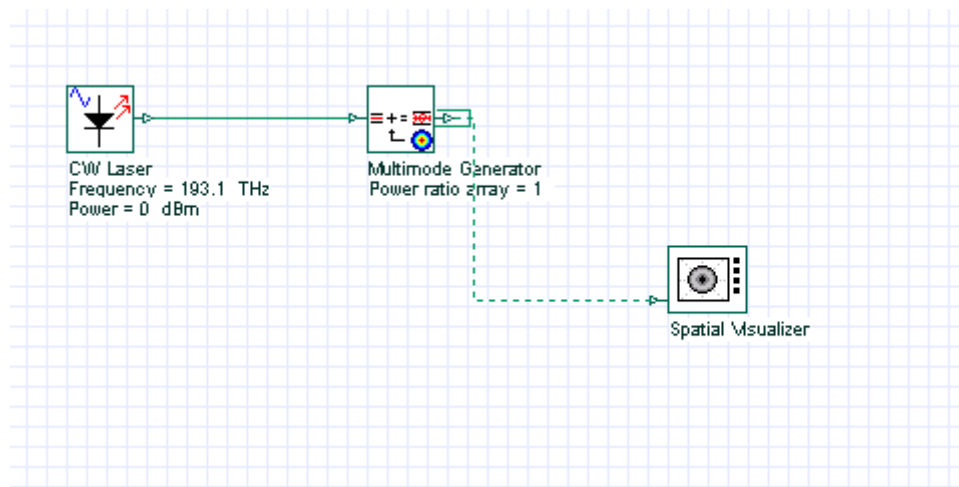
After setting the global parameters, we can start adding the components to design the basic multimode transmitter.

From the component library drag and drop the following component in to the layout:

- From "Default/Transmitters Library/Optical Sources", drag and drop the "CW Laser" into the layout.
- From "Default/Transmitters Library/Multimode", drag and drop the "Multimode Generator" component into the layout.
- From "Default/Visualizers Library/Optical", drag and drop the "Spatial Visualizer" into the layout.

The next step is to connect the components according to Figure 3.

Figure 3 Multimode transmitter



Spatial effects tab

The spatial effects tab is available for all the spatial optical sources, transmitters and pulse generators, as well as for the transverse mode generator components.

One of the main parameters of the spatial effects tab is the *Power ratio array*. This array is a string list with the power ratios for each mode.

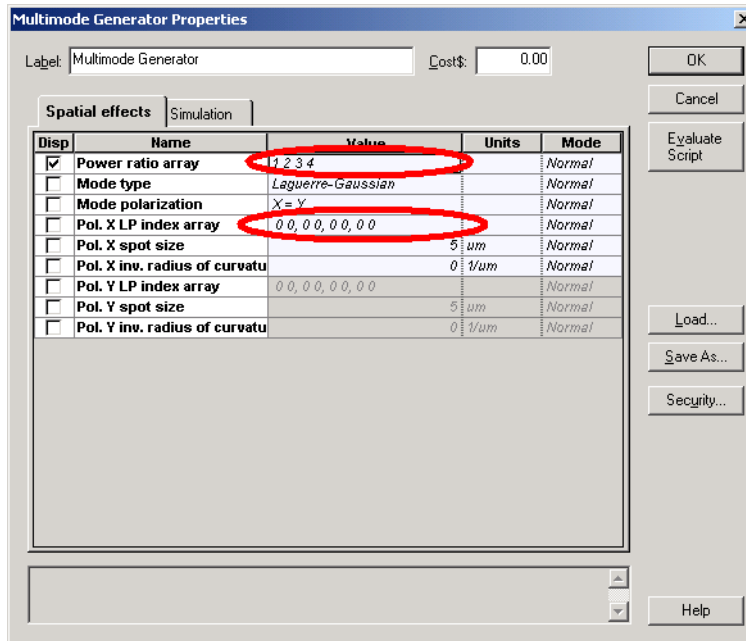
For example, entering a list of 4 values will allow the user to also specify the properties of a particular mode type for each of these values.

- Open the parameters of the "Multimode Generator" component.
- In the spatial effects tab, for the *Power ratio array* parameter, enter the following values: 1 2 3 4.

Notice that the parameter *Pol. X m,n index array* will automatically increase the number of values for the LP index pairs for the generated modes (Figure 4).



Figure 4 Multimode Generator - changing the number of power ratios



- In the spatial effects tab, for the *Pol. X m,n index array* parameter, enter the following values: 0 0, 2 2, 0 3, 1 3.

This means the multimode generator will generate 4 spatial modes per polarization.

The power ratio per mode will be 0.1, 0.2, 0.3 and 0.4. The total is equal to 1 (100% of the input power).

Each mode will have its own Laguerre-Gaussian profiles, with LP indexes LG_{00} , LG_{22} , LG_{03} , LG_{13} . Both polarizations will have the same spatial modes (*Mode polarization* parameter is equal to $X=Y$).

Running the simulation

We can run this simulation and analyze the results:

- To run the simulation, you can go to the File menu and select Calculate. You can also press Control+F5 or use the calculate button in the toolbar. After you select Calculate, the calculation dialog box should appear.
- In the calculation dialog box, press the Play button. The calculation should perform without errors.

Viewing the results

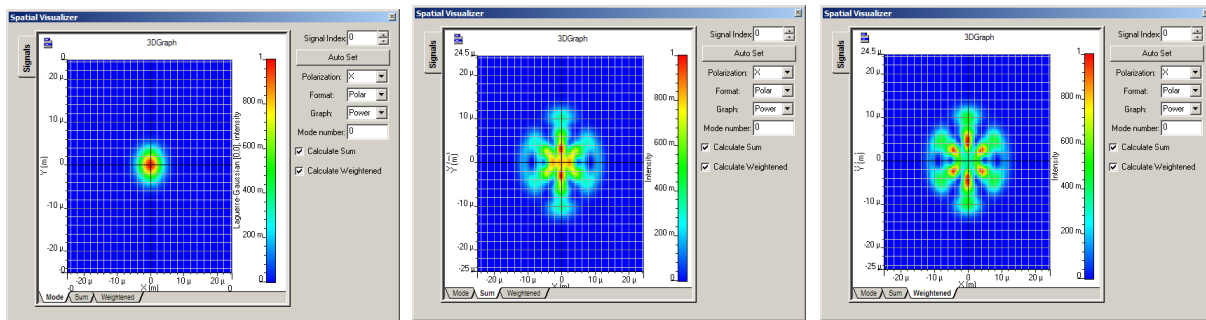
In order to see the results, double click the Spatial visualizer.

Using the spatial visualizer, you can see the profile of each mode presented at the visualizer input port. The user can select the polarization (X or Y), the format (Polar or Rectangular) and the graph (Power, Phase, Real or Imaginary part).

Additionally, the sum of the modes can be displayed, as well as the weighted sum (Figure 5).

Observe that the spatial dimensions of the graphs are the same as defined by the global parameters *Space width X* and *Y*.

Figure 5 Spatial visualizer can display the Individual, Sum and Weighted sum of modes

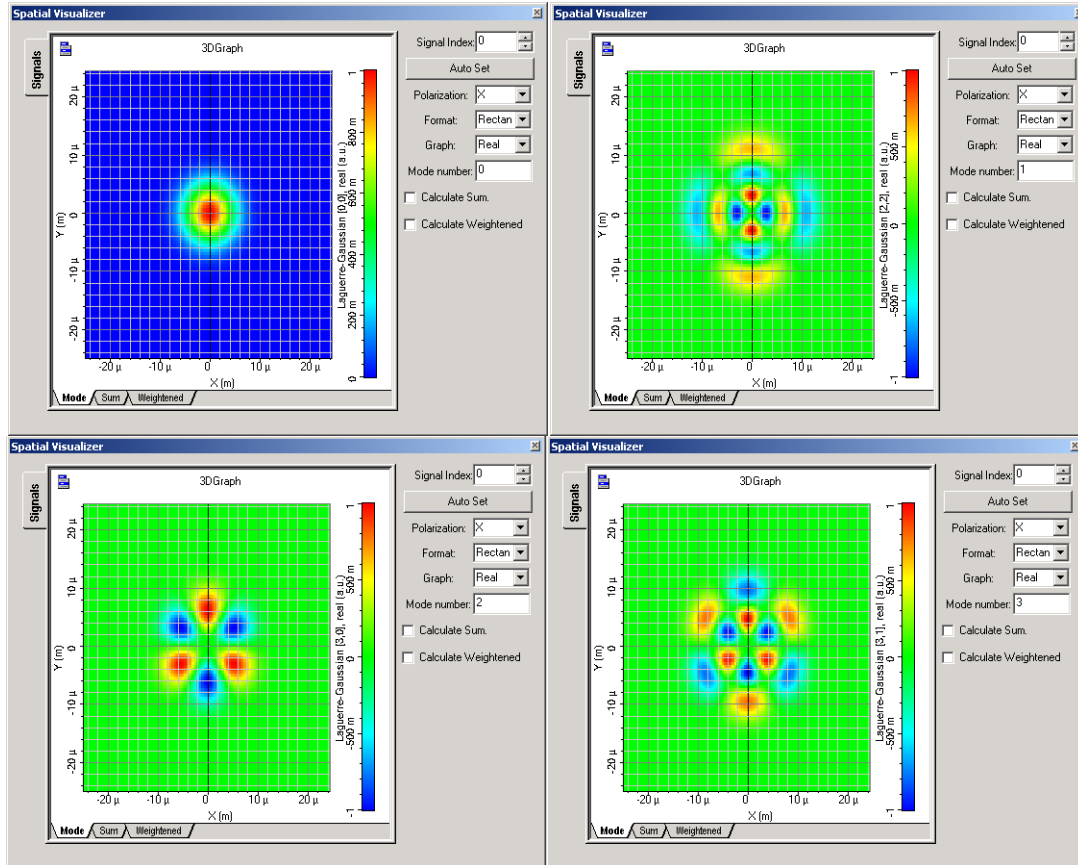


In order to browse through the modes that are available at the input port, change the mode number parameter and press <Enter>. The visualizer will recalculate the graphs and present the individual mode at the parameter mode index.



Figure 6 presents the real values using rectangular format for each available mode at the input port.

Figure 6 Spatial visualizer displays the real part of the individual modes for each mode index



The combination of the CW Laser and the Multimode Generator component can be replaced by the Spatial CW Laser component:

- From "Default/Transmitters Library/Optical Sources", drag and drop the "Spatial CW Laser" into the layout.
- In the Spatial CW Laser parameters, use the same parameter values of the spatial effects tab from the Multimode Generator

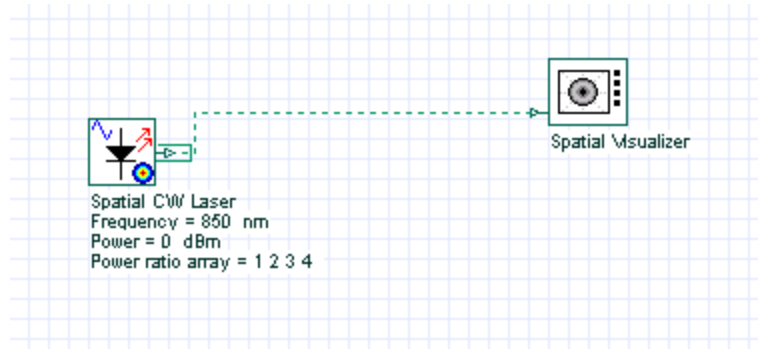
The next step is to connect the components according to the Figure 7.

You can remove the multimode generator and connect the spatial visualizer to the Spatial CW Laser.

Run the simulation again.



Figure 7 Multimode transmitter using a Spatial CW Laser



This system, built using a component from Table 1, is equivalent to the one in Figure 3, which was built using the components from Table 2.

The results from the spatial visualizer will be the same as presented in Figure 5 and Figure 6.

Multimode components and Free-Space Propagation

When coupling a signal from a laser into a multimode fiber, or from the fiber into a photodetector, we can use the components from Table 3.

Table 3 Spatial passive components: connector, aperture and lens

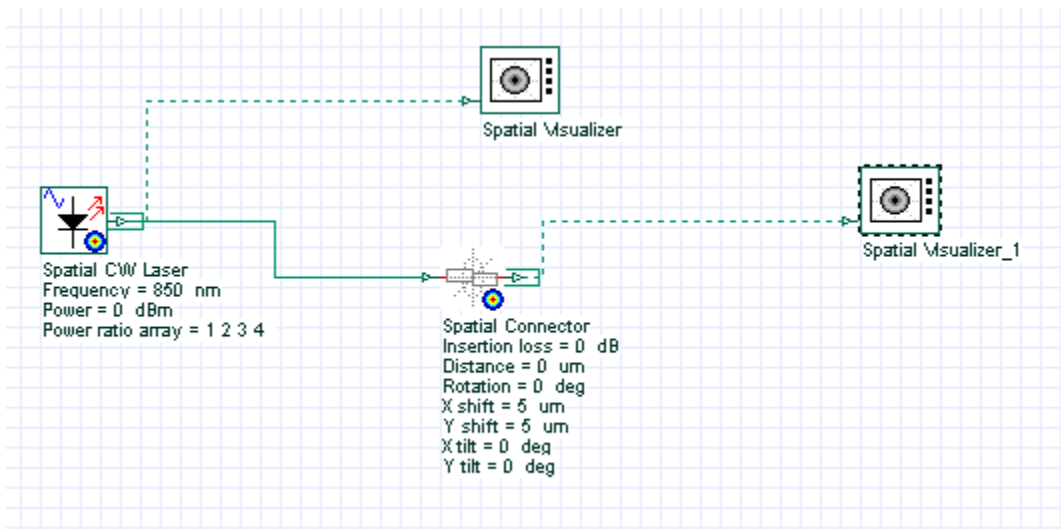
Component name	Folder	Icon
Spatial Connector	Passives Library/Optical/Connectors/	
Spatial Aperture	Passives Library/Optical/Multimode/	
Thin Lens	Passives Library/Optical/Multimode/	
Vortex Lens	Passives Library/Optical/Multimode/	

Spatial connector: translating the optical field

One of the main components is the spatial connector. This components can rotate and translate the optical field, as well as propagate the optical signal in free-space.

- From "Default/Passives Library/Optical/Connectors", drag and drop the "Spatial Connector" into the layout.
- For the Spatial Connector, change the parameters *X* and *Y shift* to 5 μm .
- Add another Spatial Visualize to the layout.
- Connect the components according to Figure 8.
- Run the simulation.

Figure 8 Multimode transmitter and spatial connector

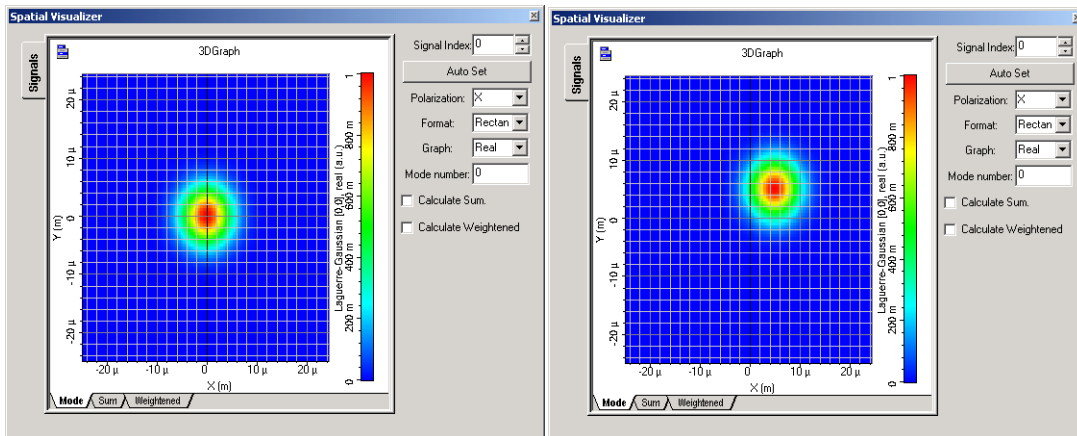


Viewing the results

In order to see the results, double click in the second spatial visualizers and compare the graphs before and after the connector.

The connector applied a spatial shift in both X and Y-axis (Figure 9).

Figure 9 Spatial visualizer displays the transverse mode before and after the connector

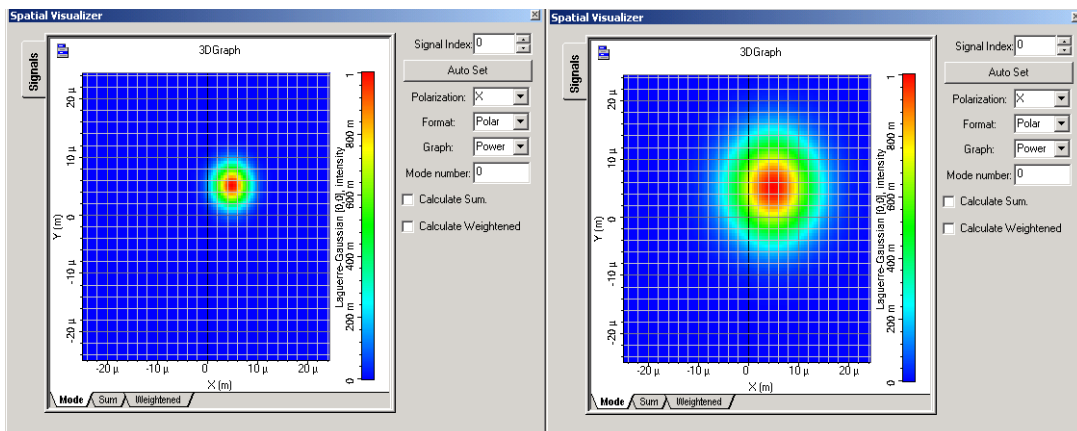


In order to see the effects of free-space propagation in the optical field, we can increase the connector parameter *Distance*:

- For the Spatial Connector, change the parameter *Distance* to $200\ \mu\text{m}$.
- For the Spatial Visualizers, change the parameters *Format* to Polar.
- Run the simulation.

Figure 10 compares the transverse mode intensity before and after applying $200\ \mu\text{m}$ of propagation in free-space, you can see that the second beam is larger.

Figure 10 Spatial visualizer displays the transverse mode intensity after a shift of $5\ \mu\text{m}$ in both X and Y directions, before and after $200\ \mu\text{m}$ propagation in free-space



Spatial connector: rotating the optical field

Besides the translation the optical field and the propagation in free-space, the user can rotate the optical field by applying a tilt in the X and Y-axis, or a rotation about the Z-axis.

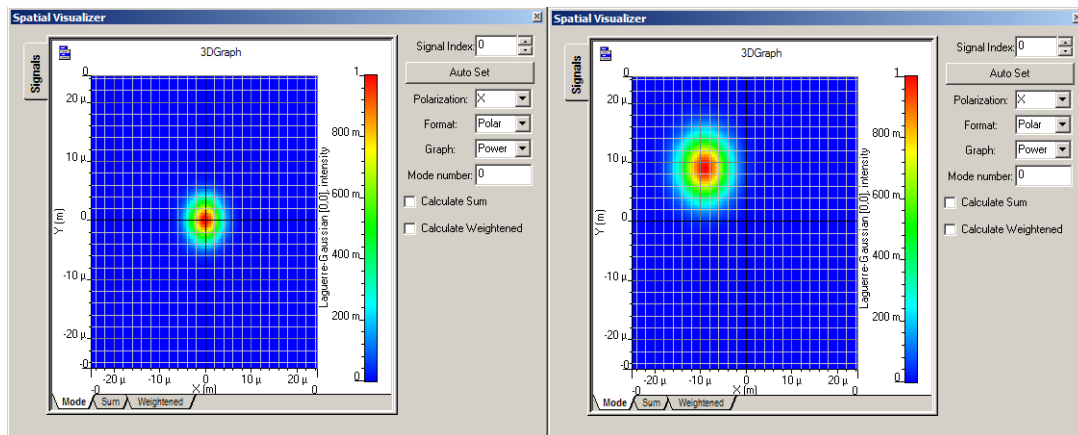
- For the Spatial Connector, change the parameters *X* and *Y shift* to $0 \mu m$, and the *Distance* to $100 \mu m$.
- For the Spatial Connector, change the parameters *X* and *Y tilt* to 5 deg.
- Run the simulation.

Viewing the results

In order to see the results, double click in the both spatial visualizers and compare the graphs before and after the connector.

The connector applied a spatial tilt in both X and Y-axis, visible after the free-space propagation (Figure 10).

Figure 11 Spatial visualizer displays the transverse mode before and after the 5 deg tilt and $100 \mu m$ propagation

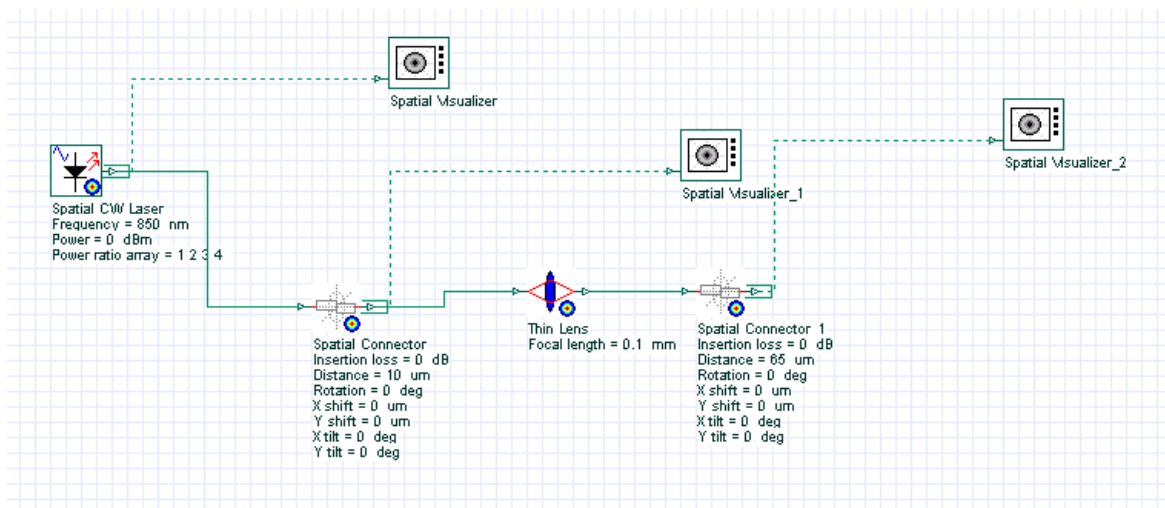


Thin Lens: focusing the optical beam

By using the thin lens component the user can focus or defocus the optical beam.

- From "Default/Passives Library/Optical/Multimode", drag and drop the "Thin Lens" component into the layout.
- Add another "Spatial Connector" and "Spatial Visualizer" to the layout.
- For the "Spatial CW Laser", set the parameter *Pol. X spot size* to $15\ \mu\text{m}$.
- For the first spatial connector, set the shift and tilt parameters to 0, and the parameter *Distance* to $10\ \mu\text{m}$
- For the second spatial connector, set the shift and tilt parameters to 0, and the parameter *Distance* to $65\ \mu\text{m}$.
- For the thin lens, the parameter *Focal length* to $0.1\ \text{mm}$.
- Connect the components according to Figure 12.
- Run the simulation.

Figure 12 Using the thin lens component to focus the optical beam



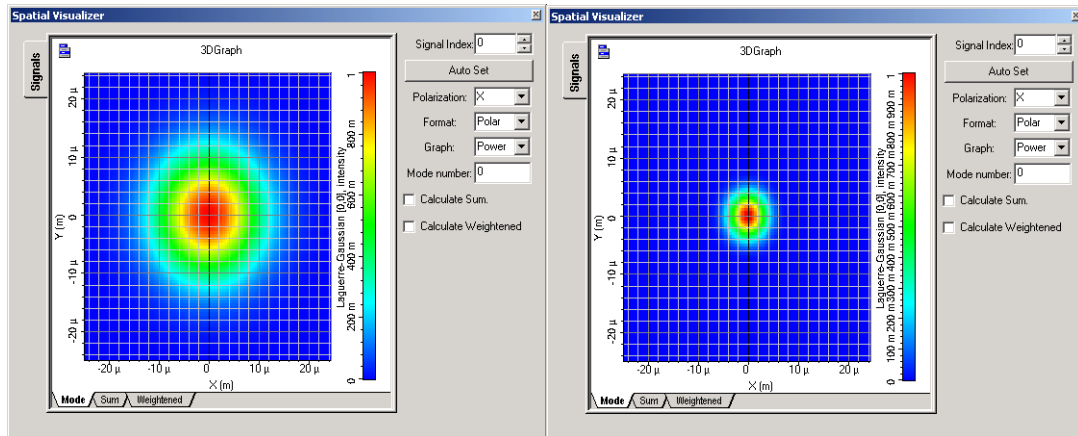
Viewing the results

In order to see the results, double click in the spatial visualizers and compare the graphs before and after the second connector. The large beam after the first connector is reduced by the combination of thin lens and free-space propagation.

Figure 13 presents the optical field at the output of the first connector and at the output of the second connector.



Figure 13 Spatial visualizer displays the transverse mode before and after focusing the beam using the thin lens

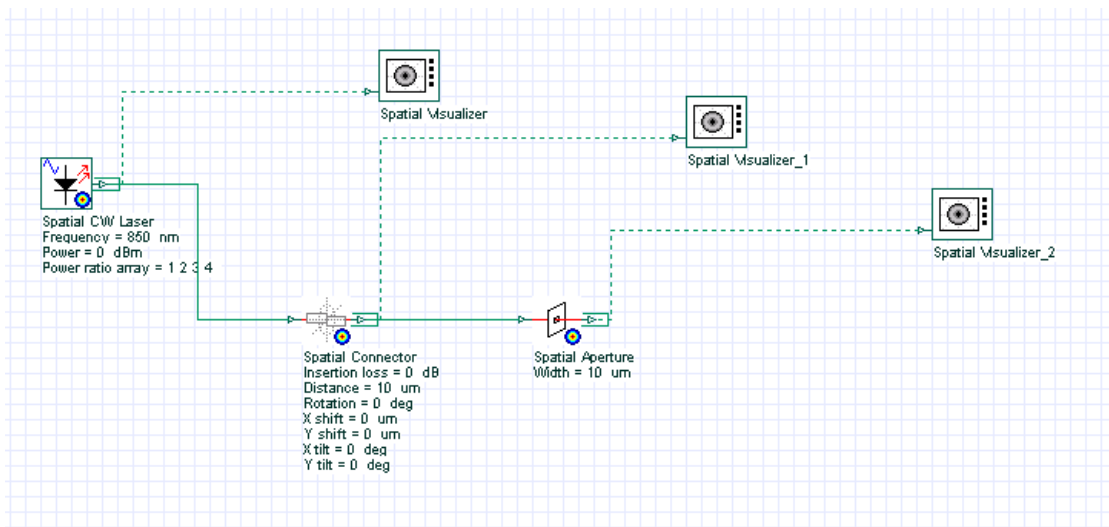


Spatial Aperture: limiting the optical field

By using the spatial aperture component the user can reduce the optical field by filtering the optical power that falls outside the aperture area.

- From "Default/Passives Library/Optical/Multimode", drag and drop the "Spatial Aperture" into the layout.
- Remove the thin lens and the second connector from the layout.
- Connect the components according to Figure 14.
- Run the simulation.

Figure 14 Using the spatial aperture component to limit the optical beam



Viewing the results

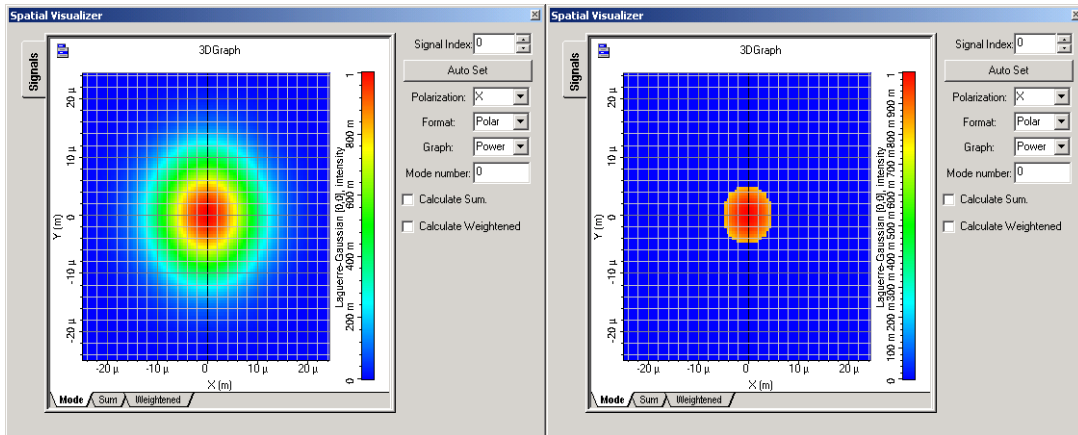


In order to see the results, double click in the spatial visualizers and compare the graphs before and after the spatial aperture.

The output beam is limited to the aperture diameter of $10\ \mu\text{m}$ (Figure 15).

You can use the spatial aperture to simulate component misalignments, e.g. fiber/detector power coupling.

Figure 15 Spatial visualizer displays the transverse mode before and after the spatial aperture component

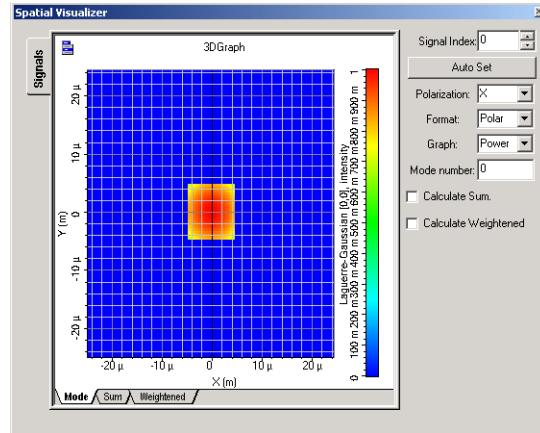


The user can also select a rectangular square type for the spatial aperture:

- For the spatial aperture, set the parameter Aperture type to “Square”.
- Run the simulation.

Observe that the optical beam will have a square shape after the spatial aperture (Figure 16).




Figure 16 Spatial visualizer displays the transverse mode after the square spatial aperture component



Multimode Receivers

Similar to the multimode transmitter, where the user can select a ready-to-use spatial component from the library, or build new spatial components by combining the spatial and no-spatial components, OptiSystem has also different multimode receiver components (Table 4). These components are a combination of the spatial aperture component and the optical receivers or photodetectors.

Table 4 Spatial receivers and photodetectors

Component name	Folder	Icon
Spatial Optical Receiver	Receivers Library/Optical Receivers/	
Spatial PIN Photodetector	Receivers Library/Photodetectors/	
Spatial APD Photodetector	Receivers Library/Photodetectors/	

The main parameter of the spatial receivers or photodetectors is related to the detector active area, defined by the parameter *Width*.

System setup

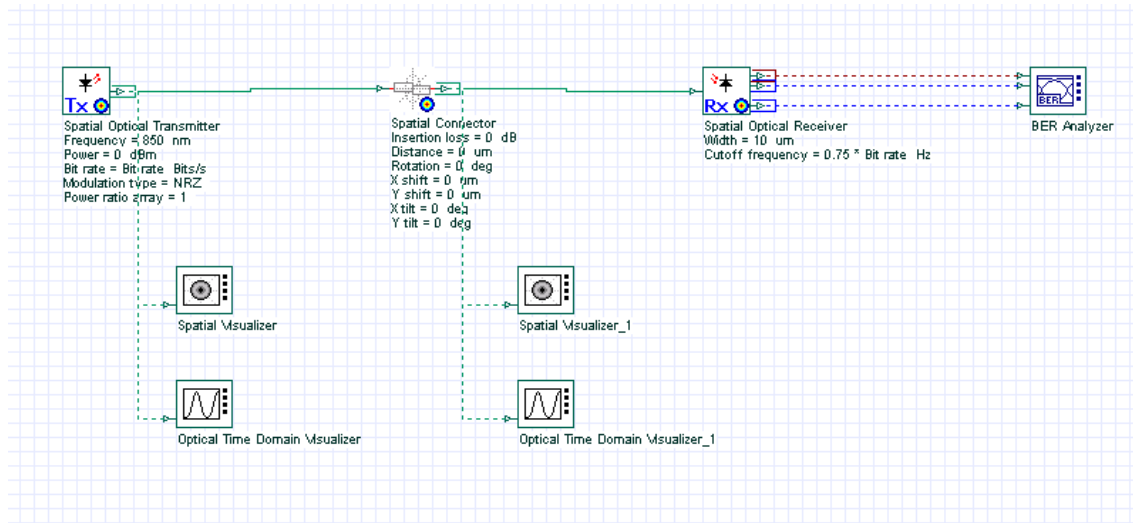
Using the same previous global parameters, we can start adding the components to design the basic multimode link. From the component library drag and drop the following component in to the layout:

- From "Default/Transmitters Library/Optical Transmitters", drag and drop the "Spatial Optical Transmitter" into the layout.
- For the Transmitter, change the parameter Pol. X. spot size (Spatial effects tab) to $15 \mu m$.
- From "Default/Receivers Library/Optical Receivers", drag and drop the "Spatial Optical Receiver" component into the layout.
- For the Receiver, change the parameter Thermal noise (Noise tab) to $5e-22$ W/Hz. This will give a receiver sensitivity of ~ -17 dBm (10 GB/s, Q = 6).
- From "Default/Visualizers Library/Electrical", drag and drop the "BER Analyzer" into the layout.
- From "Default/Visualizers Library/Optical", drag and drop two of the "Optical Time Domain Visualizer" into the layout.
- Set all the parameters of the spatial connector to 0.

Using the connector and visualizer from the previous layout, the next step is to connect the components according to the Figure 17.



Figure 17 A multimode link using the spatial transmitter and receiver



Run the simulation and after the simulation is finished observe the results from the BER Analyzer.

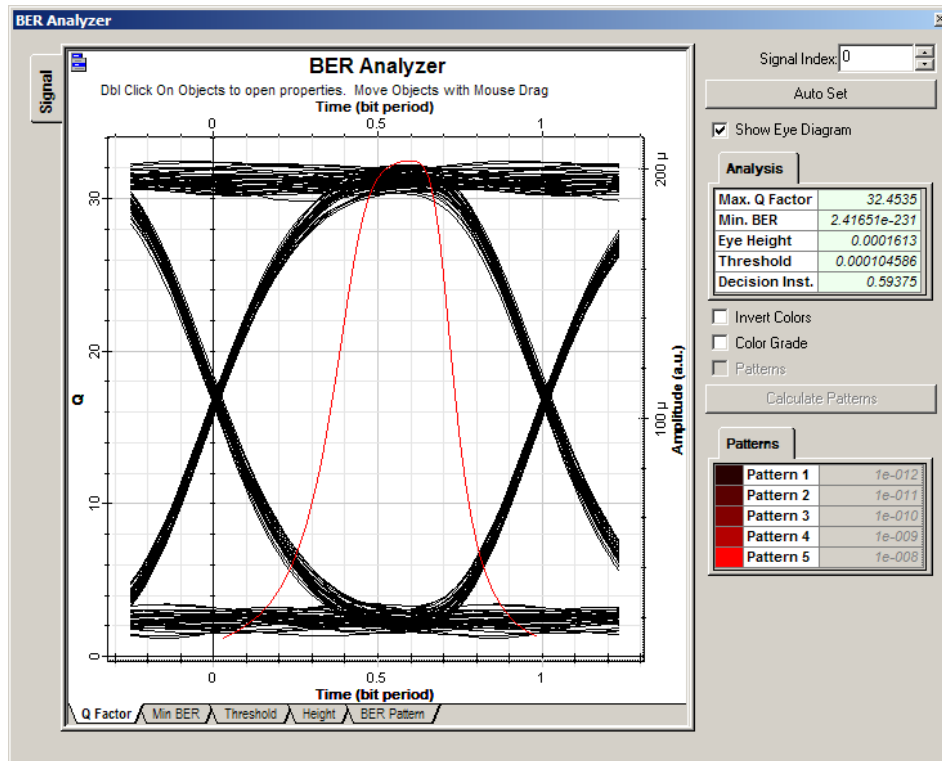
Viewing the results

The link of Figure 17 is almost ideal, because we do not have any major losses or misalignments between the transmitter and receiver. The spatial connector is ideal and there is only a small loss due to the difference between the input signal spot size and active area of the photodetector ($10\ \mu\text{m}$ and the mode spot size of $15\ \mu\text{m}$).

Figure 18 presents the eye diagram for the system of Figure 17, observe the back-to-back value of the Q factor.



Figure 18 BER Analyzer displaying the eye diagram of the ideal link



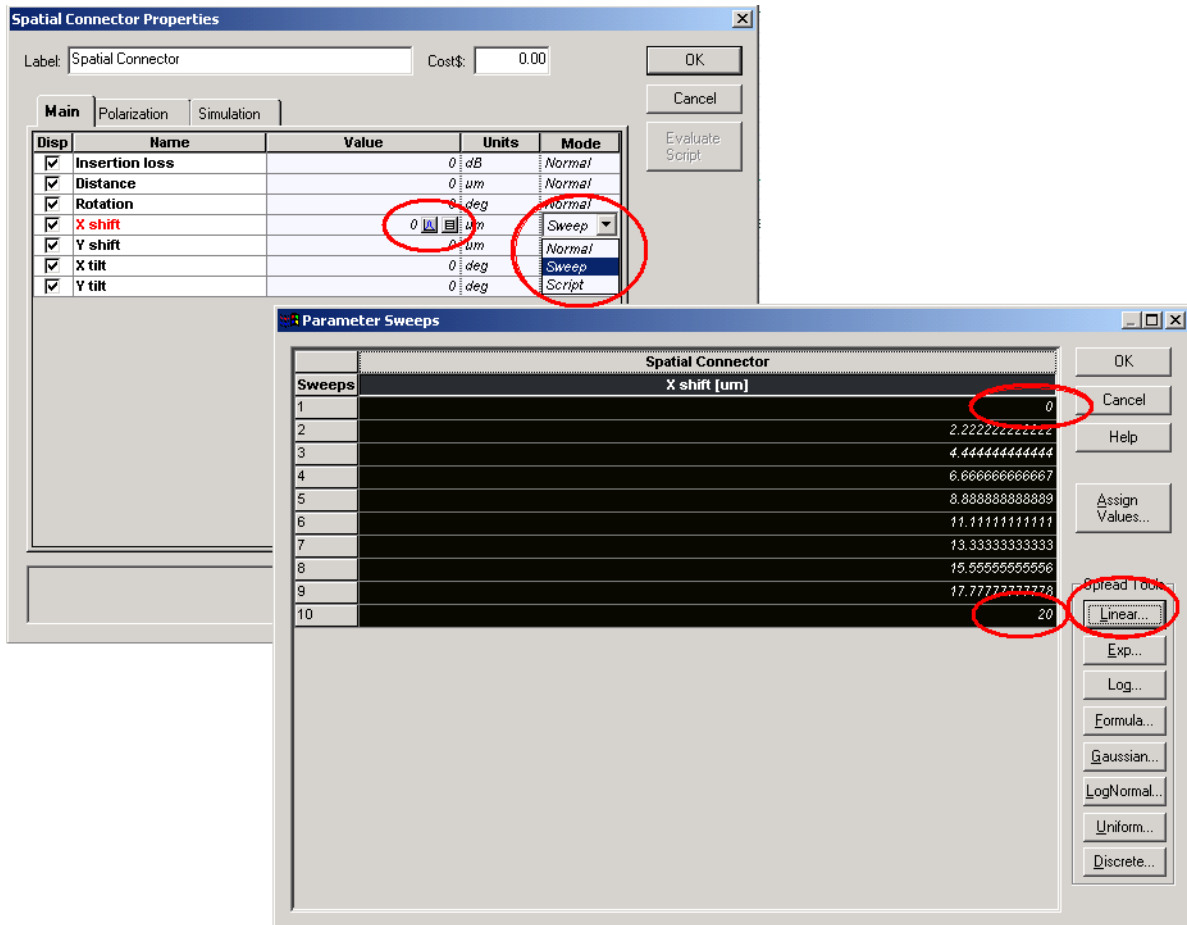
In order to visualize the effects of misalignment between the transmitter and receiver, we will apply a longitudinal shift in the optical field using the spatial connector component.

Using OptiSystem parameter sweep feature, we will apply a spatial shift from 0 to 20 μm in the optical signal.

- For the Spatial connector, change the mode of the parameter *X shift* to Sweep.
- Set the total number of sweep iterations to 10.
- In the parameter sweep dialog box, use a linear spread for the parameter *X shift* from 0 μm to 20 μm (Figure 19).



Figure 19 Using parameter sweep for the connector parameter X shift from 0 to 20 μm

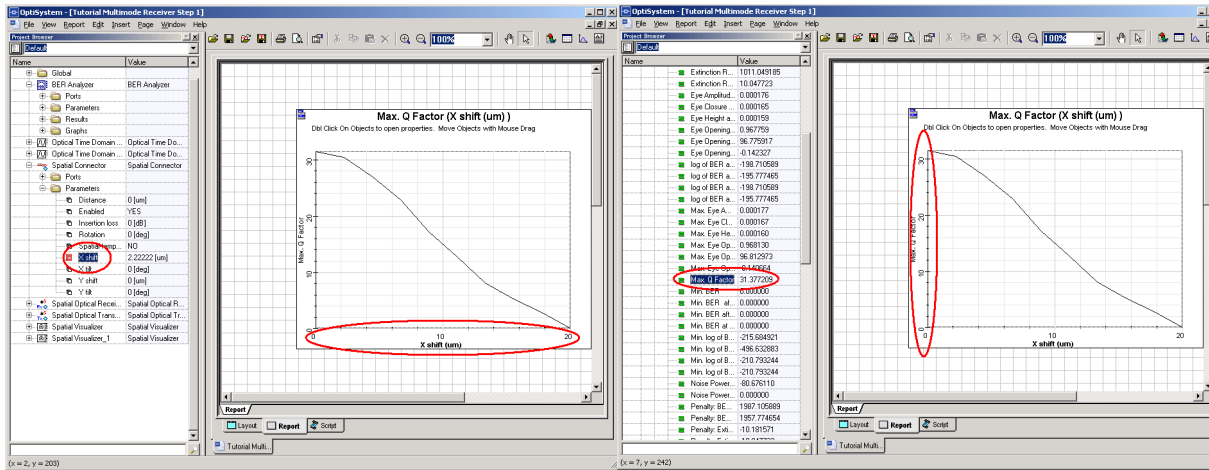


- Run the simulation

The simulation will run 10 times, one for each value of the *X shift* parameter. At the end of the calculation, we can browse the results selecting the current sweep iteration, or we can visualize the results of Q-Factor versus *X shift* using the report page.

- Select the report page.
- Drag and drop a 2D graph into the report.
- From the Project Browser, select the Spatial Connector and drag and drop the parameter *X shift* into the X-axis of the graph.
- From the Project Browser, select the BER Analyzer and drag and drop the result *Max. Q Factor* into the Y-axis of the graph (Figure 20).

Figure 20 Combining parameters and results after using report page



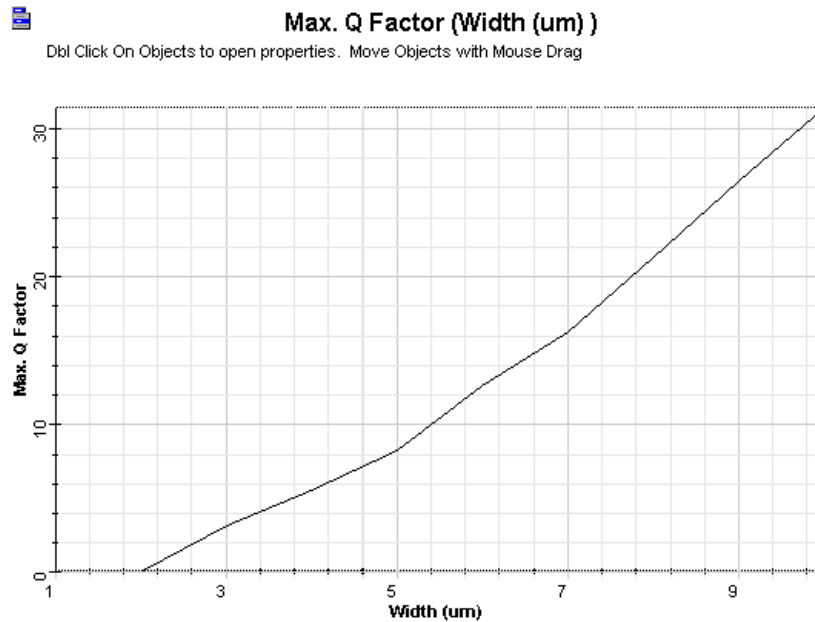
As you can see in the Figure 20, the value of Q is decreasing. This is because the optical field is being shifted outside of the photodetector active area, centered at (0,0) with a width of 10 μm . We can achieve similar results by changing the active area width additionally to the shifting of the field.

- For the Spatial connector, change the mode of the parameter *X shift* to Normal, and set the value to 0.
- For the Spatial receiver, change the mode of the parameter *Width* to Sweep.
- In the parameter sweep dialog box, use a linear spread for the parameter *Width* from 10 to 1 μm .
- Run the simulation
- From the Project Browser, select the Spatial Optical Receiver and drag and drop the parameter *Width* into the X-axis of the report page 2D graph.

The result is presented in Figure 21, where we can see the increase of the Q-Factor with the increase of the active area of the photodetector.





Figure 21 The value of Q-Factor versus photodetector active area width



Multimode Fibers and Links

- There are two multimode fibers available in the component library that take in to account the spatial properties. They are components that take in to account the power coupling and propagation of the multiple fiber modes.
- The first model is a general-purpose multimode fiber, with user-defined refractive index profile, and a numerical mode solver. The second model is a graded-index fiber with parabolic index, which uses an analytical mode solver.

Table 6 Multimode fibers

Component name	Folder	Icon
Measured-Index Multimode Fiber	Optical Fibers Library/Multimode/	
Parabolic-Index Multimode Fiber	Measured-Index Multimode Fiber	

System setup

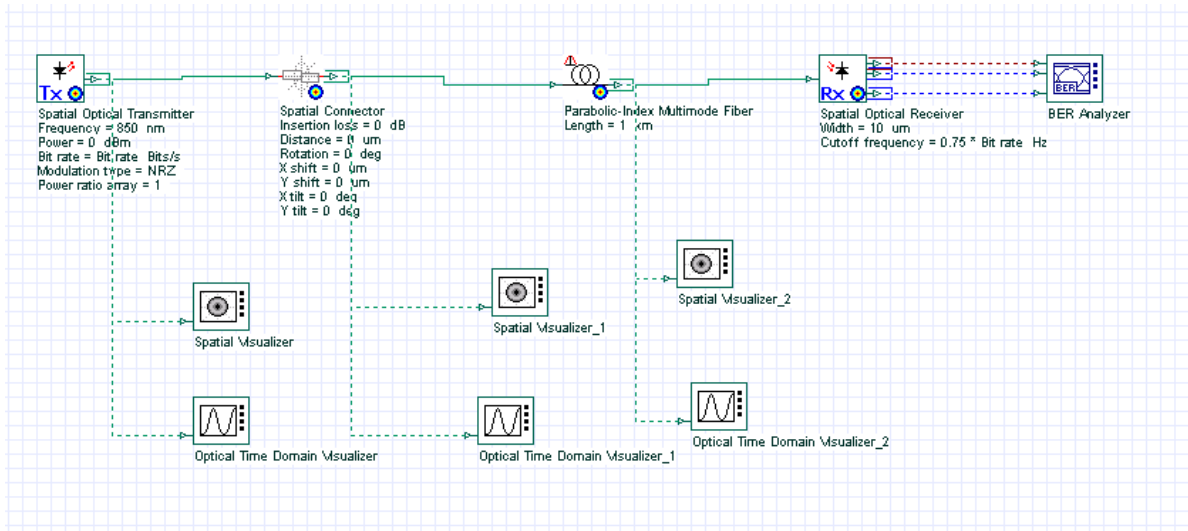
Following the same steps to generate the layout of Figure 17, we can add the fiber component to design the basic multimode link:

- From "Default/Optical Fibers/Multimode", drag and drop the "Parabolic-Index Multimode Fiber" into the layout.



Add additional visualizers to the layout and connect the system according to the Figure 22.

Figure 22 A link using the multimode fiber



Calculating Fiber Properties

The main parameter of the Parabolic-Index multimode fiber is the *Core radius*. For this link we are using a $50\ \mu\text{m}$ fiber.

In order to evaluate the fiber properties, such as effective index, group delay and the number of modes, we can calculate fiber properties before running the simulation:

- For the Fiber, enable the parameter *Calculate graphs* (Graphs tab).

We can also calculate the spatial profile graphs for individual modes, and the overfilled mode profile:

- For the Fiber, enable the parameter *Spatial profile graphs* and *Spatial overfilled graph* (Graphs tab).
- Press *OK*.

After pressing *OK*, the modes will be calculated and the mode number, radial and spatial graphs will be internally calculated.



Viewing the fiber properties

In order to see the graphs from the multimode fiber, we should use the Project Browser (Figure 23).

Figure 23 Multimode fiber graphs from the Project Browser

Parabolic-Index Multimode Fiber		Parabolic-Index Multi...
+	Ports	
+	Parameters	
+	Graphs	
	Effective index	0
	Group delay	0
	LP[m,n] index array - m	0
	LP[m,n] index array - n	0
	Power coupling and modal delay	0
	Radial profile - individual a	0
	Radial profile - individual b	0
	Refractive index profile	0
	Spatial profile - individual a	
	Spatial profile - individual b	
	Spatial profile - overfilled a	
	Spatial profile - overfilled b	

By double-clicking on the graph name, we can see the graphs for the different fiber properties.

For example, Figure 24 presents the Refractive index profile graph.

Figure 25 presents the mode number graphs, e.g. LP(m,n) indexes m and n, Effective index and Group delays.



Figure 24 Refractive index profile for the parabolic index fiber

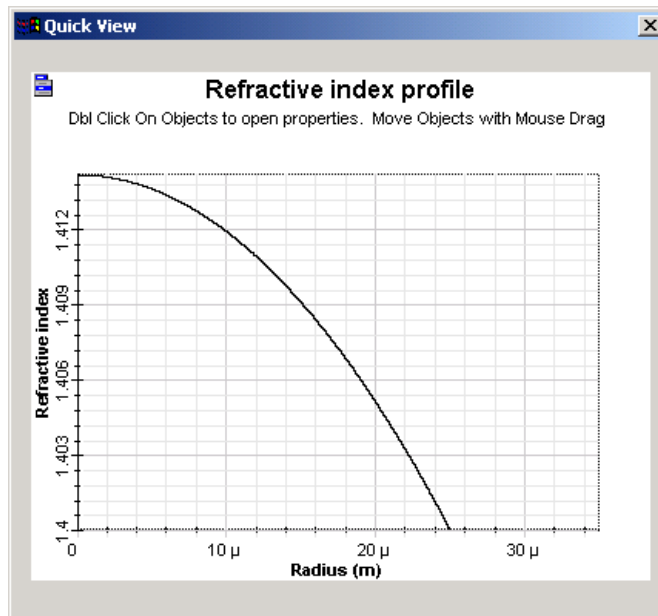
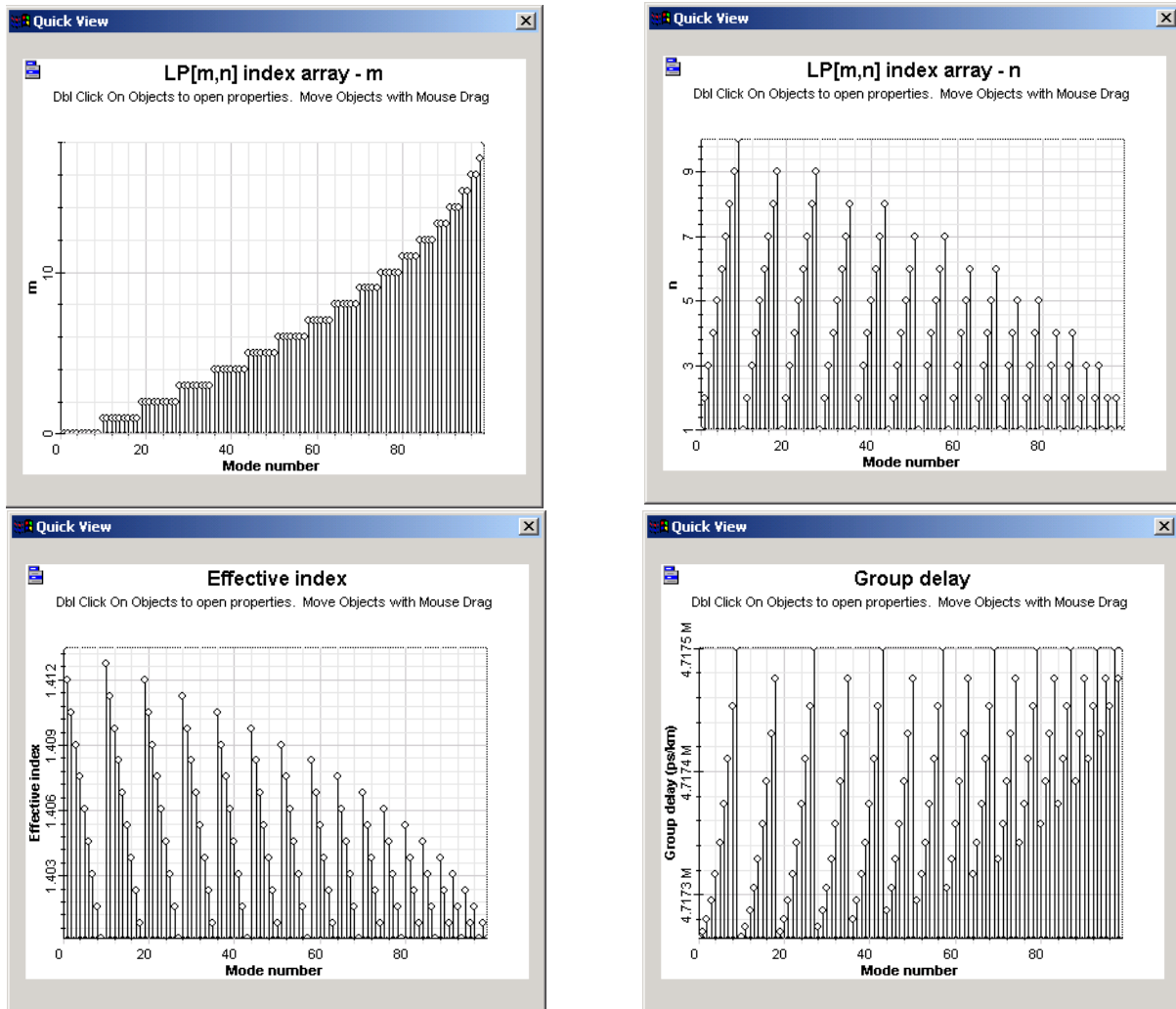


Figure 25 Mode index graphs for the parabolic index fiber



The remaining spatial and radial graph for the individual mode is presented in Figure 26.

The spatial overfilled graph is presented in Figure 27.



Figure 26 Individual mode intensity radial and spatial profile graphs

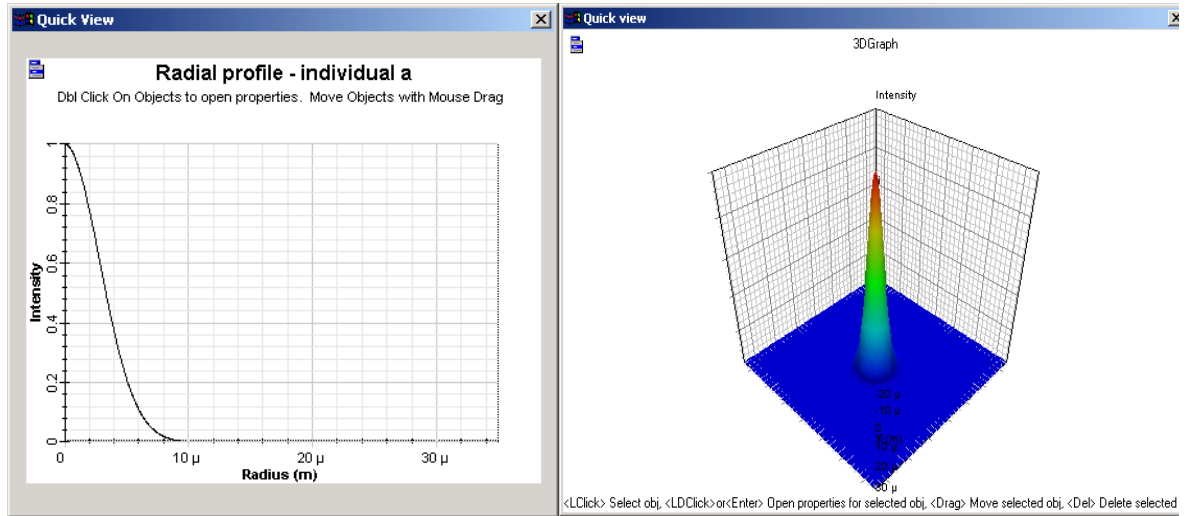
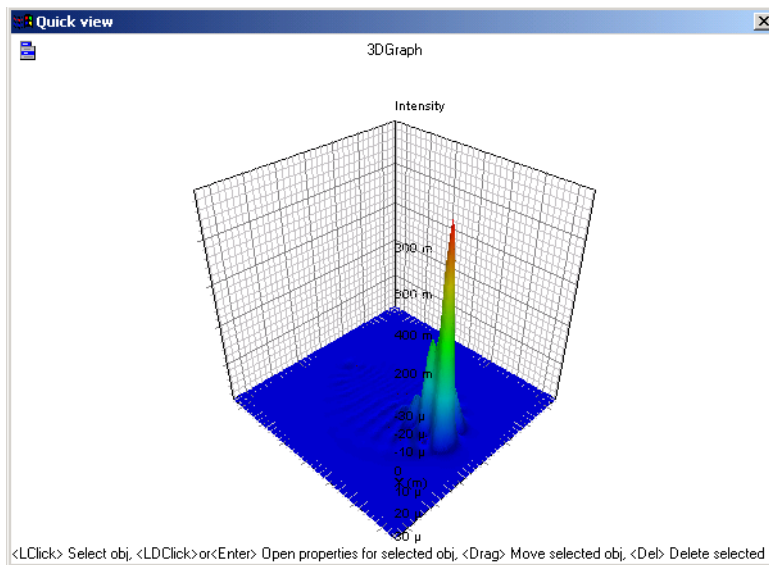


Figure 27 Spatial overfilled graph



The fiber graphs will be calculated every time you press OK in the component dialog box and after the link calculation. In order to save calculation time, we will disable the spatial graphs calculation.

- For the Fiber, disable the parameter *Spatial profile graphs* and *Spatial overfilled graph* (Graphs tab).
- Press OK.

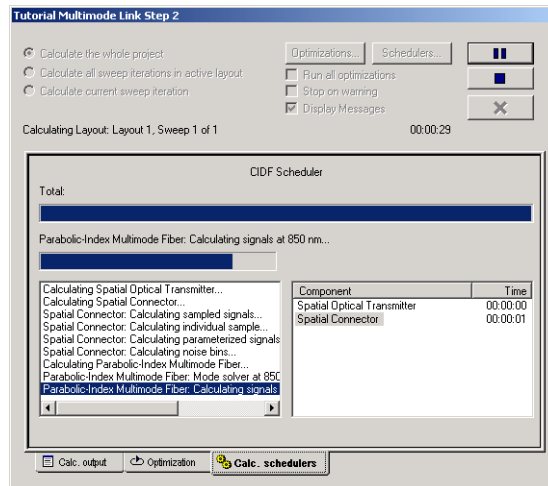


The next step is to calculate the multimode link:

- Run the simulation
- During the calculation, in the calculation dialog box select the 'Calc. schedulers' tab.

Observe the calculation progress of the multimode fiber (Figure 28). This is important to estimate the time that simulation will take when calculating multimode links.

Figure 28 Calculation progress for the multimode fiber



Viewing the results

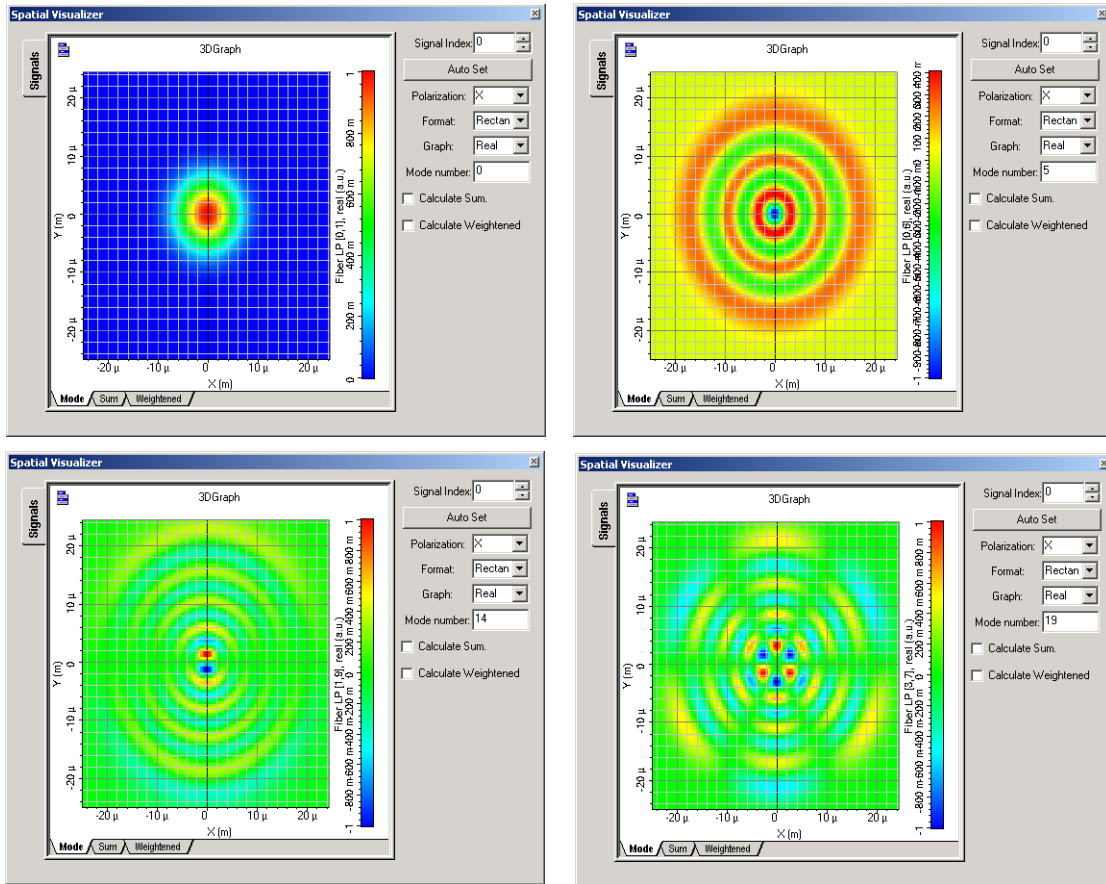
We can use the visualizer to display the simulation results. The first results that we will look at are the spatial profile of the modes at the fiber output.

In order to browse through the modes available at the fiber output port, change the mode number parameter of the third spatial visualizer and press <Enter>. The visualizer will recalculate the graphs and present the individual mode at the parameter mode index.

Figure 29 presents the real values using rectangular format for some of the modes:



Figure 29 Spatial visualizer at the fiber output displaying mode index 0, 5, 12 and 15

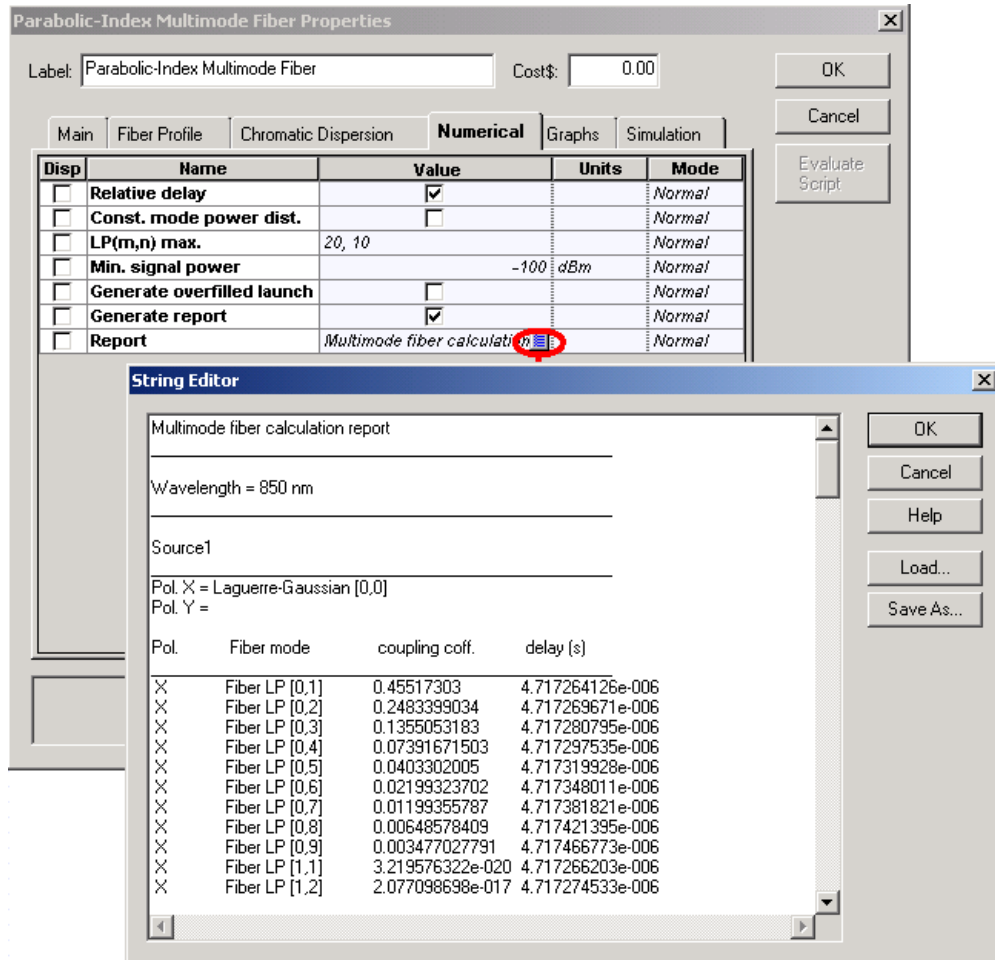


We can also visualize the coupling coefficients and the modal delay at the fiber output for each input signal wavelength and spatial mode by using the multimode fiber calculation report (Figure 30):

- For the Fiber, click on the *Report* parameter (Numerical tab).



Figure 30 Calculation report from multimode fiber

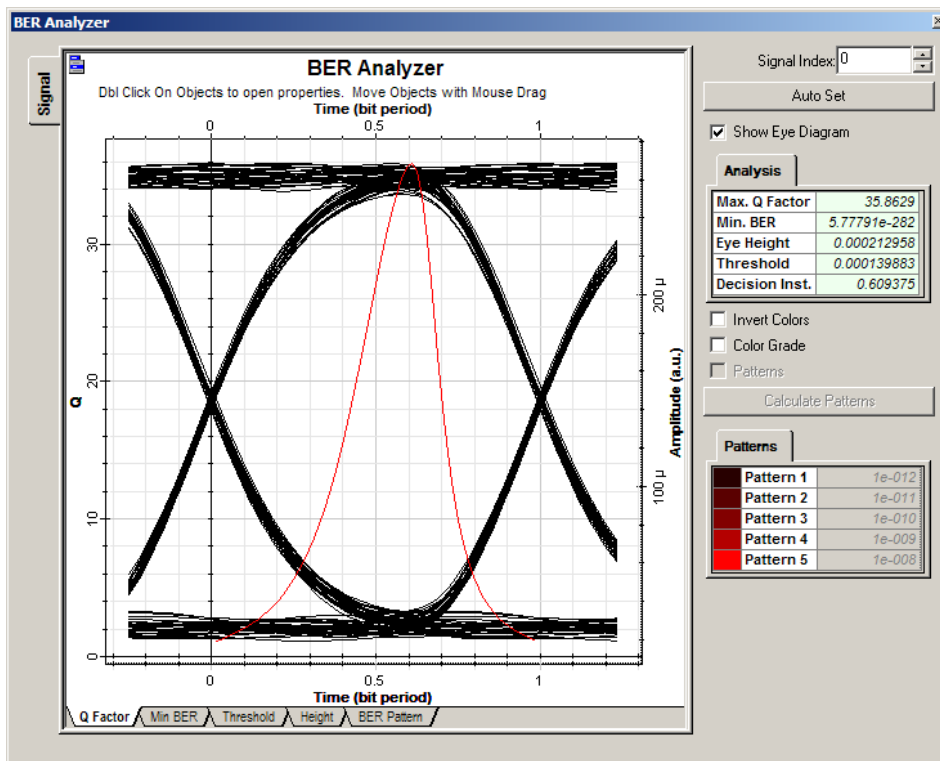


The graph with the coupling coefficients and group delay is also available from the project browser.

Finally you can visualize the graphs from the BER Analyzer (Figure 31).



Figure 31 BER Analyzer at the receiver output



The same steps can be applied to second multimode fiber model, the Measured-Index Multimode Fiber.

Because the index is measured, the user should provide the refractive index profile with the proper format instead of simply changing the core radius of the parabolic index fiber.