

OptiFDTD

Getting Started

Finite-Difference Time-Domain

OptiFDTD Version 16.0.1 for Microsoft Windows® 10 64-bit



Table of Contents

1	Installing OptiFDTD	4
1.1	Hardware and software requirements	4
1.2	Protection key	4
1.3	OptiFDTD directory	4
1.4	Installation	5
1.5	OptiFDTD samples	5
1.6	Technical support	5
2	Introduction to OptiFDTD	7
2.1	What are OptiFDTD and FDTD?	7
2.2	OptiFDTD workflow	8
2.3	OptiFDTD coordinate system	9
3	Quick Start	11
3.1	Starting OptiFDTD	11
3.2	Opening the sample ring resonator project file	12
3.3	FDTD domain dimensions	14
3.4	Material properties	15
3.5	Component Geometry (ring)	17
3.6	Properties of the input source	19
3.7	Detector properties	25
3.8	Running a 2D simulation	28
3.9	Analyzing the results	32
3.10	Working with the 3D editor	38

Copyright © 2022 Optiwave, All rights reserved.

All OptiFDTD documents, including this one, and the information contained therein, are copyright material. No part of this document may be reproduced, stored in a retrieval system, or transmitted in any form or by any means whatsoever, including recording, photocopying, or faxing without prior written approval of Optiwave.

Disclaimer

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

1 Installing OptiFDTD

Before installing OptiFDTD, ensure that the selected system meets the system requirements as described below.

1.1 Hardware and software requirements

OptiFDTD requires the following minimum system configuration:


- Microsoft Windows 7, 8.1 or higher, or 10.
- 64 bit operating system and processor (Intel x64 or AMD64)
- Personal computer with Pentium 4 or higher processor 1.2 GHz (or higher). Multi-core CPU is required for increased performance using parallel processing.
- 8 GB of RAM or higher (dependent on the size of calculated problem)
- 1 GB free hard drive disk space
- 1024 x 768 graphic resolution, 16-, 24-, or 32-bit DirectX and OpenGL capable graphics adapter

1.2 Protection key

A hardware protection key is supplied with the software. Note: Please ensure that the hardware protection key is **NOT** connected during the installation of OptiFDTD.

To ensure that OptiFDTD operates properly, verify the following:

- The protection key is properly connected to the parallel/USB port of the computer.
- Ensure that there is no conflict between the OptiFDTD protection key and any other hardware keys that may be installed.

 Use a switch box to prevent protection key conflicts. Ensure that the cable between the switch box and the computer is no more than one meter in length.

1.3 OptiFDTD directory

By default, the OptiFDTD installer creates an OptiFDTD directory on your hard disk in the default location:

```
C:\Program Files\Optiwave Software\OptiFDTD 16 \
```

The OptiFDTD directory contains the following sub-directories:

- **\bin** — executable files, dynamic linked libraries

- **\doc** — OptiFDTD support documentation

The documentation and executable files have shortcuts in the Windows **Start** menu (or tiles in Windows 8).

1.4 Installation

OptiFDTD can be installed on 64 bit versions of Windows 7, 8.1 or higher, or 10. It is recommended that all Windows programs are shut down before running the setup program.

To install OptiFDTD from a CD ROM, use the following procedure:

1. **Log on** as the Administrator, or log on using an account with Administrator privileges.
2. **Insert** the OptiFDTD installation CD into the CD ROM drive.
3. On the Taskbar, click **Start** and select **Run**.
*The **Run** dialog box appears.*
4. In the *Run* dialog box, **type** **F:\setup.exe**, where **F** is your CD ROM drive.
5. **Click OK** and follow the screen instructions and prompts.
6. When the installation is complete, a reboot of the system will be required.

1.5 OptiFDTD samples

The samples installer runs directly after the end of the installation of the main OptiFDTD application. By default, the samples are installed under the *My Documents* directory in a folder named OptiFDTD 16 Samples. The default location can be changed in the installer.

If needed, samples can be re-installed by **double-clicking** the file *OptiFDTD_Samples.exe* in the OptiFDTD 16 installation directory.

Additional samples and application cases can be found online. Simply **select** *Help->Download Samples from OptiFDTD*, or visit <http://www.optiwave.com/optiftd/> for more information.

1.6 Technical support

Phone	(613) 224-4700—Monday to Friday, 8:30 a.m. to 5:30 p.m. Eastern Standard Time
Fax	(613) 224-4706

E-Mail	support@optiwave.com ¹
URL	www.optiwave.com ²

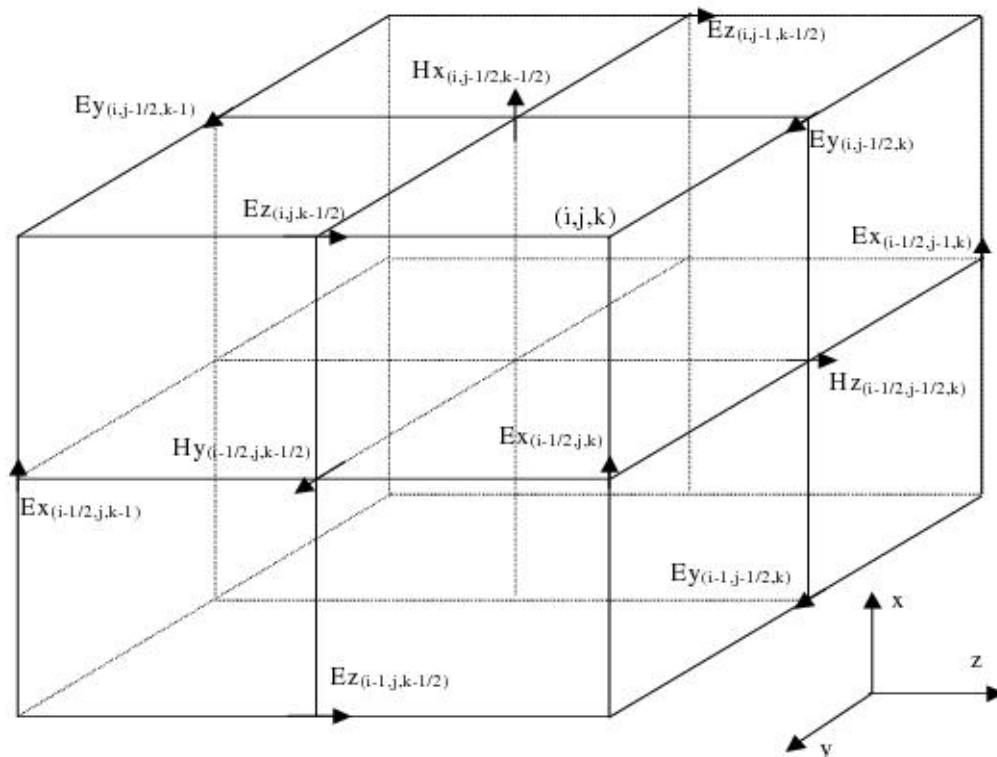
¹ <mailto:support@optiwave.com>

² <http://www.optiwave.com/>

2 Introduction to OptiFDTD

2.1 What are OptiFDTD and FDTD?

OptiFDTD is a powerful, highly integrated, user-friendly software that allows computer aided design and simulation of advanced passive photonic components. The OptiFDTD software package is based on the finite-difference time-domain (FDTD) method. The FDTD method has been established as a powerful engineering tool for the simulation of integrated and diffractive optics devices. The method directly solves Maxwell's equations with no pre-processing assumptions allowing FDTD simulations to define a geometry, a source and detectors and simultaneously model propagation, scattering, diffraction, reflection, polarization effects. It can also model material anisotropy and dispersion without any assumptions on field behavior, such as the slowly varying envelope approximation. The method enables effective and powerful simulation capabilities and analysis of sub-micron devices with very fine structural details. A sub-micron scale implies a high degree of light confinement and correspondingly, a large refractive index difference of the materials to be used in a typical device design, a limitation of other numerical approaches.



1 A Yee cell. Magnetic field components are represented in blue, electric field components are shown in red.

FDTD relies on the discretization of both space and time of the problem space. The coupled electromagnetic fields are discretized along the a mesh comprised of **Yee cells**. Magnetic and electric field components are positioned in a staggered fashion on the edges of each cell. By calculating the finite difference between two field components on opposite edges of the cube, it is possible to obtain a second-order approximation of the spatial derivative of the field. Finite differences are used in the same way to approximate time-derivatives of the fields. The algorithm used to compute and update the simulation space for each time step the electric and magnetic fields is called the Yee algorithm. More details are given in the "Technical Background" manual. The accuracy of the solution to the computational problem should converge as the size of the space and time steps tend to zero. This added accuracy comes at the cost of computational resources in terms of the time required to complete a simulation and the memory required to store it.

Each cell within the simulation grid represents a small volume with set material properties. Materials in OptiFDTD can be of varied types:

- isotropic and diagonally anisotropic dielectrics
- dispersive materials using Drude, Lorentz, or Lorentz-Drude models
- perfect conductors
- nonlinear materials (2nd order, 3rd order, or Raman)

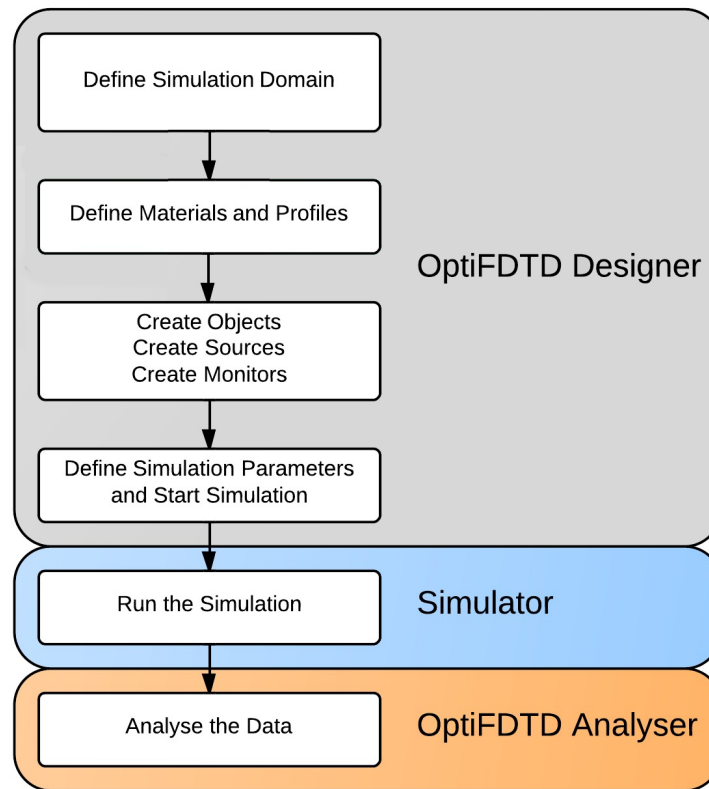
In addition to these materials, an FDTD simulation will require sources and detectors to be defined so that power can be injected into the mesh and fields fields can be monitored. Fields can be stored as time varying or in frequency using Discrete Fourier Transforms (DFTs).

2.2 OptiFDTD workflow

Creating and running an FDTD simulation using OptiFDTD can be accomplished using 4 main programs, described below:

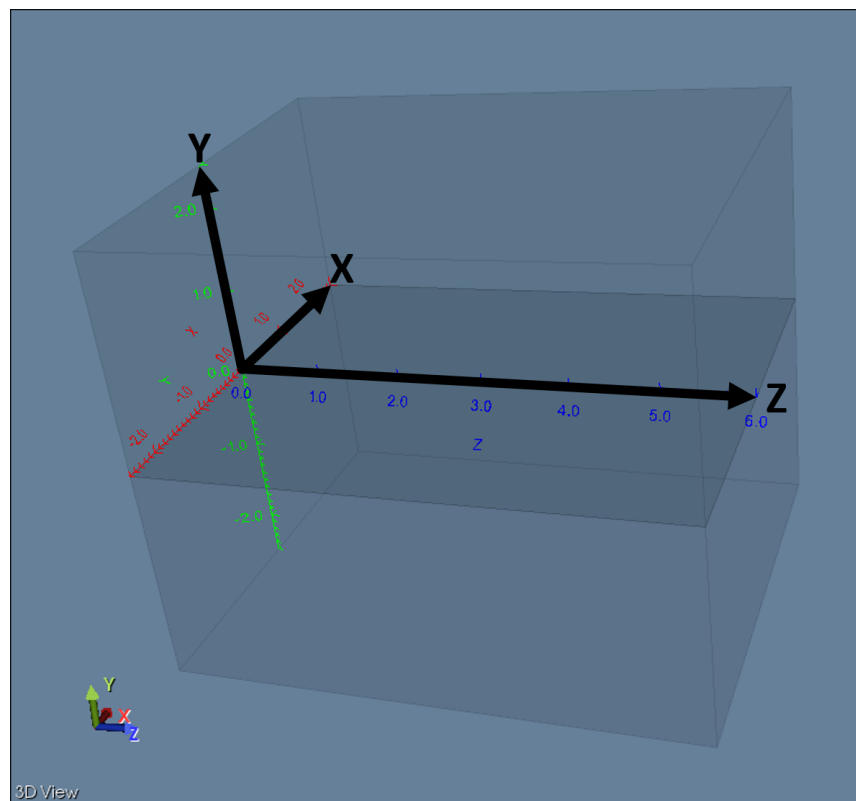
- **OptiFDTD Designer** - The main OptiFDTD program. From here, you can create new designs, set simulation parameters, write scripts and start simulations. Data is saved in a project file with the extension **.fdt**.
- **OptiFDTD Simulator** - Opens automatically when a simulation is run from Designer and processes the project files in the **fdt** file in order to perform the simulation. Simulation results are stored in a file with the extension **.fda**.
- **OptiFDTD Analyzer** - Loads and analyzes the result files produced by OptiFDTD Simulator (**.fda**). Contains extensive viewing options, analysis and post-processing features, and has the facilities to export data to other file formats.

A typical FDTD simulation design sequence can be defined as this: first define the simulation domain sizes, then define materials and profiles to be used in the simulation, then create the objects, light sources, and detectors that compose the simulation, run the simulation, and finally, analyze the results. The workflow of a typical simulation using OptiFDTD is shown below.



2.3 OptiFDTD coordinate system

OptiFDTD uses a coordinate system typical of waveguiding applications in which the transverse axes are **X** and **Y** while the **Z** axis is the axis of propagation. The use of this particular coordinate system is based upon the need to ensure compatibility with other Optiwave software such as OptiBPM. A representation of the coordinate system used is shown below.



The regions in **Y** that are above and below the **XZ** ($Y=0$) plane are named **Substrate** and **Cladding**, again matching traditional waveguide technology in which waveguides are fabricated on a Substrate. OptiFDTD does provide the ability to set the Substrate and Cladding regions to be different materials.

While working with OptiFDTD, it is necessary to take note of the following conventions:

- The **XZ** plane is the primary view in the Layout of the OptiFDTD Designer
- **X** is referred to as the width or vertical component
- **Y** is referred to as the depth
- **Z** is referred to as the length or horizontal component
- The 2D and 3D simulation domains are referred to as 2D and 3D wafers with their respective material properties defined in the corresponding tabs in the *Wafer Properties* form

It is possible to run either 2D or 3D simulations using OptiFDTD. Note, that when running 2D simulations the simulation region is the **XZ** plane.

3 Quick Start

This section will walk through the standard steps of working through an FDTD simulation:

- Opening a sample project
- Navigating the user interface and verifying/editing parameters
- Running a simulation
- Visualizing/Post-processing results

The example used in this section will be a ring resonator. Included in the documentation is a series of lessons with more advanced designs at [online examples](#)³. Additionally samples come with the installation package for review.

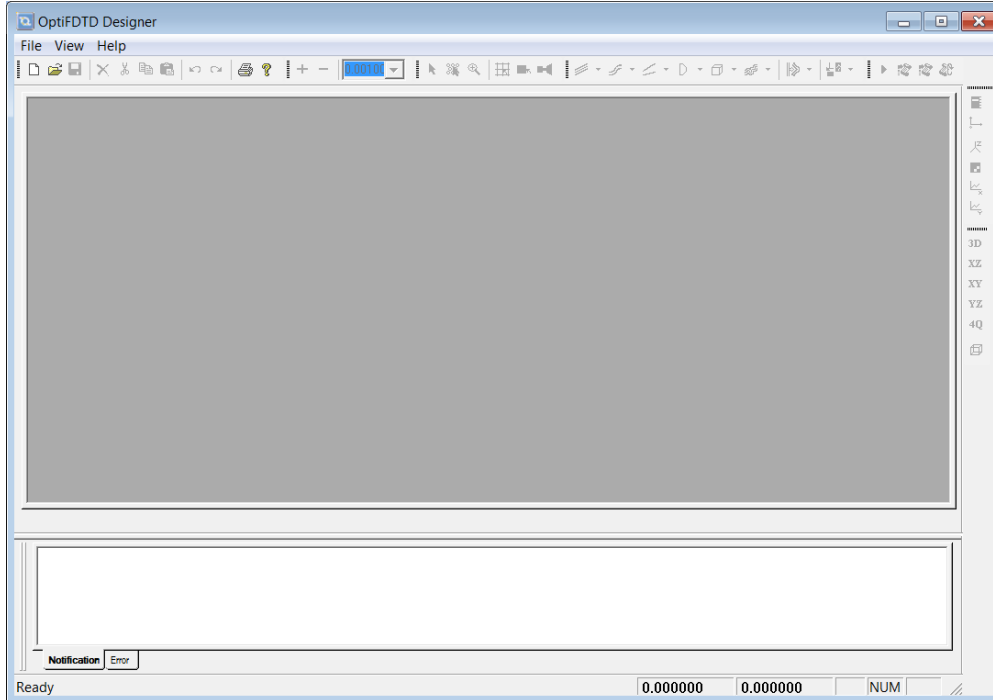
3.1 Starting OptiFDTD

Work related to the creation of designs and configuration of simulations is accomplished within the **OptiFDTD Designer** environment. To open the application, perform the following steps:

- **Open** *Windows Start Menu*
- **Select Programs > OptiFDTD 16.0 RC > OptiFDTD Designer.**

OptiFDTD Designer loads and the main application window appears.

³ <http://www.optiwave.com/optifdtd/>

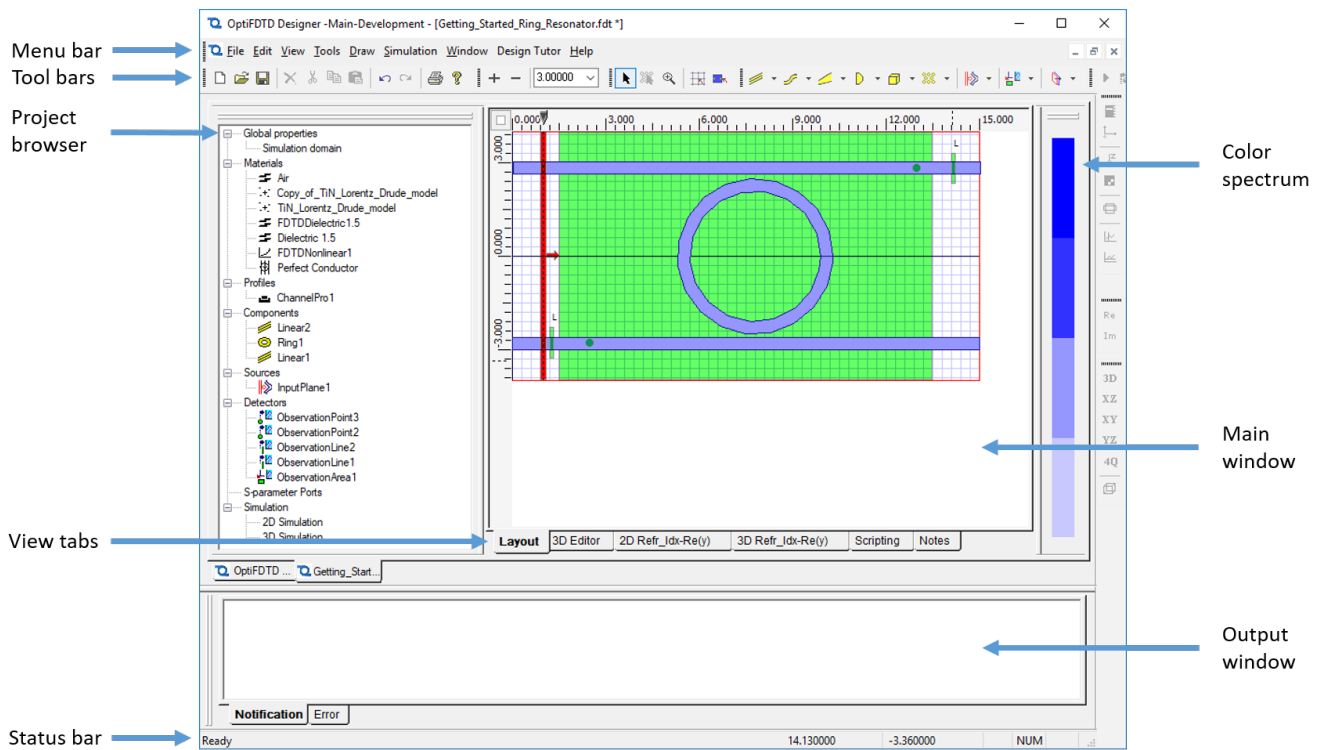


2 Main OptiFDTD Designer application window

3.2 Opening the sample ring resonator project file

To load the ring resonator sample file, perform the following procedure.

1. From the *File* menu, **select** *Open*.
2. Navigate to the user documents folder and **select** *OptiFDTD 16 Samples > 2D_Samples > photonic_components > Getting_Started_Ring_Resonator.fdt*
The ring resonator sample project opens in the main window



3 Ring resonator sample project

The default view displayed in the *Main window* represents the ring resonator design in the **XZ** plane. A complete 3D view of the structure can be seen in the *3D Editor*, which is activated through the *3D Editor* tab in the *view tabs*. The horizontal dimension corresponds to the Z axis, while the vertical dimension corresponds to the X axis.

The left frame is the *Project browser*. This is where all materials (all defined for this project), objects, sources, and detectors in use are displayed. Each node of the project browser represents an element. Parent elements can be collapsed or expanded by clicking the + or - sign next to their name. Right clicking on a node presents a contextual menu for either deleting the node or accessing its properties. The properties of a given element can be accessed by double-clicking the node.

Below the *Main window* is the *Output window* which includes tabs for notifications from scripts (Notifications) and errors (Error).

The Toolbars are shortcuts to many functions of the software, such as creating objects and waveguides, resizing the view, selecting elements and display options. They can be activated or deactivated by right clicking in an empty area of the toolbar regions. All the functions present in the toolbars are also accessible via the *Menu bar*. Toolbars are configurable in terms of their position, they can be moved through drag and drop functionality and will be stored for future sessions.

The *Status bar* displays information on the position of the mouse cursor inside the layout and if the program is running calculation or ready for user input.

⚠ Zoom functionality is available through the + and - buttons or the magnifying glass in the toolbars.

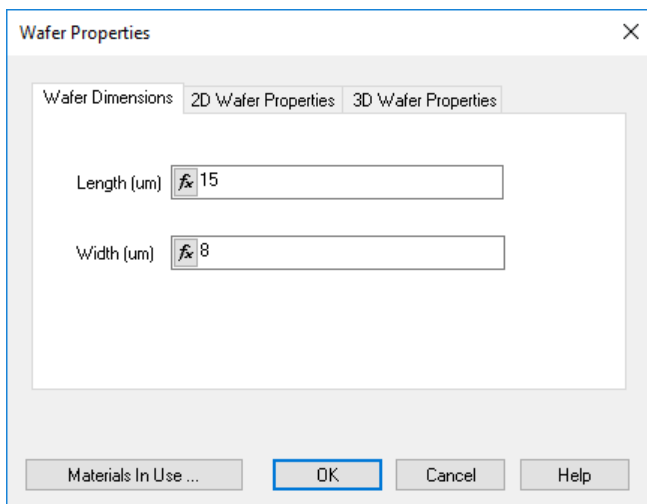
3.3 FDTD domain dimensions

The global dimensions of the FDTD calculation domain can through the wafer properties dialog:

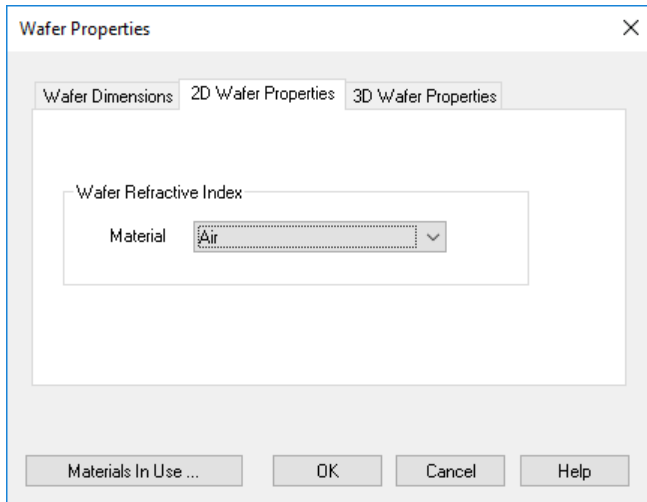
- **Double-click** on the node named Simulation Domain, under *Global Properties* in the project browser
The Wafer properties dialog opens.

OR

- **Select** *Edit > Wafer Properties* from the Menu bar



4 Wafer properties dialog



In the *Wafer Dimensions* tab, the 2D dimensions (in microns) of the FDTD grid can be set:

- *Length* defines the **Z direction** dimension of the grid
- *Width* defines the **X direction** dimension of the grid

The background material for the 2D simulation is set In the *2D Wafer Properties* tab, you can define the background material to use for the simulation. Any material that has been defined within this project can be assigned, for the purpose of this example this is set to the preset material Air. If a material needs to be created this can be done from the *Project Browser* or from the *Profile Designer* accessed through the *Materials in Use...*

3.4 Material properties

3.4.1 Opening the dielectric material properties

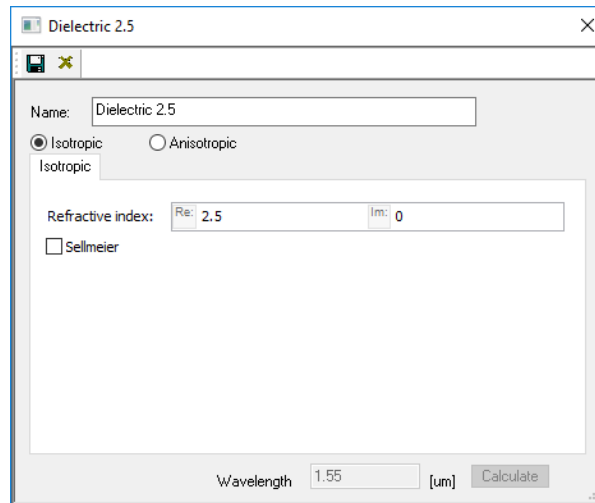
A material called Dielectric 2.5 has been defined and used as the material for the waveguide profile that will be used for the waveguides in this structure. The Dielectric 2.5 material definition can be access through the following:

- **Double-click** on the node named Dielectric 2.5 within the *Materials* node in the *Project browser*.

OR

- **Right-click** on the Dielectric 2.5 material and **select** properties

The Dielectric 2.5 material properties dialog opens.



5 Dielectric 2.5 material properties dialog

In this dialog, a basic dielectric material can be defined by its refractive index. The material can be **isotropic** or **anisotropic** and both the real value and imaginary values of the refractive index can be set in the boxes labeled *Re* and *Im* respectively.

- ⚠** The refractive index definition within OptiFDTD follows the engineering convention:
- **$Im > 0$** in the refractive index will model a material **gain**
 - **$Im < 0$** in the refractive index will model a material **loss**

- ⚠** The refractive index defined here is only valid for the central wavelength of the light source used. For simulations using Sine-Modulated Gaussian Pulse waves, this will result in non physical representation of the material as the wavelength departs from the central wavelength. For more accurate models, taking into account chromatic dispersion, please consider using dispersive material definitions.

3.4.2 Opening the profile properties

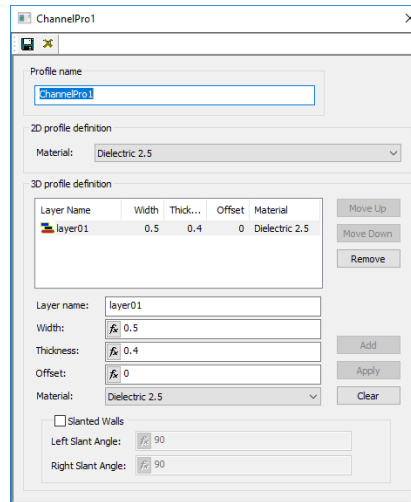
In OptiFDTD waveguides are defined as a path while the cross-sectional profile is defined in the *Profiles* section of the *Project browser*. To gain access to the waveguide profile properties proceed to the following:

- **Double-click** on the node named ChannelPro1 of the *Profiles* node in the *Project browser*.


OR

- **Right-click** on the ChannelPro1 profile and **select** properties

The ChannelPro1 dialog opens.



6 ChannelPro1 profile properties

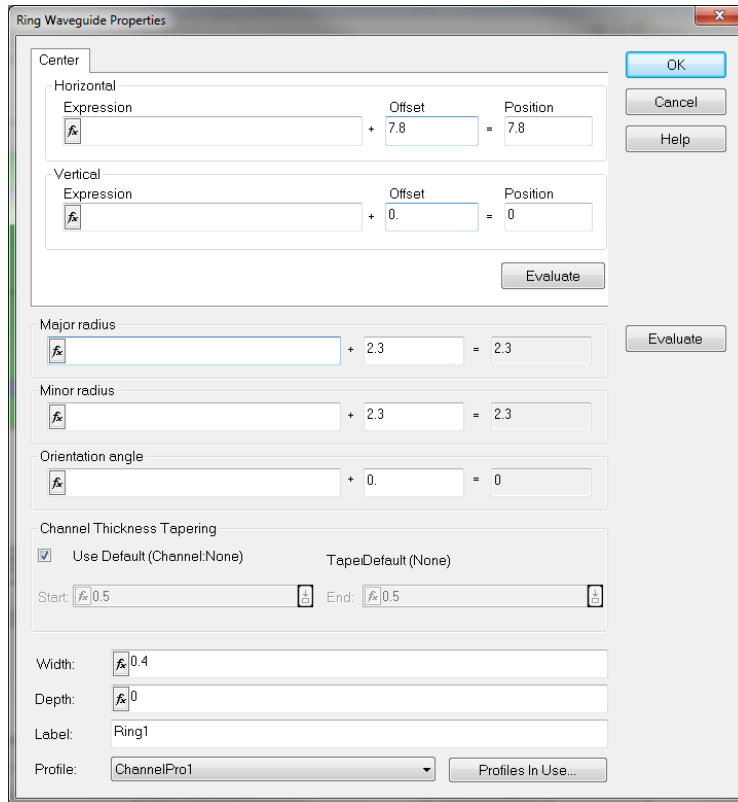
In 2D simulations, the profile has only a material definition. It can be seen that in the *2D profile definition* list, the Dielectric 2.5 previously described is selected. If assigned material is changed the profile must be stored ().

In 3D simulations it is necessary to define the layer structure of the waveguide.



3.5 Component Geometry (ring)

The geometrical properties of the ring can be accessed and modified in 3 different ways:

- By double-clicking on the ring drawn in the Main window
- By double-clicking on the node named *Ring1* in the project browser
- By right-clicking on the node named *Ring1* in the project browser and selecting **Properties**



7 Ring Waveguide properties dialog

⚠ Any function with an asterisk or  will accept numerical inputs, equations, and the use of user-defined parameters. Clicking on the  icon will launch the Variables and Functions dialog which will allow parameters to be defined. Details on the use of parameters in expressions or scripting can be found in the User's Reference Manual accessed through the help menu or by pressing F1.

⚠ Some controls consist of 2 fields and will hold the value of the sum of the two - Expression and Offset :

- *Expression* - used to input either a numerical value or an expression, which can be an equation with user defined parameters.
- *Offset* - Only accepts a numerical value and is used to offset the expression.

If the given property of an object is modified through the editors using the mouse the modifications will be reflected in the *Offset* field.

The tab on the top of the dialog, named *Center*, is used to define the position of the center point of the ring (Note: All units are defined in microns)

- The **Z center position** is controlled by the *Horizontal* control
- The **X center position** is controlled by the *Vertical* control.

The *Major radius*, *Minor radius* and *Orientation angle* can be defined in the same way. The ring is an ellipse, having a major radius and a minor radius.

- *Major radius* specifies the major radius of the ring
- *Minor radius* specifies the minor radius of the ring
- *Orientation angle* specifies the angle between the major direction and the Z axis (in degrees).

The *Width* box controls the size of the ring, which is the distance between the outer edge and the inner edge of the ring.

The *Depth* box controls the position of the ring in the **Y dimension**.

The *Profile* combo box enables you to choose between the different material profiles defined in the Profiles and Materials editor. More detailed explanations on the *Ring waveguide* and all other objects can be found in the User's Reference Manual.

3.6 Properties of the input source

Now that we have verified the material properties and the geometry of our simulation, we can check the properties of the light source. The source is defined under the *Sources* node of the project browser. You can define as many sources as needed. They can be of four types:

- **Z-direction Input Plane** - The light intensity distribution is defined across an incident plane oriented in the X-Y plane, light propagates in the **Z direction**
- **X-direction Input Plane (2D)** - The light intensity distribution is defined along a line parallel to the Z axis, light propagates in the **X direction**. **NOTE: this source is only implemented for 2D simulations.**
- **Point-source** - Light is emitted in every direction (spherical waves) from a source point which is the excitation of a single mesh point. Polarization options include Ey (2D TE), Hy (2D TM), or any E field component (3D).

Additionally the user can specify the transverse field distribution along the input plane:

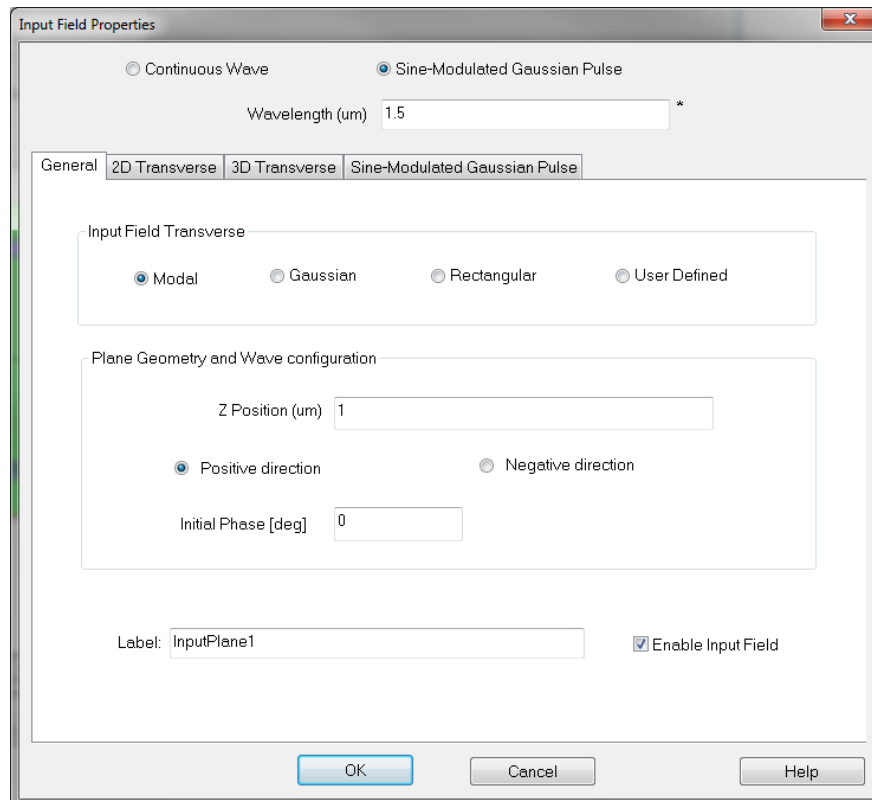
- **Modal** - The field is calculated from the **OptiMode** mode solver. The user can select the desired waveguide that intersects the input plane and the solver will use the cross-section of this waveguide for mode calculations.
- **Gaussian** - Standard Gaussian distribution.
- **Rectangular** - Distribution follows a rectangular transverse profile.
- **User-defined** - Field profiles from other software packages such as **OptiBPM** or third party tools can be imported and injected through the input plane.

Finally, a source can be defined as either a continuous wave (CW) source, with a fixed wavelength or frequency, or as a Sine-Modulated Gaussian Pulse (**SMGP**). The **SMGP** is a convolution of a **CW** light source and a Gaussian envelope function, giving the signal a pulse shape and time dependence and in turn a range of frequency components.

3.6.1 Geometrical parameters of the source

To verify the geometrical parameters of the source used, proceed as follows:

- **Double-click** on the node named *InputPlane1* in the *Project browser*.



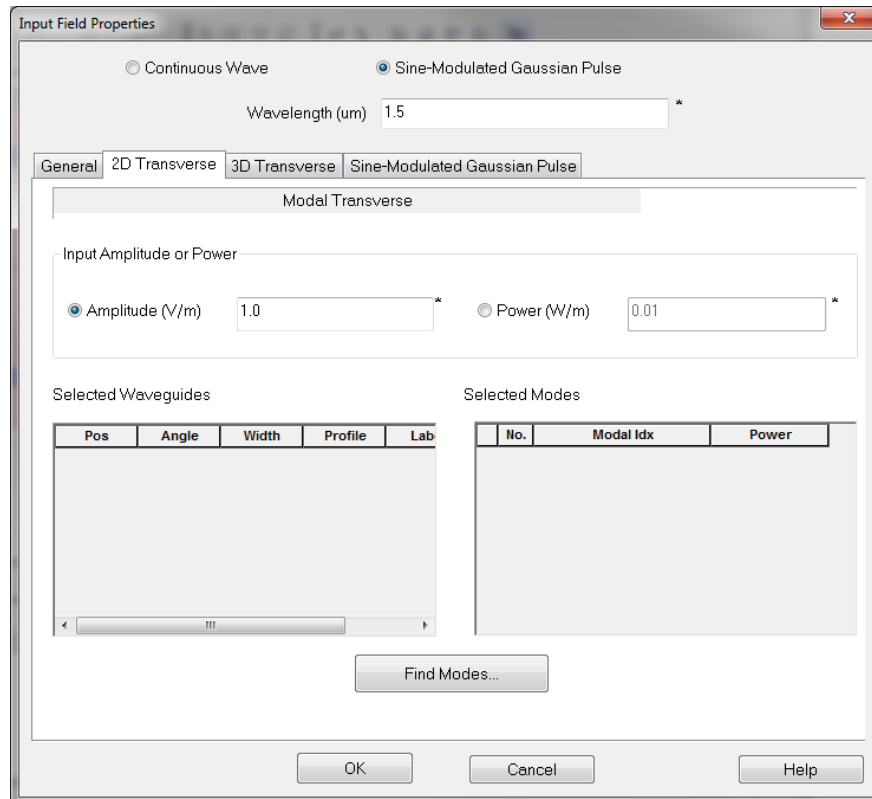
8 Input Field properties dialog

In this *Input Field Properties* dialog, you can define the type of excitation (Continuous Wave or Sine-Modulated Gaussian Pulse), the light distribution model for the plane (in the case of this example, **Modal**), and the position of the plane in the Z-direction.

3.6.2 Transverse spatial distribution of the source

The properties of the transverse distribution can be accessed by doing the following:

- **Click** on the tab labeled "2D Transverse". Similarly "3D transverse" would be used for a 3D simulation

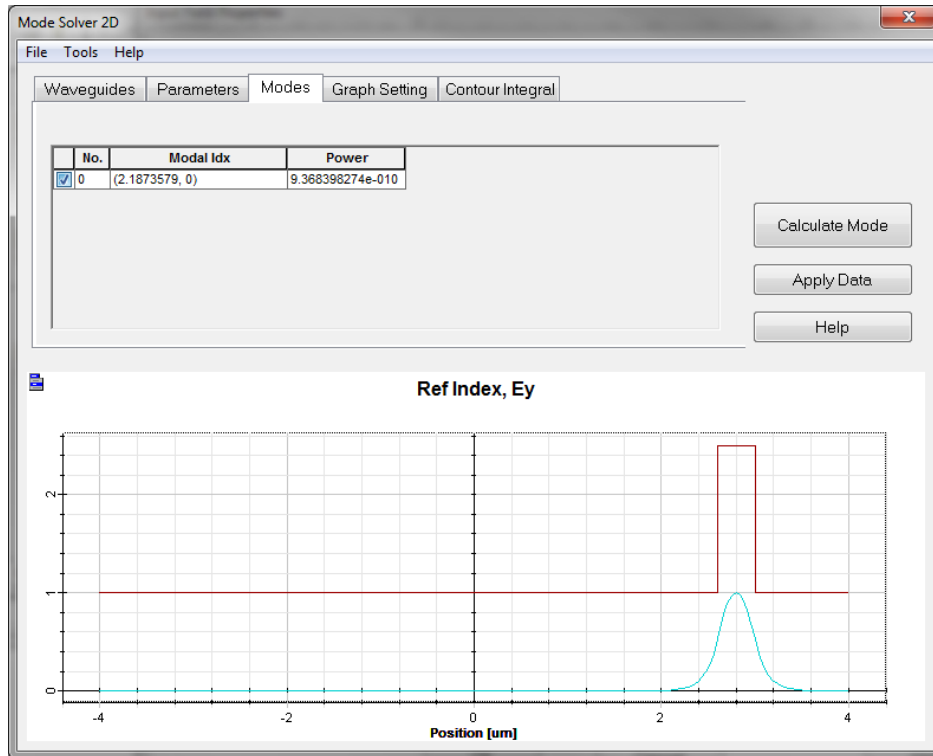


9 2D Transverse tab

For the ring resonator example, it can be observed that there are no entries in the *Selected Waveguides* or *Selected Modes* windows. It is necessary to calculate the modes and then select the desired ones for the 2D transverse plane. To do so:

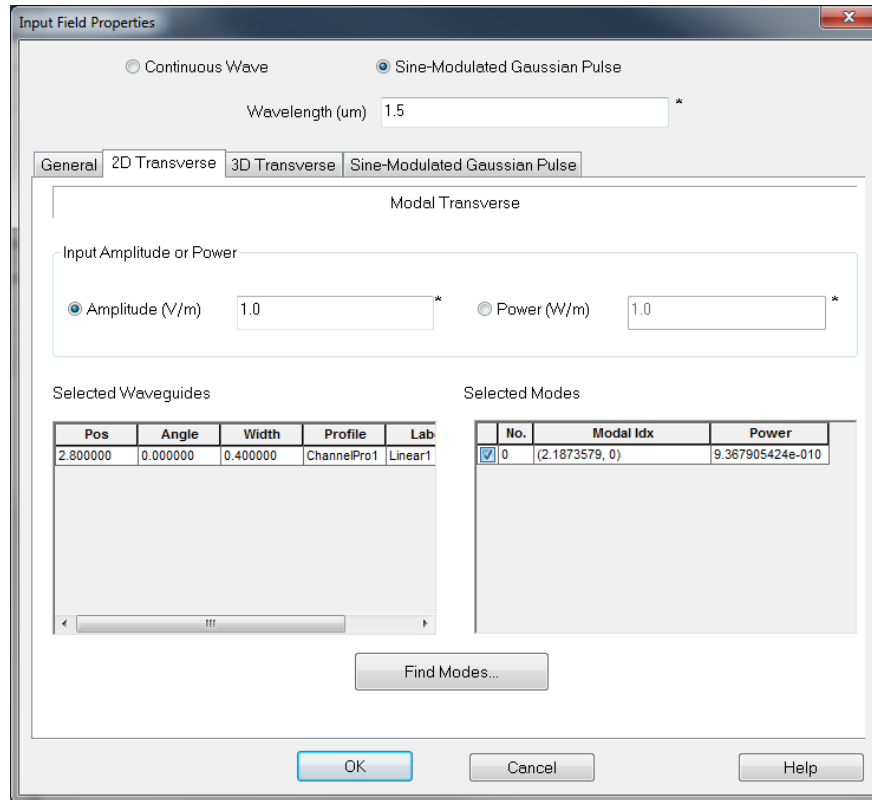
1. **Click** the *Find Modes...* button to open the mode 2D solver
2. In the *Select waveguides for mode calculating* list, select only the waveguide labeled Linear1.
3. **Click** the *Calculate Mode* button to solve the modes of the waveguide cross-section. *Any modes supported will be displayed.*
4. **Click** the checkbox next to Mode No. 0 to display the mode intensity distribution. The graph window updates to show the mode and geometry of the waveguide, as seen below.
5. **Click** *Apply Data* to apply the selected mode to the input source.
6. **Click** *OK* to close the *Input field properties*.

In the Mode Solver, you can modify parameters such as polarization, number of modes to find and sampling in the **Parameters** tab.



10 Calculated Mode (2D Mode Solver)

Now the 2D Transverse dialog box includes a selected mode (this is the mode that will be used for the simulation):

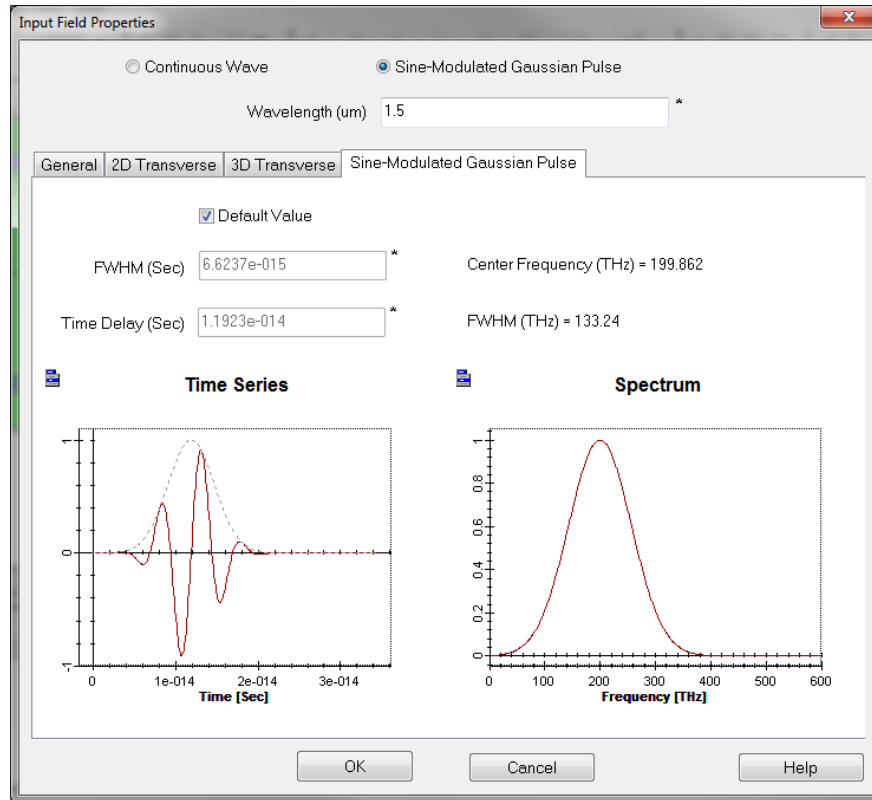


11 2D Transverse tab (with selected mode)

3.6.3 Temporal definition of the source

To review the temporal definition of the source (which in this case is selected to be Sine-Modulated Gaussian Pulse - SMGP), perform the following:

- Click on the tab labeled *Sine-Modulated Gaussian Pulse*. **NOTE: This is only present if the Sine-Modulated Gaussian Pulse option is selected.**



12 Sine-Modulated Gaussian Pulse tab

In this tab, the settings for the time and frequency domains for the Gaussian pulse can be set and graphs of both domains are presented. The *Wavelength* of the modulated signal has been set to 1.5 μm and the time domain parameters FWHM and Time Delay have been automatically calculated (by checking the Default Value check box). These quantities can be user specified. Based on these settings, the spectral characteristics of the SMGP can be verified. For this design the *Center Frequency* of the SMGP, controlled by the source wavelength, is 199.862 THz and the FWHM (THz), controlled by the *FWHM* field, is approximately 133 THz. In depth details about defining sources can be found in the "Technical Background".

- ⚠ It is important to ensure that the pulse waveform has both a smooth turn-on and turn-off. This ensures that higher frequency components (resulting from rapid amplitude changes) are kept to a minimum. If uncertain then the *Default Value*, obtained by selecting the checkbox, will ensure that this condition is met as well as the largest bandwidth possible for the given wavelength.

Full zoom and pan capabilities are available for both the *Time Series* and *Spectrum* through the **right-click** context menu. If you zoom in on the time series window, you will notice that the

amplitude of light is asymptotically zero at the origin. (Double clicking in zoom mode will return the view to the original size).

3.7 Detector properties

The detectors in use for the example are listed under the *Detectors* node of the *Project browser*. While there is no upper limit on the number of detectors that can be used, it should be noted that the storing and processing of each detector will add overhead both in terms of memory requirements as well as the time the simulation takes. There are 3 classes of detectors:

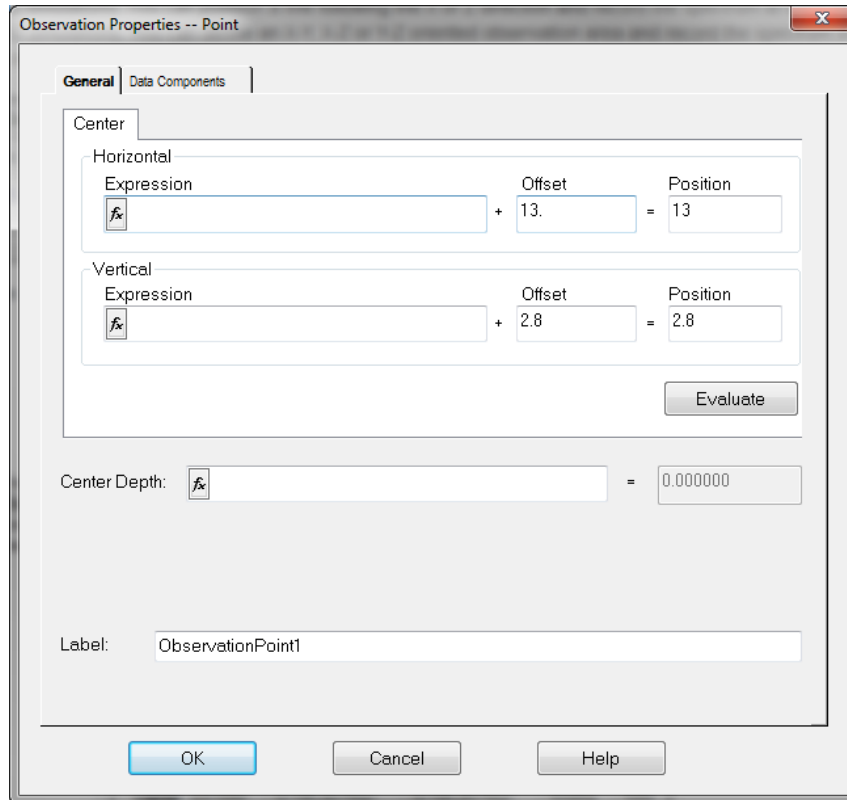
- **Observation Points** - Used to record time varying field data from a single point in the simulation space. Spectral information can be extracted through post-processing and using a Discrete Fourier Transform (DFT) on the time data.
- **Observation Lines (2D)** - Observations are only available in 2D simulations and can be positioned along either the X or Z direction and the spectral information at each point on this line will be stored.
- **Observation Areas (3D)** - Observation planes are only available in 3D simulations and can be positioned along any **XY**, **XZ**, or **YZ** plane the spectral information each point in this area is stored. The result is a 2D graph that presents the DFT amplitudes across the observation area for a given frequency. The *OptiFDTD Analyzer* will allow the user to select what frequency they are looking at. A movie of the time varying amplitude for given components across the observation plane can be recorded during the simulation.

To access the Observation Point located on the right side of the upper waveguide for the ring design, do the following:

- **Double-click** the top-left small green dot in the design

OR

- **Double-click** the *ObservationPoint1* node in the project browser.



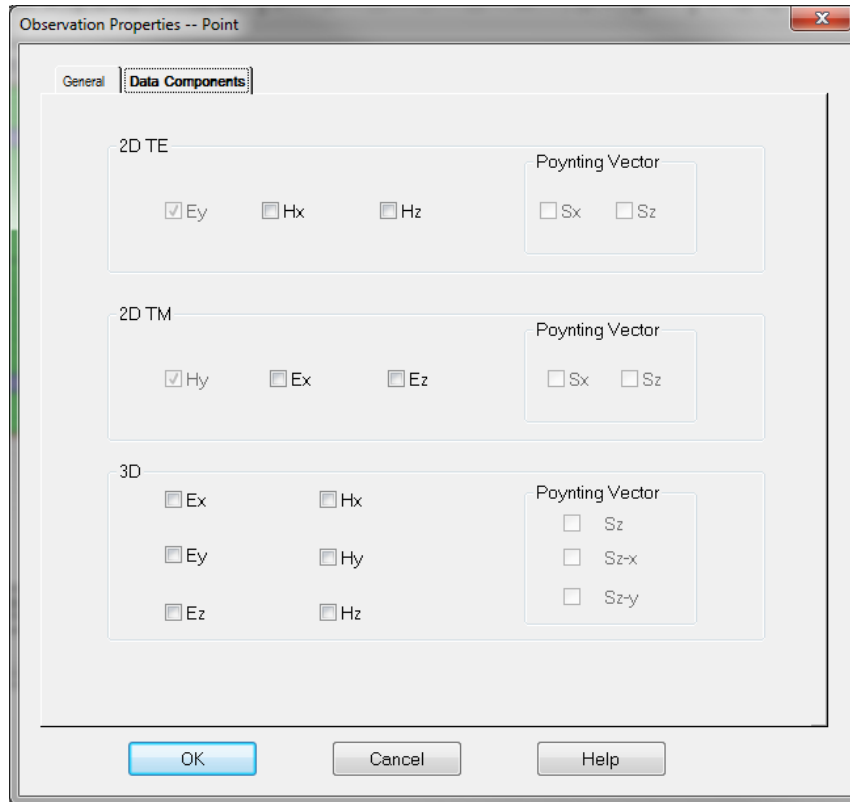
13 Observation Properties – Point dialog

The position of the observation point is defined in the same way as sources. In this example the detector is positioned at coordinates $Z = 13 \text{ um}$ and $X = 2.8 \text{ um}$.

3.7.1 Setting up data components and dynamic visualization of results

All detectors (Observation points, lines and areas) include a *Data Components* tab which allows the user to specify which fields to include (and save) at the various detection locations during the simulation. To include a specific field, check the box next to the field name (Ex, Ey, Hx, Hy, etc). All selected Data Component fields will then be made available for analysis after the completion of the simulation.

It is possible to directly observe, in a dynamic way, the optical field during the simulation run time. This feature, called *Runtime Field View*, provides the ability to access and view the spatial field components for the current simulation time steps. Temporary observers can be specified, but it should be noted that when observing predefined detectors only pre-selected field components will be available for visualization during the simulation.

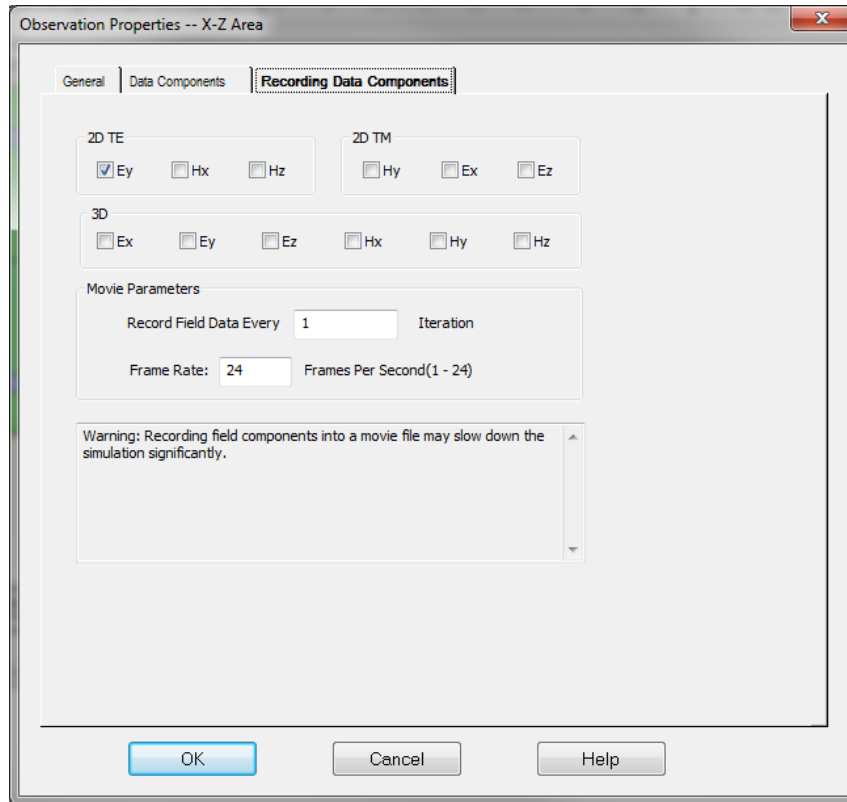


14 Observation Properties Data Components tab

3.7.2 Recording field data from an area detector (movie files)

As previously noted OptiFDTD can save a movie recording of the specified time varying field components for any Observation area. To setup the data components for recording do the following:

- **Double-click** on *Detectors/ObservationArea1* in the *Project browser* (recording can only be setup from Observation areas).
- **Select** the *Recording Data Components* tab
- **Check** the field component that the recording is required. For this 2D TE simulation select the *Ey* field component box.
- **Select** *OK*



15 Recording Data Components

Each selected component will have its own movie file. The filename format for movie files is [`<Name of the layout file>_<Name of the observation area>_<Name of the component>.avi`]. In this example the movie for the Ey component would be stored as “Test_ObservationArea1_Ey.avi”.

- ⚠ Depending on the size of the area detector and the number of the selected field components, recording component data into movies can significantly degrade the efficiency of the simulation and increase the time a simulation takes as well as the amount of resources required. It is necessary to ensure that there is enough free space on the hard disk for storing the movies.

3.8 Running a 2D simulation

In order to run the simulation, OptiFDTD will need to write temporary and result files to disk and as such it is necessary that write access is granted for the location that the project file is saved. For example, saving a project file to the c:\ drive is not advisable as users do not typically have write access. The file must be saved in order to the simulation to complete

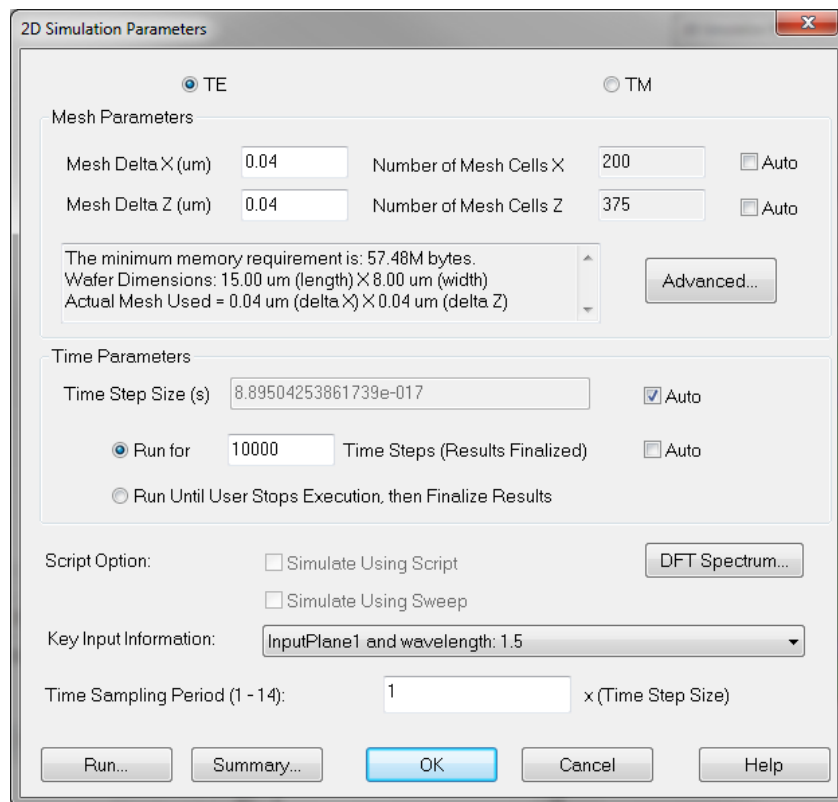
- **Select** *File > Save As...* in the menu bar and choose an appropriate folder (for example My Documents) and filename before **clicking** *Save*.

Setting up the parameters and running the simulation are both accomplished through the *2D Simulation Parameters* dialog.

- **Double-click** on the 2D simulation... node in the project browser.

OR

- **Select** *Simulation > Simulate 2D...*



16 2D Simulation Parameters dialog

The *2D Simulation Parameters* (or 3D) dialogs control all parameters that affect the simulation, such as discretization and boundary conditions:

- *Type of simulation* - The 2D simulation can be setup to analyze the **TE** (transverse electric - Ey, Hx, and Hz) or **TM** (transverse magnetic - Hy, Ex, and Ez) fields. For the purpose of this example the **TE** mode simulation is selected, this also happens to be the default for any 2D simulation.
- *Mesh parameters* - The mesh parameters define the spatial resolution of used to discretize the simulation domain. The mesh delta can be defined separately for the X and Z directions and by default (activated by checking the Auto option) to $\lambda/10$, where λ is the central wavelength of the source, defined by the *Key Input Information* drop down, in the material with the highest refractive index. While this ensures stability in the numerical calculations for the source it does not ensure that the physical dimensions are properly discretized. A general suggestion is to apply the 1/10 rule of thumb to the smallest feature size as well as the wavelength. Convergence testing is also highly encouraged. For the purposes of this simulation the mesh will be a square mesh of size 0.04 microns (X) by 0.04 microns (Z). The total number of associated mesh cells (which will encompass our simulation region) are automatically calculated based on the defined dimensions of the 2D wafer. Further information can be found in the Discretization section of the "Technical Background" manual.
- *Time parameters* - are two controls which set the *Time Step Size* and the number of time steps, labelled *Run for*:
 - *Time Step Size* defines the time-domain resolution of the simulation. It is initially recommended to set this parameter to *Auto* (when Set to *Auto*, the time step size will be set to a level well within the defined limits of the Courant-Friedrichs-Levy (CFL) condition - see the Space and Time Steps section of the "Technical Background" manual.
 - *Run for* is the number of time steps that the simulation will be run for. The *Auto* setting will set this to a value that should normally provide a steady state solution which is based on the time it would take light travelling through the design's highest refractive index material to travel from one corner of the simulation domain to the opposing corner. However depending on the type and complexity of the design (especially for designs which include resonant cavities or rings) additional time steps may be required before reaching a steady state solution. It is thus recommended to run additional simulations with a larger number of time steps and to compare the simulation results to ensure that the design has converged to a stable solution.
 - **NOTE** - When the source selected as the *Key Input Information* source is set to CW, additional steps will be performed (4 wavelength cycles) after the final time step to capture the DFT of the propagating field (this stage is called finalization of results). When the input field source is set to SMGP, the additional finalization step is not necessary and the DFT will be applied to all the time steps in the simulation.
- *DFT Spectrum* - The DFT Spectrum button provides access to the DFT simulation parameter settings. Here the start and end wavelengths and the number of wavelength samples that are to be used for the simulation are set. These settings are important for Observation Lines and Observation Areas as this DFT data is captured during the simulation, note that this means that the specifications for the spectral range cannot be changed in post-processing after the simulation has completed. Observation points are exempt from this limitation as they store the time-series data and as such DFTs can be performed at the users request.
- *Time Sampling Period* - The Time Sampling Period is used to define the desired sampling rate for the Fourier domain analysis of Observation Lines and Observation Areas. By default the time sampling period is set to 1 (recommended setting), meaning that the DFT is calculated at every time step of the simulation. If a value higher than 1 is used, the sampling period is increased (resulting in fewer sampling points or lower frequency resolution for the simulation).

- a. Launch the simulation by **clicking** *Run...*
The Simulator window appears and the simulation is performed
- b. When prompted, **select** Yes to open the Analyzer

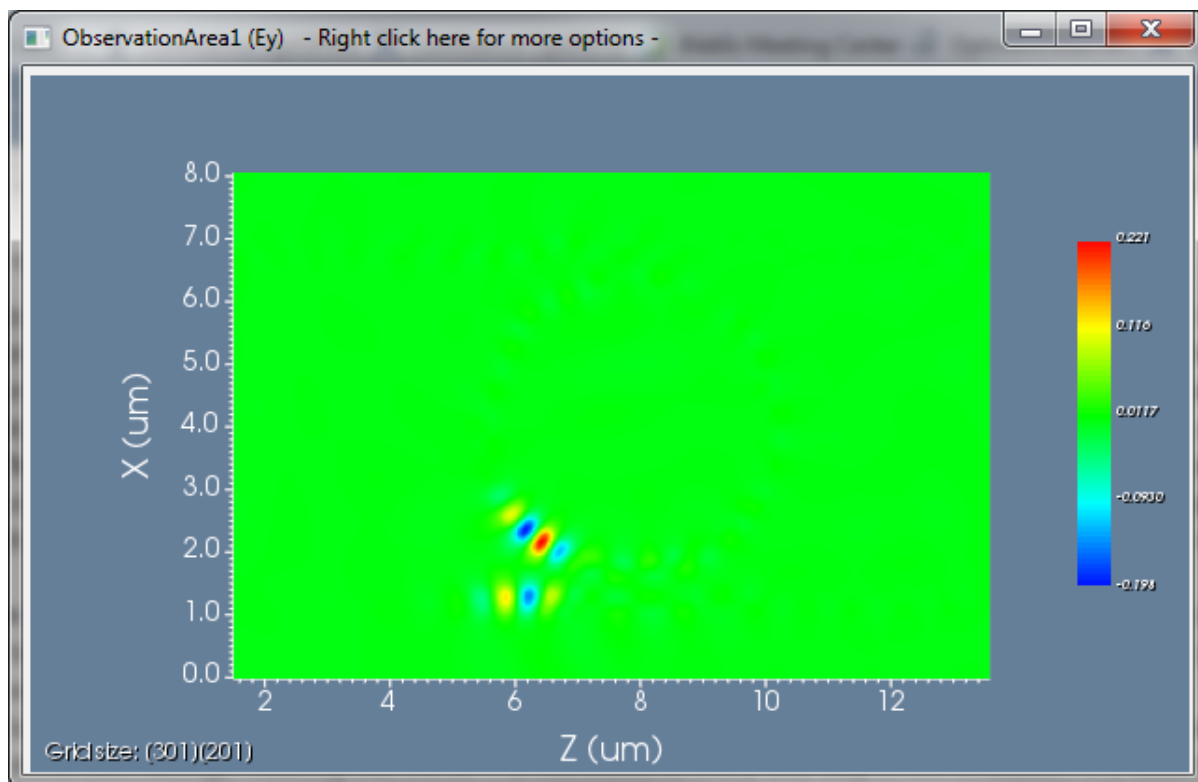
3.8.1 Viewing the simulation real time with the Run Time Viewer

During the simulation the runtime field view can be used to monitor the progress of the simulation. To do so, proceed as follows:

- **Left-click** on *Runtime Field View* on the top menu bar of the Simulator progress dialog box
- **Select** Use a *Pre-defined Observer in Layout*
- **Left-click** on *ObservationArea1* in the tree view
- **Left-click** on *Ey* and select *OK*

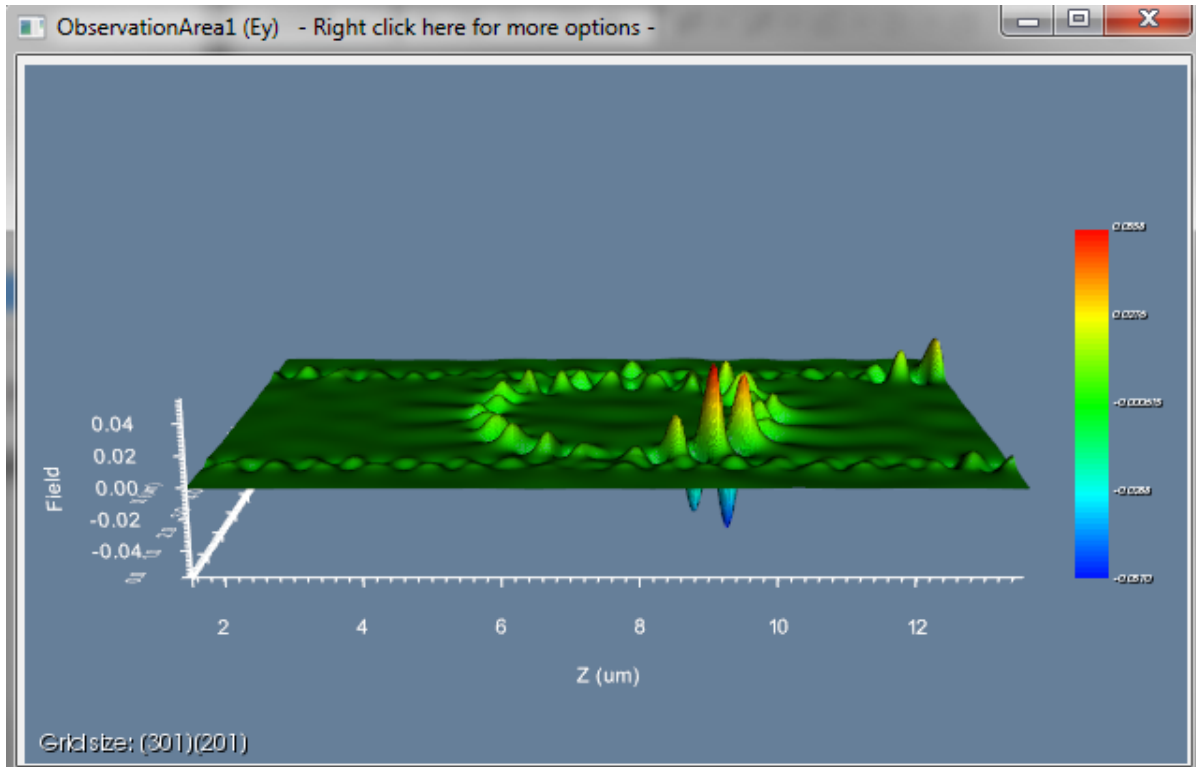
⚠ Please note that the system needs resources and time to process all windows for the *Runtime Field View*, as a result the simulation will be slower while the runtime viewer is being accessed and viewed.

The 2D profile of the mode field propagation will appear as follows:



17 2D Image Map

As the simulation progresses the window will update as the simulation progresses. Simulations can be paused and resumed by **clicking** the *Pause* in the Simulator progress dialog box. The viewing options for the viewer can be changed through the **right-click** context menu within the viewer. For example if *Height Plot* is selected the field will be presented in a 3D format.



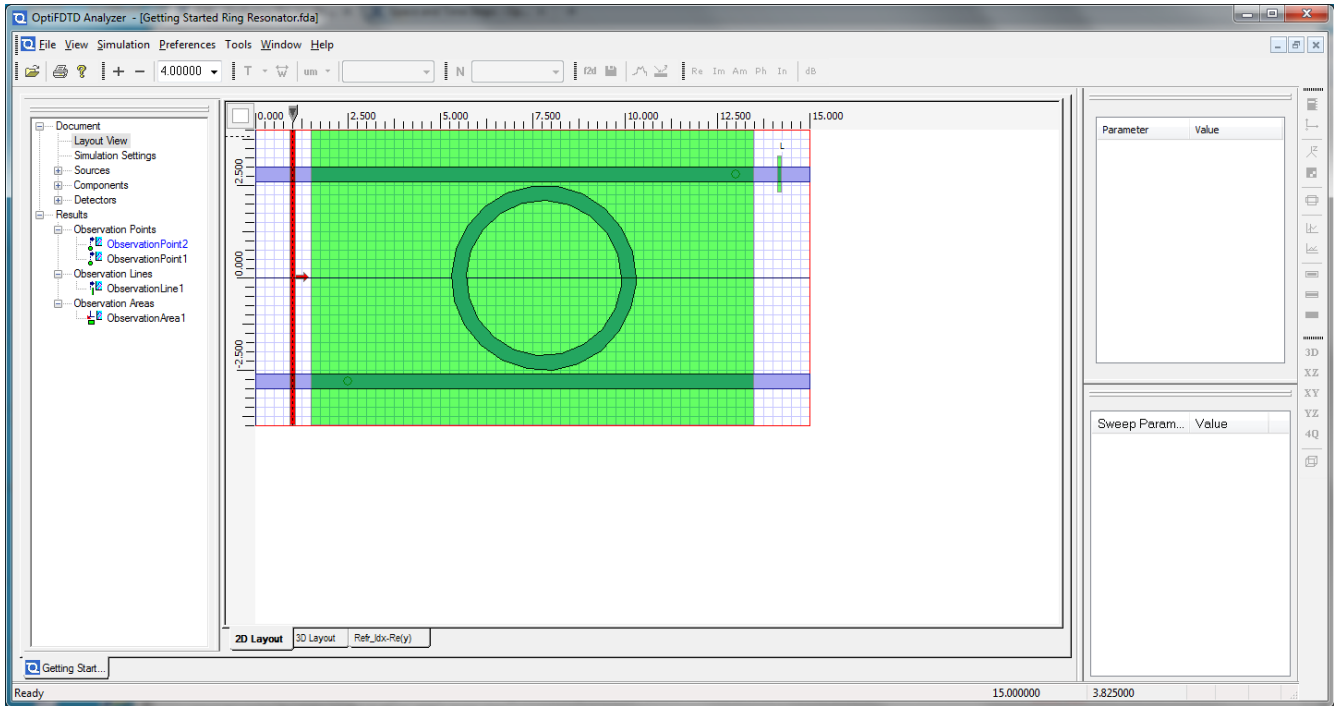
18 Height plot

For further information on the runtime Field View please see "Design Tutor" and Simulation under the help menu.

3.9 Analyzing the results

The data resulting from a simulation is stored in a separate file from the design. The file name will be the same as that of the project and have an *fda* extension. All the simulation results are housed in this file (for further analysis at any time). The analysis of this data is done through *OptiFDTD*

Analyzer. The first tab in the Analyzer window is a read-only copy of the layout. It is provided as a reminder of the structure that was used for the simulation:



19 Analyzer window

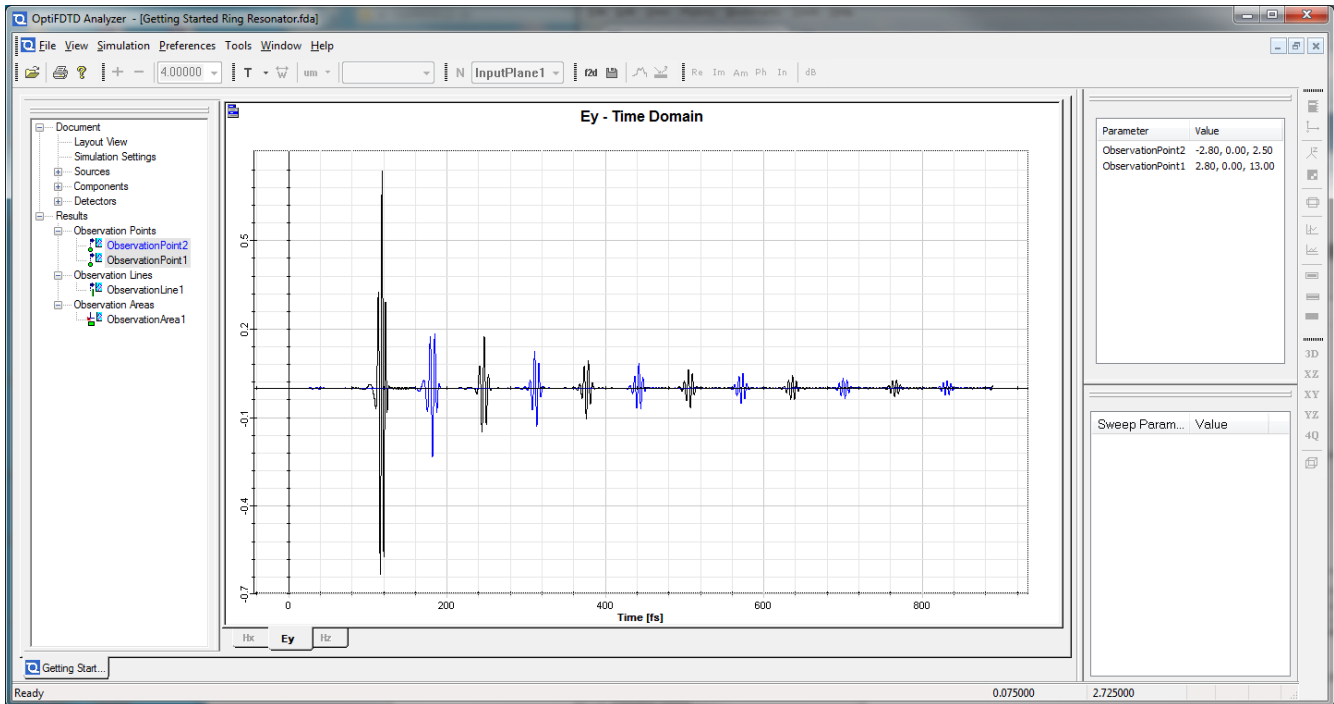
The geometry of the design can be reviewed in the *Layout* or *3D Layout* views tabs. The refractive index distribution can also be seen in the *Refr_Idx* tab.

⚠ If you use an anisotropic material in your simulation, you can change the component displayed in the refractive index view by going to *View > Visualizer Data* and choosing the appropriate component.

The detector data can be accessed by the following action:

- **Click** on desired observation point located in the *Results > Observation Points* node in the *Project Browser*. As this is an observation point the field as a function of time will be displaced in the window. Holding the Shift key while selecting will allow the user to select multiple observation points and have the results to be shown on the same graph.

3.9.1 Observing the time series data



20 Observation Point Analysis dialog

It can be seen that the output as measured at the observation points located at the output ports of the ring resonator are decaying. The source was a pulse and this decay is the light slowly propagating through the resonant structure.

3.9.2 Extracting spectral information

Using the **Discrete Fourier Transform (DFT)**, it is possible to extract the spectrum from the time dependent field. To do this, follow these steps:

- **Select** the pull down menu arrow next to the **T** icon on the Frequency tool bar:



- **Left-click** to select the menu item "Frequency domain". The tool bar will update as follows:

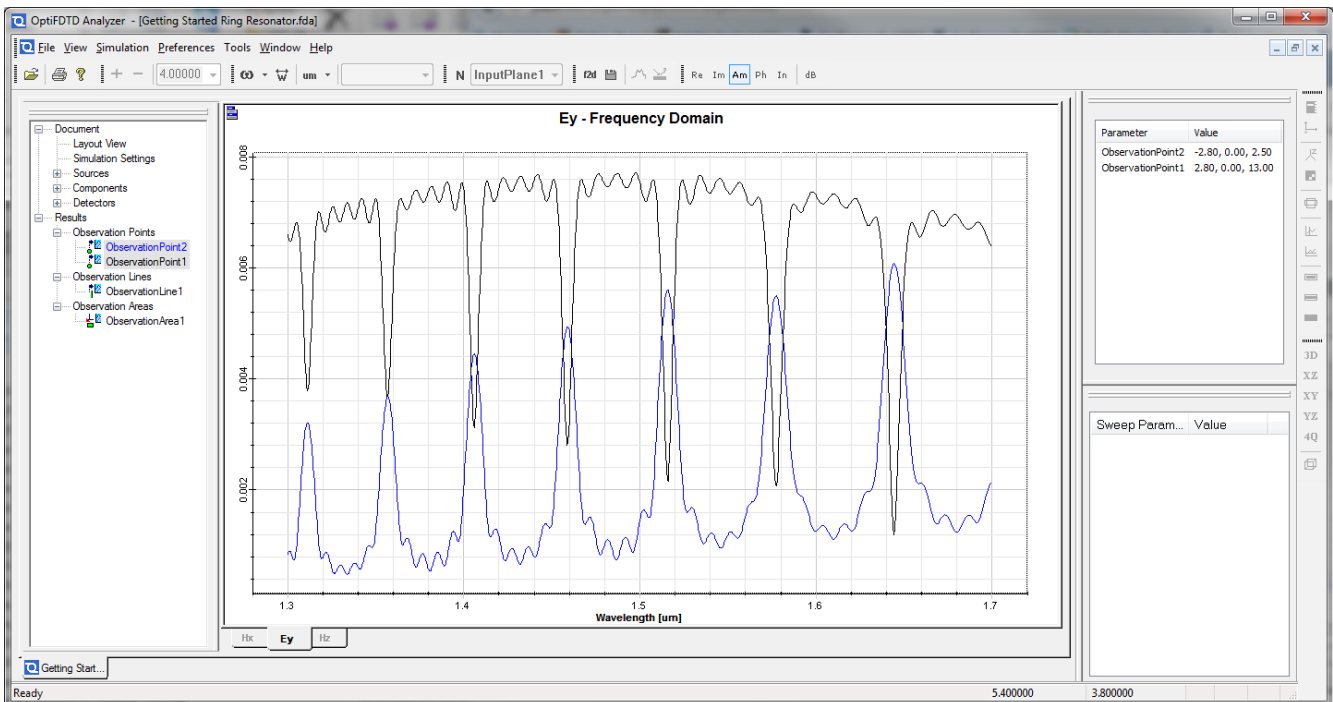
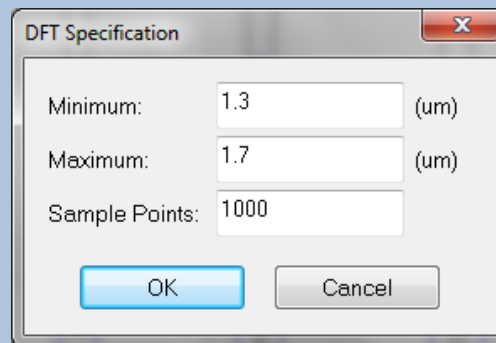


Left-click

theDFT Specification

icon,

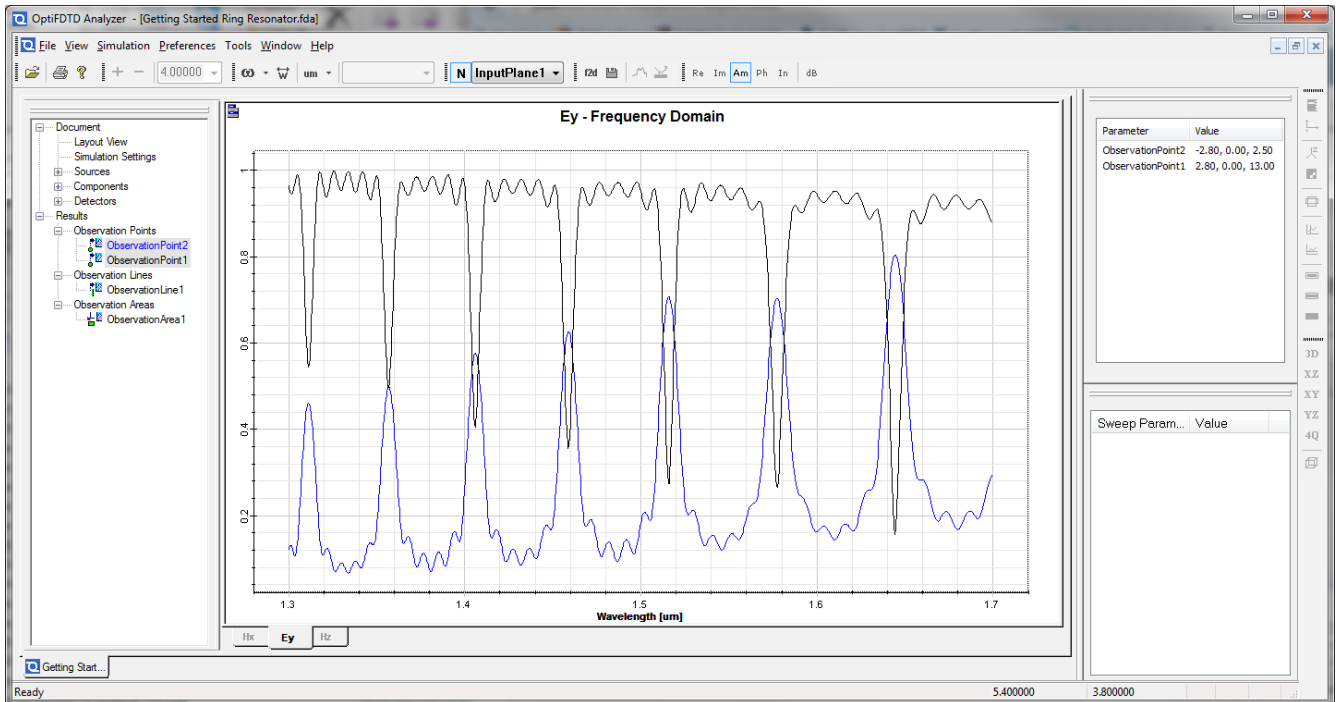
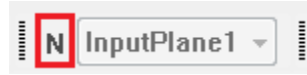
, launching the dialog to control the bounds of the spectrum as well as the number of sample points. **Set** the *Sample Points* to 1000.



21 DFT spectrum of the point detectors

The characteristic spectrum of a ring resonator is now visible. The dips and the peaks of the two curves represent the resonant wavelengths of the ring design which correspond to wavelengths where the waveguide mode couples to the mode in the ring resonator.


A user may wish to normalize the results to a source. this can be done by selecting the "N" button on the Normalization tool bar. Additionally the source that is used for the normalization can be selected.



22 DFT spectrum of the point detectors (normalized)

3.9.3 Exporting the data to file

It may be desirable to export the data results for further analysis or use in external applications. To export the *DFT* spectrum for *ObservationPoint2*, it is necessary to do the following:

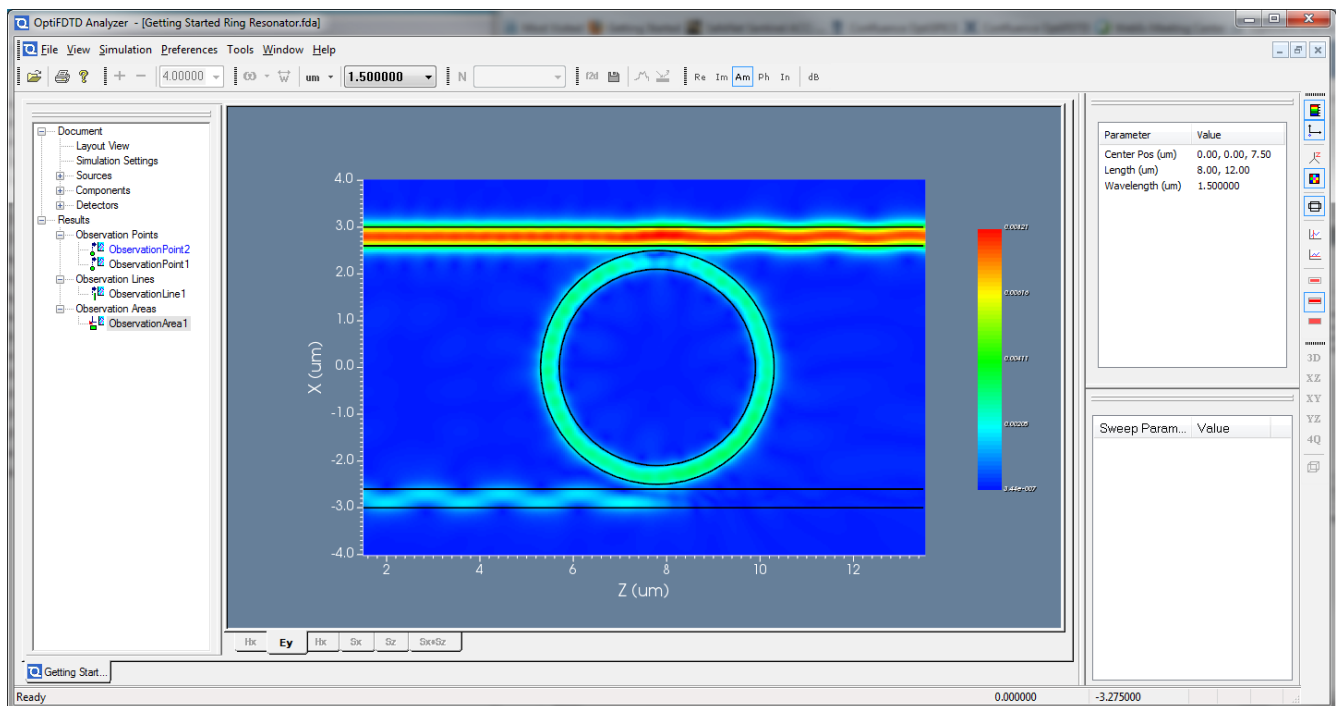
1. **Click** on the blue-grey folder icon, , in the top left corner of the 2D graph window
2. **Select** *Export Data...* from the pull down menu
3. In the *Data Export* dialog box, **select** the *ObservationPoint2* curve from the *Curve* combo box
4. **Click** *Export Curve...*
5. In the *Save As* dialog box, **select** a path and a file name.
6. **Click** *Save*

3.9.4 Observing the 2D observation area results

To view results from an Observation area detector, do the following:

- **Click** on desired observation area located in the *Results > Observation Areas* node in the *Project Browser*. For the purpose of this example select *ObservationArea1*.

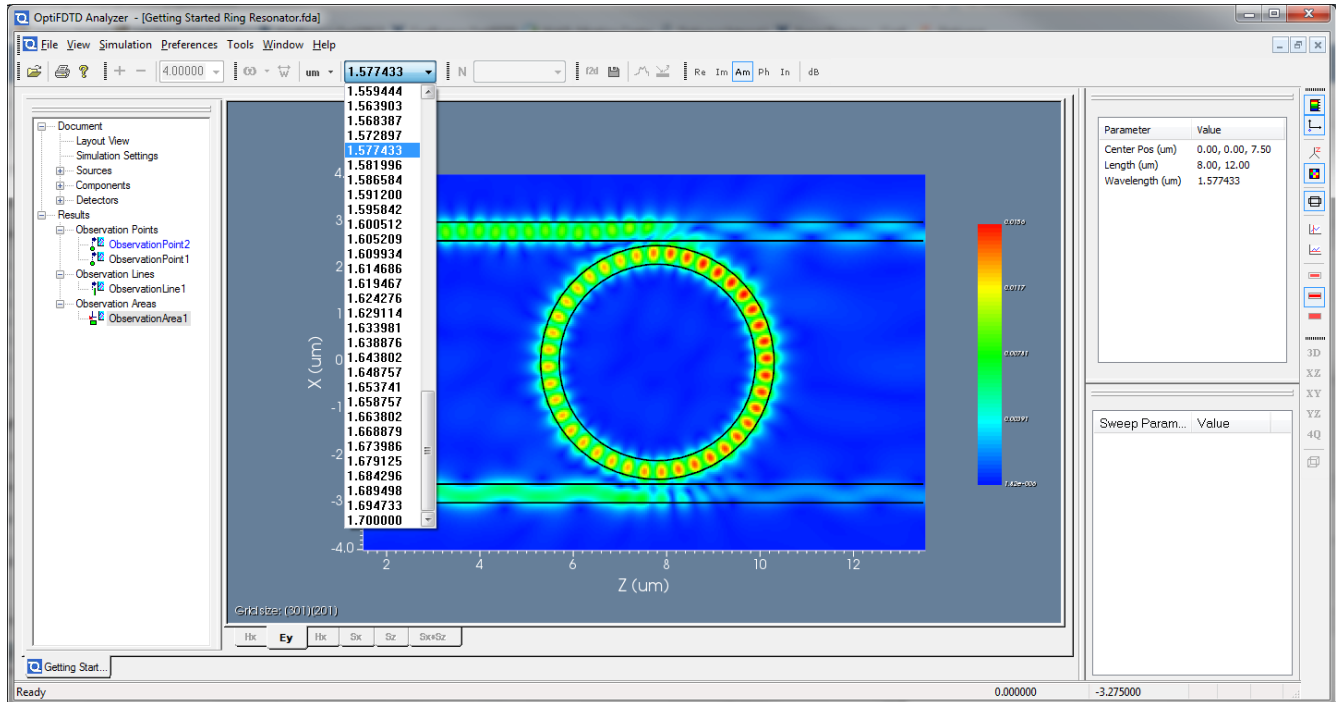
The field propagation data for the Ey component will be displayed (by default the central wavelength setting for the SMGP is shown - 1.5 μm in our case). As this was a 2D TE simulation there are only 3 field components (Ey, Hx, and Hz) and only the Ey component is available as the was the only component selected for this observation area in the design. Additionally, the components of the Poynting vector will be available if they were selected by the user during the design stage.



23 Observation Area Results

As the Observation area provides information in the spectral domain and the source was multi-wavelength (a Gaussian pulse) it is possible to view the field information for multiple wavelength sampling points based on the DFT settings that were set in the simulation parameters. To change the wavelength sampling point, **left-click** on the pull-down menu of the frequency tool bar and scroll to select the desired wavelength, for example 1.577433 μm . This wavelength is chose as it is close to one of the resonant peaks observed from the DFT analysis of the point detectors. For this

wavelength the observation area shows that the majority of the waveguide mode field is coupled into the ring and exits from the lower left waveguide:



24 Observation Area Results (Resonance wavelength)

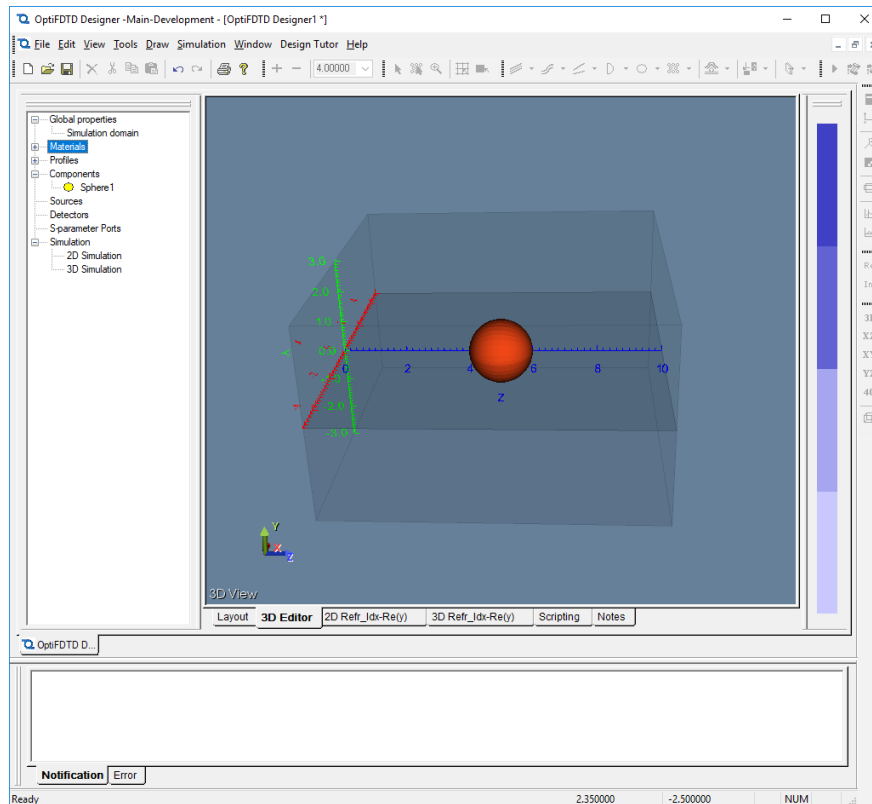
3.9.5 Observing the 2D observation area results (movie)

Recall that the observation area was set to record the time varying E_y component. To view a movie recording of an Observation area detector:


1. **Open** the folder where "Getting Started Ring Resonator.fda" has been saved, the default location is the same folder that stores the main project file
2. **Find** the file named "Getting Started Ring Resonator_ObservationArea1_Ey.avi"
3. **Right-click** on the file and select "Open with/Windows Media Player"
A movie showing the progression of the time series field propagation (for E_y) will be shown

3.10 Working with the 3D editor

When working on a 3D simulation, you can view the 2D X-Z view from the Layout Editor (which is displayed by default) or you can use the 3D Editor (which is better suited to view and select objects in a 3D environment).



25 The 3D editor

The 3D editor can be accessed by **selecting** the *3D Editor* tab () from the tabs that run along the bottom left hand corner of the viewer window. In this view the full 3D simulation domain is displayed along with all of the objects that make up the design. The coordinate system axes are indicated in the bottom left corner indication the orientation of the current view in 3D space.

3.10.1 Available 3D Views

The 3D editor toolbar comes with a range of viewing options:

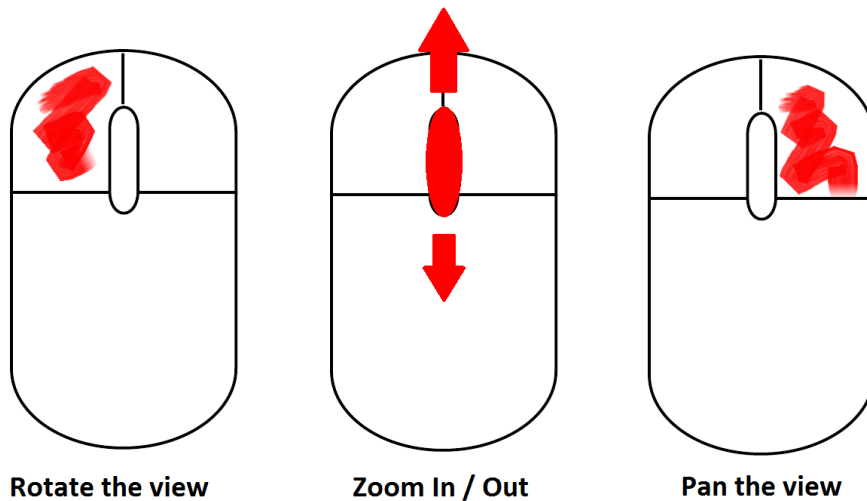


26

- 3D view
- Orthographic projections
 - **XZ** (Same view as *Layout*)
 - **XY**
 - **YZ**
- Quadrant view, **4Q**, showing the 3D view and the three orthographic projections in the 4 quadrants
- Wireframe view toggles whether objects are presented using facets or wireframes. This view mode is available in any of the 3D views.

3.10.2 Mouse Navigation

The mouse can be used to orient the view in the 3D viewer:



27 Mouse Operations

- To rotate, **hold the left mouse button while moving** the mouse to the desired direction. the axes of rotation can be changed by **holding down the CTRL key prior to and during a rotation.**
- Zooming in the 3D viewer is done by **scrolling the wheel forward to zoom in**, and **scrolling it backward to zoom out.**

- If lateral or vertical movement (panning) is necessary this is accomplished by **holding the right mouse button while moving** the mouse in the desired direction.

3.10.3 Selecting objects in the 3D view

In the 3D View, an individual object can be selected directly by left-clicking on it. It will become red and a grey box will be displayed around its name in the project browser. In the OptiFDTD Designer, double-clicking an object in the Project Browser will pop-up its properties dialog box, and its geometry can be modified more precisely, or a new material can be assigned to it.