

OptiFDTD

Tutorials

Finite-Difference Time-Domain

OptiFDTD Version 16.0.1 for Microsoft Windows® 10 64-bit



Table of Contents

1	Using OptiFDTD Designer	4
1.1	Introduction	4
2	TE planar waveguide coupler splitter	7
2.1	Definition of the simulated device	7
2.2	Design of the simulation geometry	7
2.3	Configuration of the simulation	25
2.4	Simulation of the designed Coupler	28
2.5	Analysis of the results	28
3	Photonic Crystal Band Diagram	33

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 Using OptiFDTD Designer

1.1 Introduction

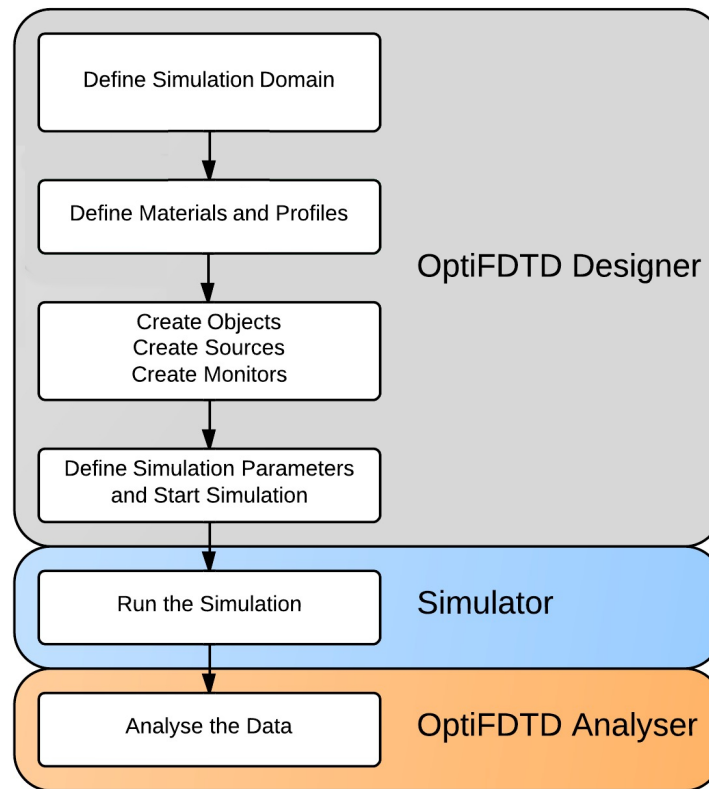
In the tutorials, you will learn the basics of working with OptiFDTD Designer to create your simulation objects and define their properties. We will define some basic geometry and learn how to define sources and detectors, run a simulation and review the results. OptiFDTD Designer is one of the 4 components of OptiFDTD, as shown below.

OptiFDTD workflow

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

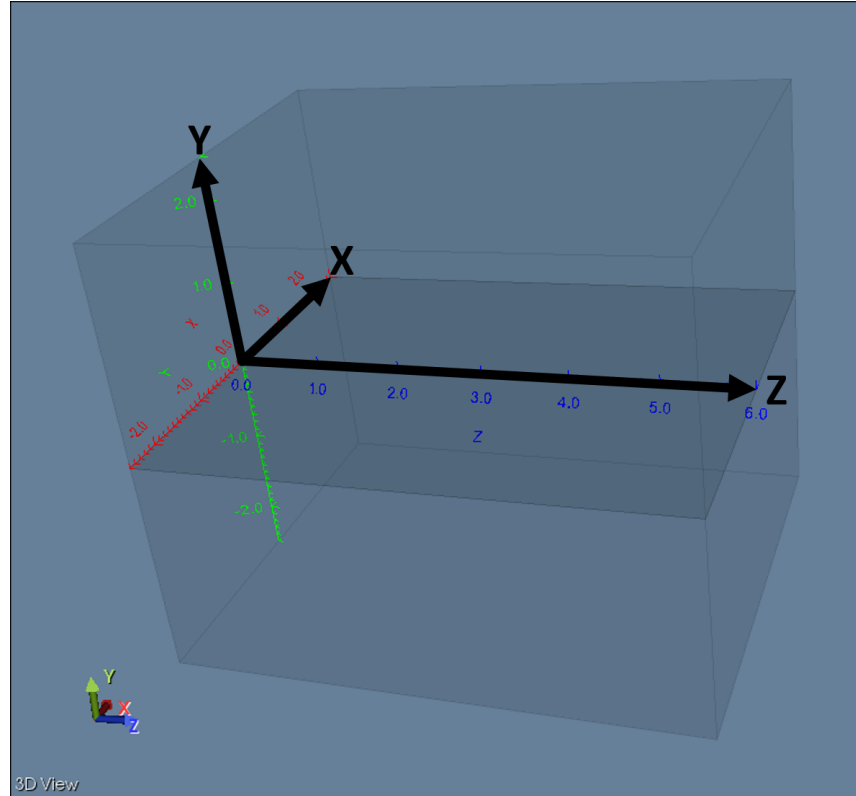
- **OptiFDTD Designer** - The main OptiFDTD program. From here, you can create new designs, set simulation parameters, write scripts and start simulations. Data is saved in a project file with the extension **.fdt**.
- **OptiFDTD Simulator** - Opens automatically when a simulation is run from Designer 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.



OptiFDTD coordinates 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:

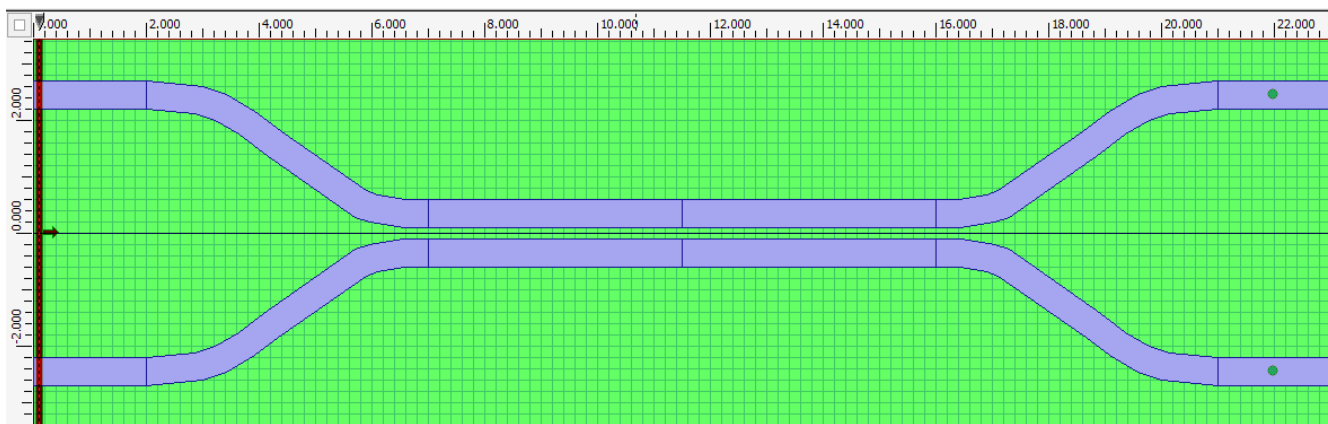
- The **XZ** plane is the primary view in the Layout of the OptiFDTD Designer
- **X** is referred to as the width or vertical component
- **Y** is referred to as the depth
- **Z** is referred to as the length or horizontal component
- The 2D and 3D simulation domains are referred to as 2D and 3D wafers 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.

2 TE planar waveguide coupler splitter

2.1 Definition of the simulated device

The geometry we want to define in this tutorial is shown below. It consists in a simple planar TE splitter/ coupler. To simplify our model, the waveguide material is a dielectric of refractive index 2.0 and the surrounding material is set to air. The input wavelength is 1.55 μm . We will run a 2D simulation in this tutorial.



1 geometry to be defined in OptiFDTD Designer

2.2 Design of the simulation geometry

For reference the structure has been created and saved in the in the OptiFDTD samples directory. The default location of the design file is:

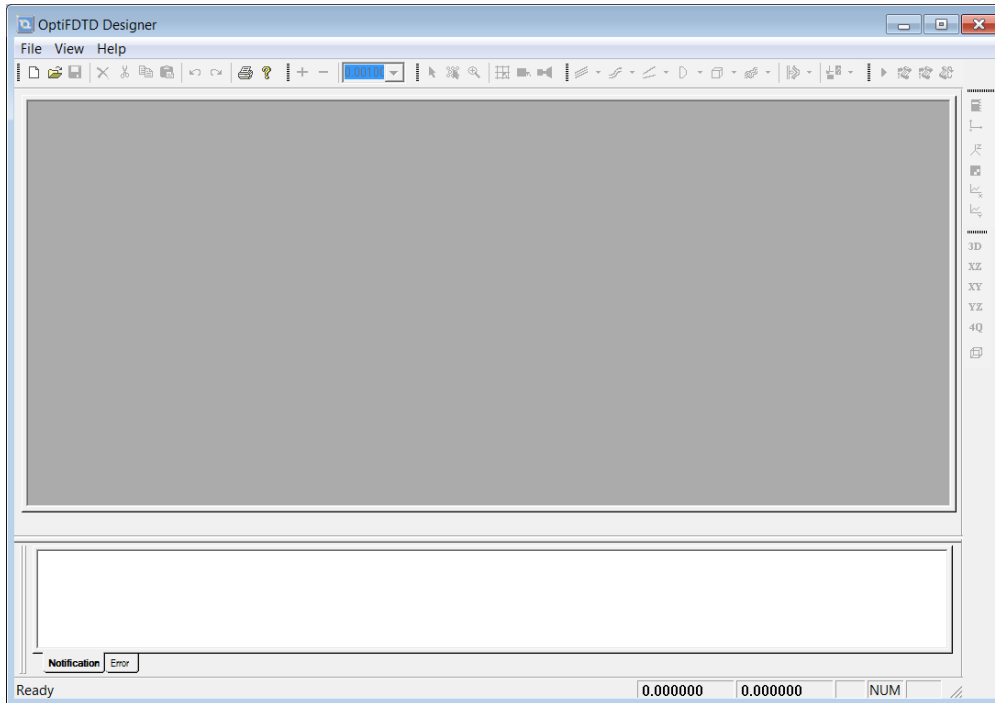
```
c:\Users\yourname\My Documents\OptiFDTD 16
samples\2D_samples\photonic_components\TE_coupler.fdt
```

2.2.1 Starting OptiFDTD Designer

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.




2 Main OptiFDTD Designer application window

2.2.2 Creating a new project

If a new design is required, the user would like to create the structure from the beginning a new project can be created by

- **Select File > New...**

OR

- **Click** on the New Project icon  in the toolbar


2.2.3 Setting up the simulation domain properties

Prior to any structures being created the simulation domain must be defined. This is accomplished by **double-clicking** on *Simulation domain* in the project browser, the *Wafer properties* window appears.


- In the Wafer Dimensions tab: Length (μm) = 23 and Width (μm) = 6.9
- In the 2D Wafer properties tab:
 - Verify that the selected material is Air. This value defines the background refractive index for 2D simulations.
- **Click** *OK* to close the dialog box.

2.2.4 Defining material and profiles properties

The next step is to define the materials to be used in the simulation and the profiles (waveguide cross-sections) of the waveguides required for the we will design.

- **Double-click** on the *Air* material under the *Materials* heading within the *Project Browser*. The refractive index of 1 is automatically set for the Air material. This is the default material in OptiFDTD.
- **Close** the *Air* material dialog box.
- **Right-click** on the *Materials* heading entry in the *Project Browser*.
- **Select** *Add Dielectric Material* from the contextual menu that appears.
- Set the following values:
 - Name: Silicon nitride
 - Refractive index (Re): 2.0
 - Refractive index (Im): 0.0
- **Click** *Store* () to save the material.

When creating a waveguide within OptiFDTD it is necessary to define the waveguide cross-section (Profile) and the waveguide path separately. To create the profile

- **Right-click** on the *Profiles* entry in the *Project Browser*
- **Select** *Add Channel Profile* from the contextual menu that appears.
- Set the following values
 - Profile name: Silicon nitride channel
 - 2D profile definition: Silicon nitride (this is a drop down menu containing all currently defined materials).
- **Click** *Store* () to save the material.

⚠ As this is a 2D simulation, the profile will simply use the material that has been defined for the profile and extend it to infinity in both +/- y directions.

2.2.5 Creating the waveguide geometry

As previously explained, waveguides are defined through their cross-section profile, already created, and the path. The geometry for this structure consists of 2 parallel rectangular waveguides brought to a lateral coupling distance (200 nm in this case) by S-shapes sections. Start by creating the input section of the top most waveguide:

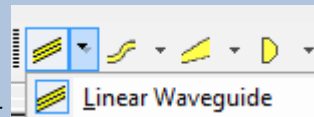
2.2.5.1 Drawing the input waveguide

- **Select** *Draw > Simple Waveguides > Linear Waveguide*

OR

- **Select** the *Linear Waveguide icon* in the design toolbar

The mouse pointer is replaced by a cross-hair



- **Select** *Draw > Simple Waveguides > Linear Waveguide*

OR

- Select the *Linear Waveguide icon* in the design toolbar

- The mouse pointer is replaced by a cross-hair and the path is drawn by holding the left mouse button at the start position and dragging the mouse to the destination position. Start at $Z = 0\mu m$ and $X = 2.45\mu m$ and extend to the Z direction for $2\mu m$. This position can be modified from the waveguide property window for accurate positioning.

⚠ Recall that in the layout view for OptiFDTD the Z direction is the horizontal and the X direction is the vertical.

To access the properties of the waveguide, **double-click** on it in the *layout view* or **double-click** the object named *Linear1* in the *Project Browser*. The Linear waveguide properties window will appear.

Linear Waveguide Properties: Input ✕

Start End

Horizontal

Expression: fx + Offset: 0. = Position: 0

Vertical

Expression: fx + Offset: 2.45 = Position: 2.45

Evaluate

OK

Apply

Cancel

Help

Channel Thickness Tapering

Use Default (Channel:None) Taper: Default (None)

Start: fx 1.000000 ↓ End: fx 1.000000 ↓

Width: fx 0.5

Depth: fx 0

Label: Input

Profile: WG_Channel_Example ▼ ...

3 Linear waveguide properties

Change the base values (bottom of the dialog) to:

field	value
Channel Thickness Tapering	Check "Use Default (Channel None)"
Width	0.5
Depth	0

field	value
Label	Input
Profile	Silicon nitride channel

In the *Start* tab:

field	value
Horizontal, Offset	0
Vertical, Offset	2.45

In the *End* tab:

field	value
Horizontal, Offset	2
Vertical, Offset	2.45

Click *OK* to save the changes.

A straight waveguide has now been defined named *Input*, having a 0.5 μm width (X dimension) and consisting of the profile that was defined earlier. The *Start* and *End* settings define the beginning and end of the waveguide.

⚠ In OptiFDTD, the start and end positions of waveguides are defined in the middle of the edge face of the waveguide as indicated by the green dots when the waveguide is selected.

2.2.5.2 Drawing the S-bend section

To define the S-bend section, it is necessary to use a special type of waveguide called *S-bend Sine waveguide*. Other types are available (such as cosine or arc), but sine provides the best curvature for this design.

- To snap the waveguide to the previously designed waveguide section, **select** the *Snap to Waveguide* icon

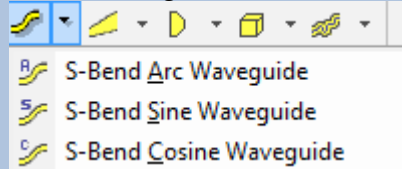


in the *Tools* toolbox.

- Select** *Draw > S-bend Waveguides > S-Bend Sine Waveguide*

OR

- Select** the *S-Bend Sine Waveguide icon* in the design toolbar



- The mouse pointer is replaced by a cross-hair
- Draw** a section of waveguide extending from the end of the last waveguide to an approximate position of ($Z = 7 \text{ } \mu\text{m}$, $Z = 0.35 \text{ } \mu\text{m}$).
- Double-click** on the new waveguide in the layout view

OR

- Double-click** the object named *SBendSin1* in the *Project Browser*. The S-bend Sine waveguide properties window will appear.

S-Bend Sine Properties: SBendSin1 ✕

Start End

Horizontal

Expression: fx + 0.7 = 0.7

Vertical

Expression: fx + 7.1 = 7.1

Evaluate

Enable Orientation Angle

Angle: 0.

Channel Thickness Tapering

Use Default (Channel:None) Taper: Default (None)

Start: fx 1.000000 ↓ End: fx 1.000000 ↓

Width: fx 1.0

Depth: fx 0

Label: SBendSin1

Profile: WG_Channel_Example v ...

OK
Apply
Cancel
Help

4 S-bend Sine waveguide properties

Change the base values (bottom of the dialog) to:

field	value
Enable Orientation Angle	Unchecked
Channel Thickness Tapering	Check "Use Default (Channel None)"
Width	0.5

field	value
Depth	0
Label	SbendIn
Profile	Silicon nitride channel

In the *Start* tab:

field	value
Horizontal, offset	2
Vertical, offset	2.45

In the *End* tab:

field	value
Horizontal, offset	7
Vertical, offset	0.35

Click *OK* to save the changes.

2.2.5.3 Drawing the coupling section

Using the same procedure as the input section above, draw a linear waveguide snapped to the end of the S-bend section.

Change the base values (bottom of the dialog) to:

field	value
Channel Thickness Tapering	Check "Use Default (Channel None)"
Width	0.5

field	value
Depth	0
Label	CouplingLeft
Profile	Silicon nitride channel

In the *Start* tab:

field	value
Horizontal, offset	7
Vertical, offset	0.35

In the *End* tab:

field	value
Horizontal, offset	11.5
Vertical, offset	0.35

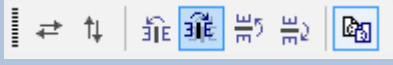
and save the changes.

At this point the left portion of the upper waveguide is defined. This can be mirrored onto the right to complete this waveguide.

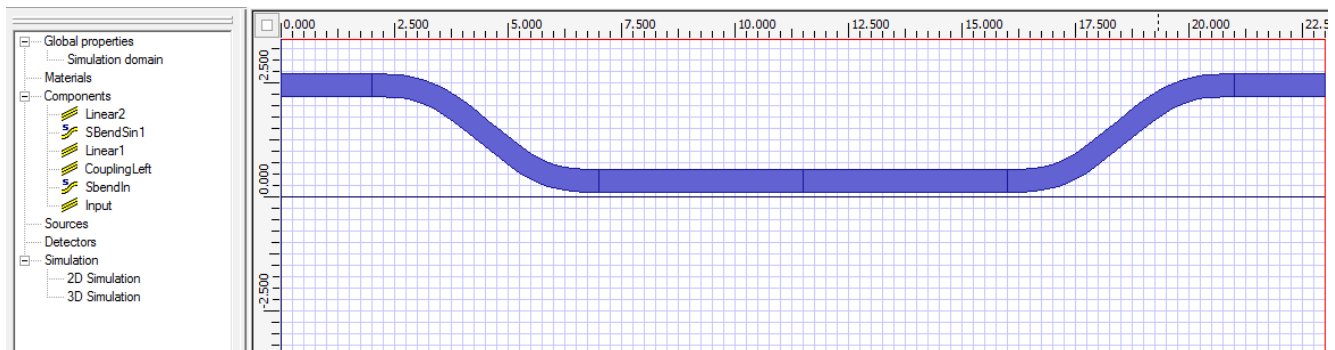
2.2.5.4 Mirroring the design

OptiFDTD provides that ability to mirror a design both horizontally and vertically in order to simplify the creation of complete structures.

- **Select** each of the three waveguides (Input, SbandIn, Coupling) that are present while **holding** the *Shift* key.

- **Click** on the *Mirror right* icon  in the *flip and mirror* toolbox.

The layout should now look like the figure below.



5 Top waveguide after mirroring

2.2.5.5 Creating the bottom waveguide

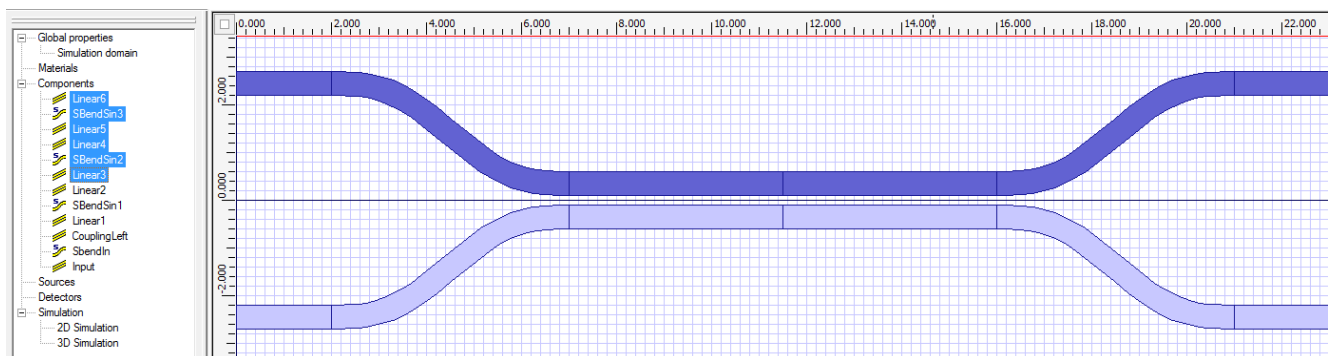
The same process can be repeated to complete the bottom waveguide with the exception that this time, after selecting all of the waveguides, **click** on the *Mirror to Bottom* icon



The newly created waveguides will be mirrored, but the coupling sections are on top of each other. This will need to be corrected and it can be done using the *Move Group* operation.

1. **Select** all newly-created waveguides (in the image below they are named *Linear6*, *SBendSin3*, *Linear5*, *Linear4*, *SBendSin2* and *Linear3*)
2. **Select** *Edit > Move Group*
3. In the Move Group dialog, **enter** a vertical *Group Origin Coordinates* of $-1.4\mu\text{m}$
4. **Click OK** to save the settings


The final result should be the following:



6 Both waveguide after mirroring

2.2.6 Adding a light source

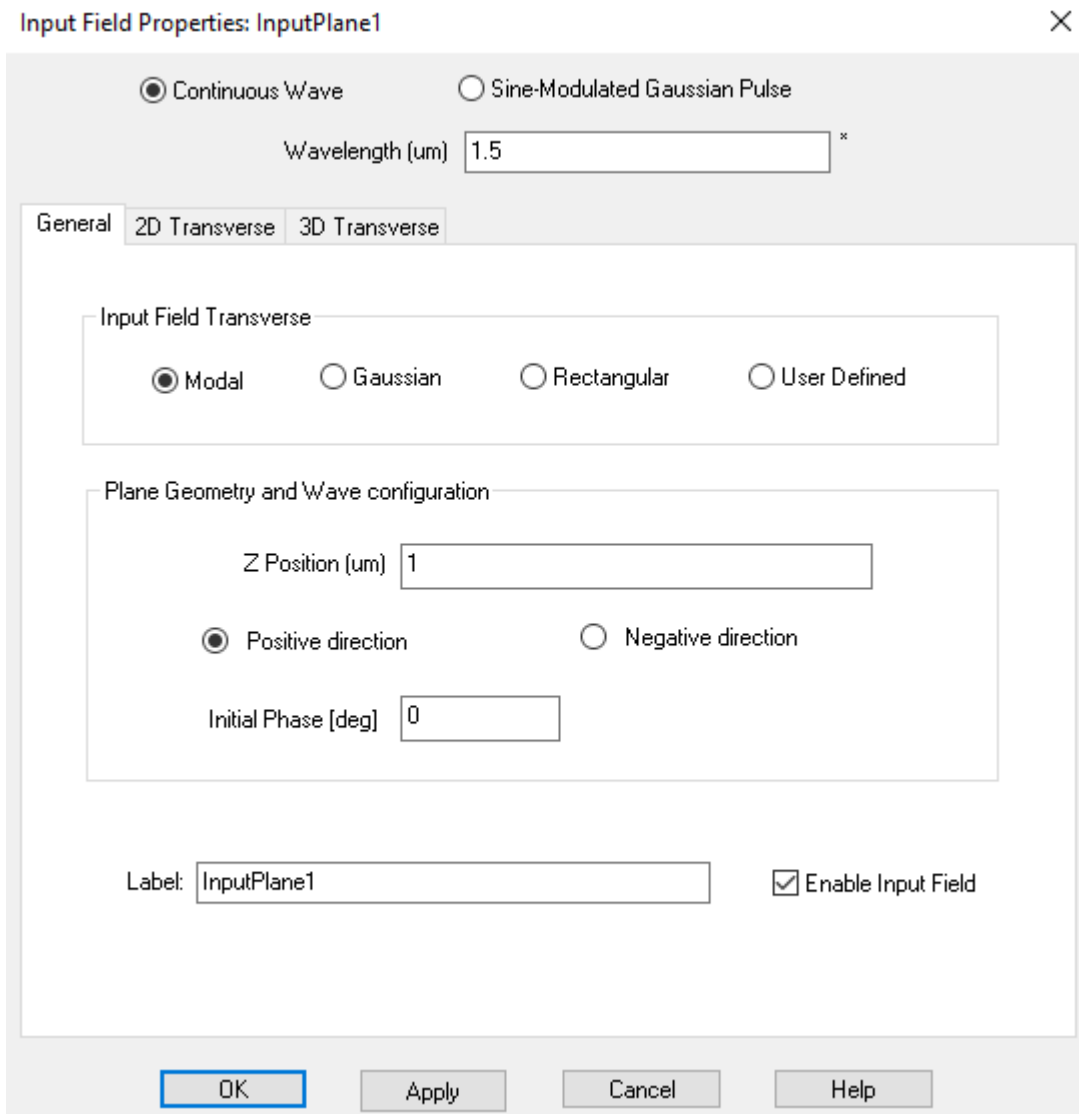
Since the design is based on waveguides the optimum source would be a modal light source placed on the top-left waveguide section (the *Input* waveguide). The modal light source will use the OptiMode mode solver on a waveguide cross-section to determine the distribution of light in the waveguide. The position of this modal source is defined through the creation of an input plane.

 Input planes in 2D OptiFDTD can be either emitting light in the Z direction (the horizontal axis on screen) or in the X direction (the vertical axis).

To define an input plane

- **Select** *Draw-> Z-direction Input Plane*
- The cursor changes into a cross-hair
- **Click** somewhere on the left of the design layout area (where the waveguides are straight)

A red line with an arrow pointing to the right (positive Z-direction) will appear in the layout view. The properties of this light source can be edited by **double-clicking** either the red line in the layout or the item *InputPlane1* under *Sources* in the project browser.



7 Input Field properties dialog

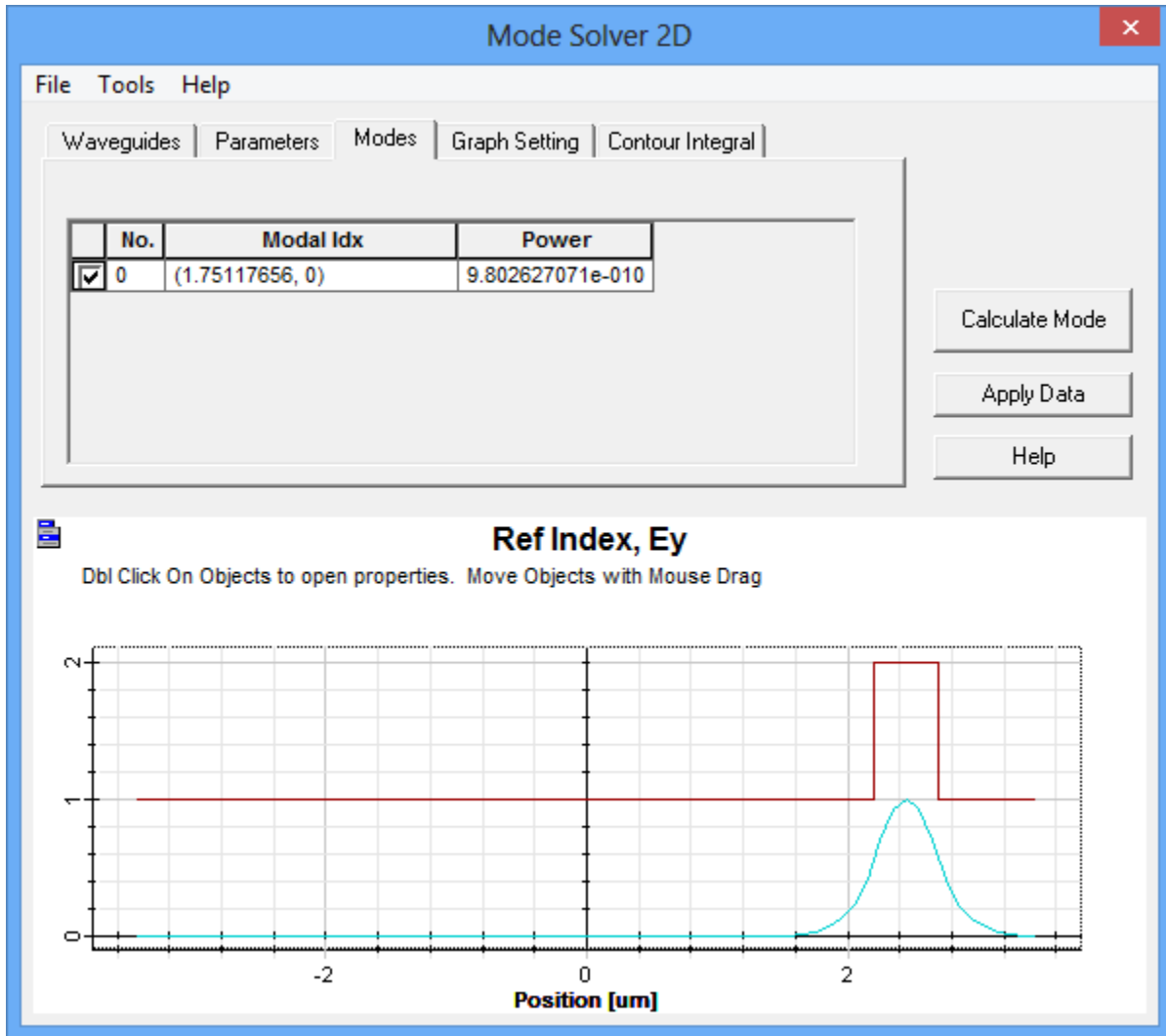
For the purpose of this example the source will be a continuous wave (CW) single-wavelength light source at 1.55 μm . The direction of the emitted light is set propagating along Z in the positive direction (to the right in the layout) and the initial phase is set to 0. Note that you can create multiple light sources and enable or disable any of them using this dialog box. To setup this source change the following values:

In the *General* tab:

field	value
Continuous Wave	Selected
Wavelength (um)	1.5
Input Field Transverse	Modal
Z position (um)	1
positive direction	Selected
Initial phase	0
Enable Input Field	Checked

The modal distribution still needs to be set. This is done by

1. **Select** the *2D transverse* tab to select the 2D modal transverse field properties.
2. **Click** the *Find Modes...* button. This will open the 2D mode solver window
3. In the *Waveguides* tab, there will be 2 entries corresponding to the 2 waveguides that intersect the plane.
4. **Select** the last entry in the list (labelled *Input*) to select the input waveguide. Notice the refractive index distribution on the graph at the bottom which represents the refractive index profile of the selected waveguide. The position (um) axis corresponds to the position along the input plane. The vertical axis corresponds to the refractive index.
5. Within the *Parameters* tab. Verify that the simulation type is set to TE. It will indicate to the mode solver the mode polarization that is required.
6. **Click** on the *Calculate Mode* button on the right.
7. In the *Modes* tab, **select** the calculated mode. The resulting mode is displayed in the graph in blue. To apply the calculated mode to the input field, **click** the *Apply Data* button on the right.



8 2D mode solver dialog

2.2.7 Adding detectors

To complete the design detectors are required to record the field and provide data for post-processing.

2.2.7.1 Observation Points

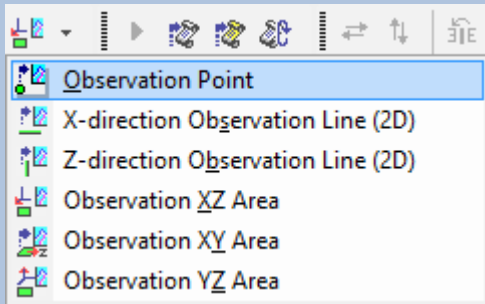
The first type of detector to add to the design is the *Observation Point*. These detectors record the intensity of the fields over time at a single specified point in space. This is particularly useful to

ensure that the input wave has actually reached the end of the simulation grid before the simulation is stopped. If spectral information is require a Fourier analysis can be done.

For these reasons 2 observation points are required in the design, one in each output branch of the coupler.

- **Select** Draw -> Observation point

OR



- **Select** the *Observation Point* icon in the design toolbar
- The mouse pointer is replaced by a cross-hair
- **Click** in the top output branch of the coupler in the layout editor.
- **Edit** the properties of the *Observation Point* by either **double-clicking** the green dot in the layout editor or the item named ObservationPoint1 in the *Project Browser*.

The properties that need to be changed are the following:

In the *General* tab:

field	value
Horizontal offset (um)	22
Vertical offset (um)	2.45
Label	Output1

Leave the defaults in the *Data Components* tab. This tab is used to select which fields are collected during simulation.

Use the same process for the bottom output branch, using the following values

field	value
Horizontal offset (um)	22
Vertical offset (um)	-2.45
Label	Output2

2.2.7.2 Observation Areas

In addition to the observation points it will be necessary to define an **observation area** to the simulation. The area is a rectangle used to collect steady-state discrete Fourier transform (DFT) or spectral information on the fields over its surface. Time varying movies can also be generated.

- **Select** *Draw > XZ Observation Area*

OR

- **Select** the *XZ Observation Area icon* in the design toolbar
- The mouse pointer is replaced by a cross-hair
- **Click** near the center of the simulation domain.

An observation area will have been created. It is now necessary to edit the properties of the observation area using the same method as described above. Change the values to the following:

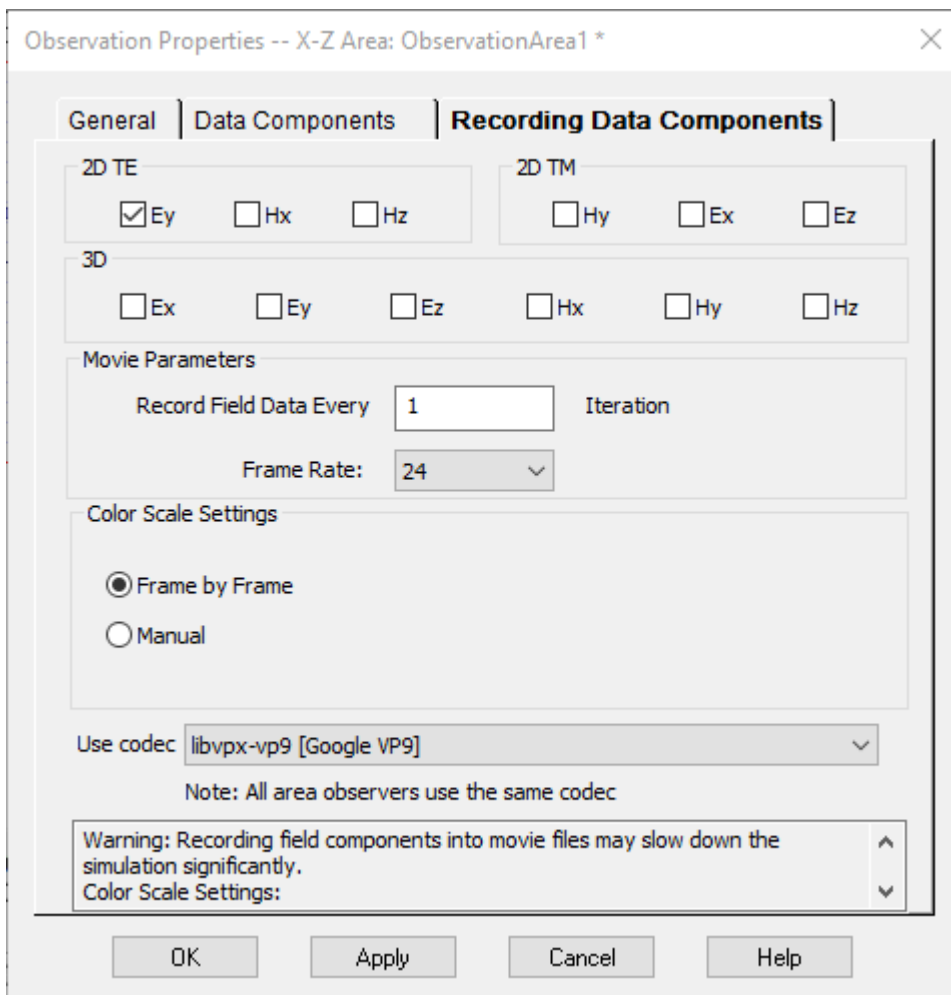
In the *General* tab:

field	value
Horizontal offset (um)	11.5
Vertical offset (um)	0
Z Length (um)	23
X Length (UM)	6.9

field	value
Label	ObservationArea1

Leave the defaults in the *Data Components* tab. This tab is used to select which fields are collected during simulation.

For the purpose of this example setup a movie in order to get a better view the propagating fields. Go to the *Recording Data Components* tab and select the Ey field in the *2D TE* box. The window should look similar to the image below:



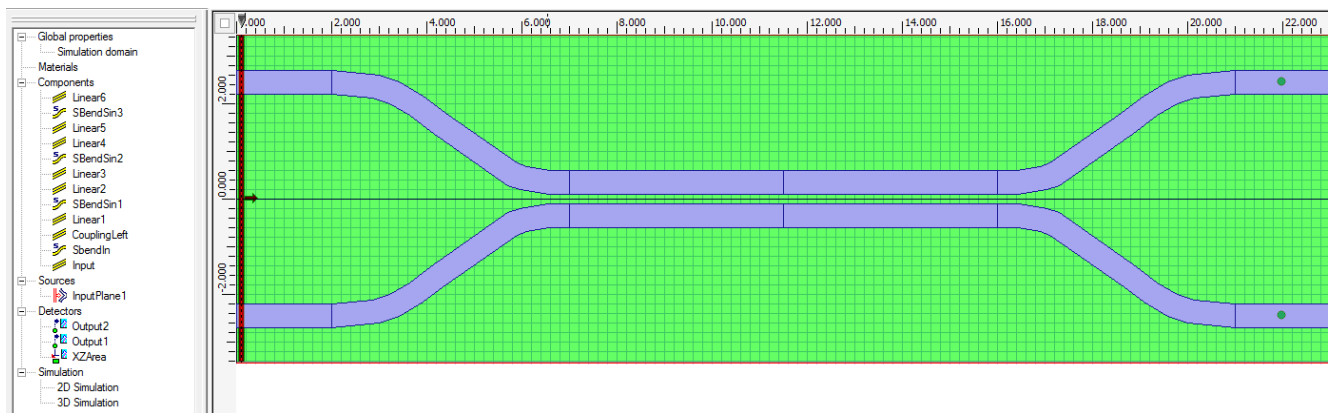
9 Observation Area properties dialog

This will tell the simulation to create a movie of the temporal evolution of the Ey field when running a 2D TE simulation. The properties for the observation area can now be saved by **clicking Ok**.

The design of the geometry is now complete.

2.3 Configuration of the simulation

The structure in layout view should now closely resemble the figure below



10 Completed 2D layout

If the green observation area is covering the waveguides, it can visually be moved to the background by **right-clicking** anywhere in the layout and selecting *Move to Back*.

The last step is to configure the simulation parameters.

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

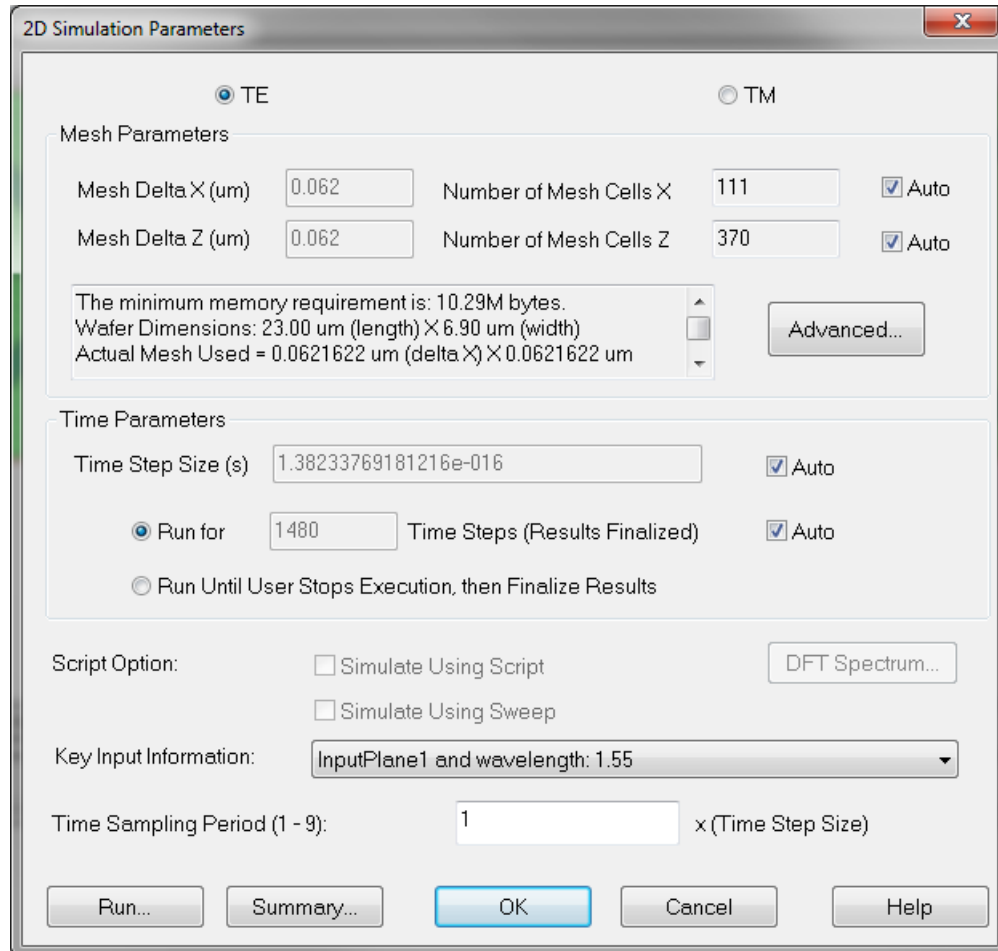
OR

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

As this is a TE simulation, leave the default value (*TE*) on top of the dialog selected. The mesh size parameters are automatically calculated by default (*Auto* boxes checked for both Delta X and Delta Z). For this simulation, the mesh size calculated corresponds to wavelength / 10, where wavelength is evaluated in the medium of highest refractive index (2), which is more than enough to ensure convergence and stability of the results for this design. It is important to always consider both the wavelength of the source as well as the size of the structural features in deciding the spacing for a computational mesh.

Leave the simulation time steps and simulation time to *Auto*. This ensures that the **Courant** condition is fulfilled (see the [Technical Background](#)¹ document for more details) and that the input wave travels to the far end of the grid.

The simulation parameters dialog box should look similar to the figure below



11 Simulation parameters dialog

Close the dialog by **clicking OK**.

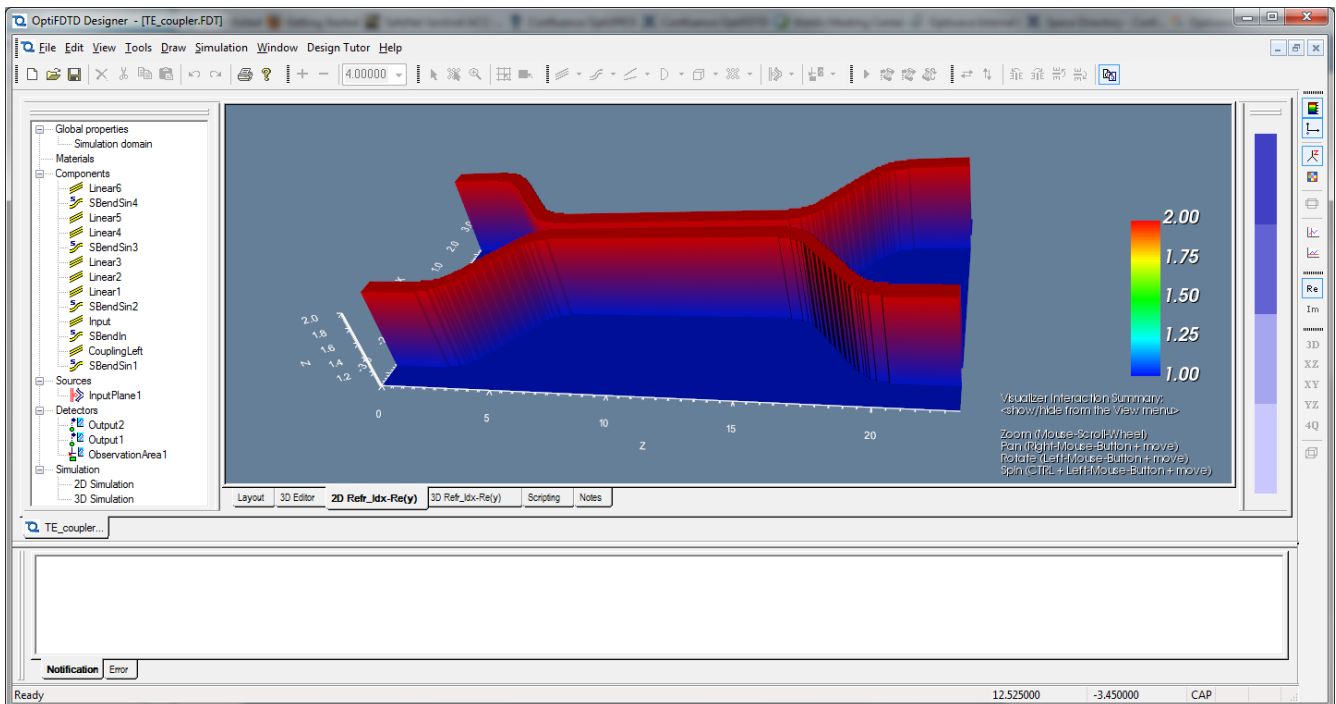
A vital step prior to running a simulation is to inspect the discretized refractive index distribution and meshing characteristics of the simulation domain.

2.3.1 Inspecting the refractive index distribution and mesh

Select the *2D Refr_Idx-Re(y)* tab in the main OptiFDTD Designer window to open the 2D refractive index viewer. Initially the view will be a color mapped planar image of the layout. Selecting *Height*

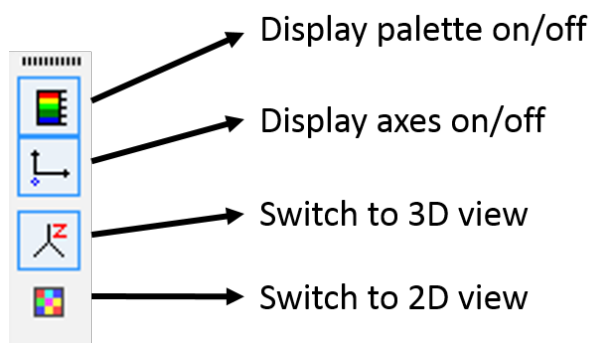
¹ <http://confluence:8090/display/OD/Technical+Background>

Plot from the right click contextual menu will present the 3D graph representing the refractive index of each mesh point in the simulation domain. The view can be rotated or zoomed using the mouse. By clicking on an individual mesh point, the location and refractive index values is shown in the bottom left corner of the window.



12 2D refractive index viewer

Along the right of the refractive index viewer, controls are available to assist in customizing the view



13 2D refractive index viewer

Using this viewer, effects of changing the mesh size on the geometry and the actual refractive index used in the simulation can be observed. Once the meshing is set to obtain a visually acceptable layout it is time to run the simulation.

Now, it's time to run a simulation.

2.4 Simulation of the designed Coupler

Prior to running the simulation, the design should be saved through *File -> Save as...* and picking a name and location for the project file.

⚠ OptiFDTD Designer project files have the extension *.fdt* and are represented by the icon:



OptiFDTD Analyzer result files have the extension *.fda* and are represented by the icon:

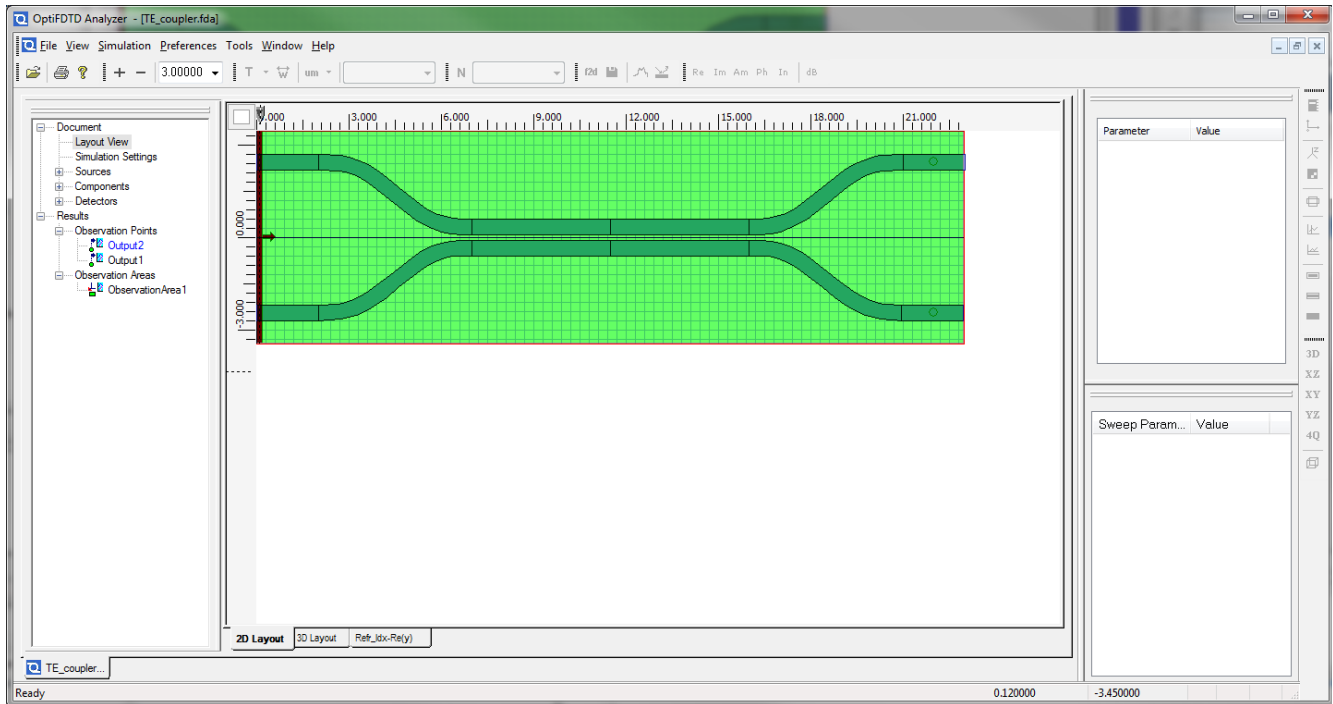


Re-open the the simulation parameters window by **double-clicking** *Simulate 2D* in the *Project Browser*. **Select** the *Run* button on the button left of the window to start the simulation.

After the simulation completes, a dialog box will appear requesting that **OptiFDTD Analyzer** be opened, **Click** *Yes*

2.5 Analysis of the results

The OptiFDTD Analyzer has a very similar layout to OptiFDTD Designer.



14 OptiFDTD Analyzer main window

The same views are available within Analyzer as tabs: 2D Layout, 3D Layout, and the refractive index distribution.

The simulation results can be displayed through the steps

- **Select** *Tools > Detector Analysis*

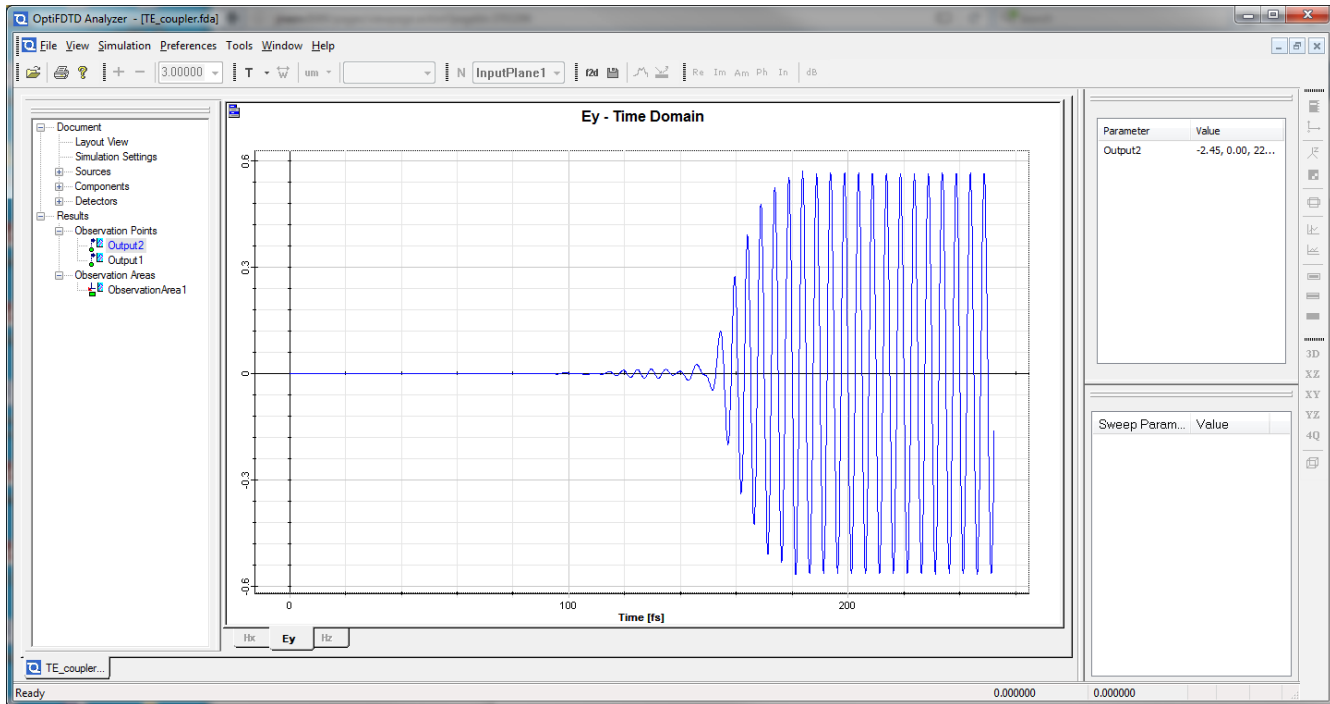
OR

- **Double-click** on any of the Observation Points or Observation Areas detectors (under the *Results* menu of the *Project Browser*)

2.5.1 Observation points


First, select one of the *Observation Point* detectors. **Click** on "Output2" under Observation Points

The window will be similar to the following:



15 Observation point analysis

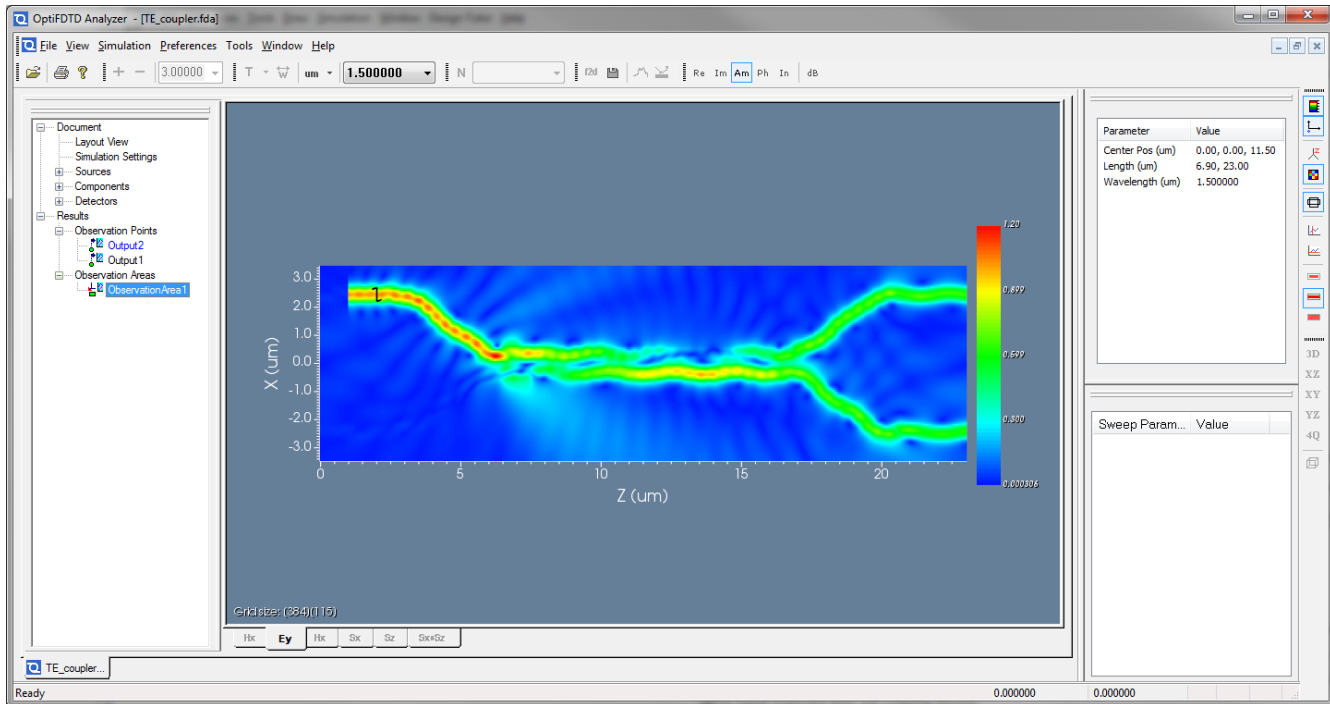
The other observation point can be added to the graph by **holding the SHIFT key** and **left-clicking** on "Output1", both graphs will then be superimposed. From the results it can be observed that the system reaches the steady-state after approximately 200 fs.

 The results window can be resized at any time by grabbing the bottom-right area with the mouse.

2.5.2 Observation areas

Under the Observation Areas node, **click** on "ObservationArea1"

The window displayed will be similar to the one below



16 Observation area analysis

This result is a color map of the steady-state amplitude of the E_y field as calculated using DFT during the simulation (see [Technical Background](#)² for details). The propagation of the mode through the input waveguide, the approximately 50/50 splitting of the signal and the output through the two output waveguides can be seen.

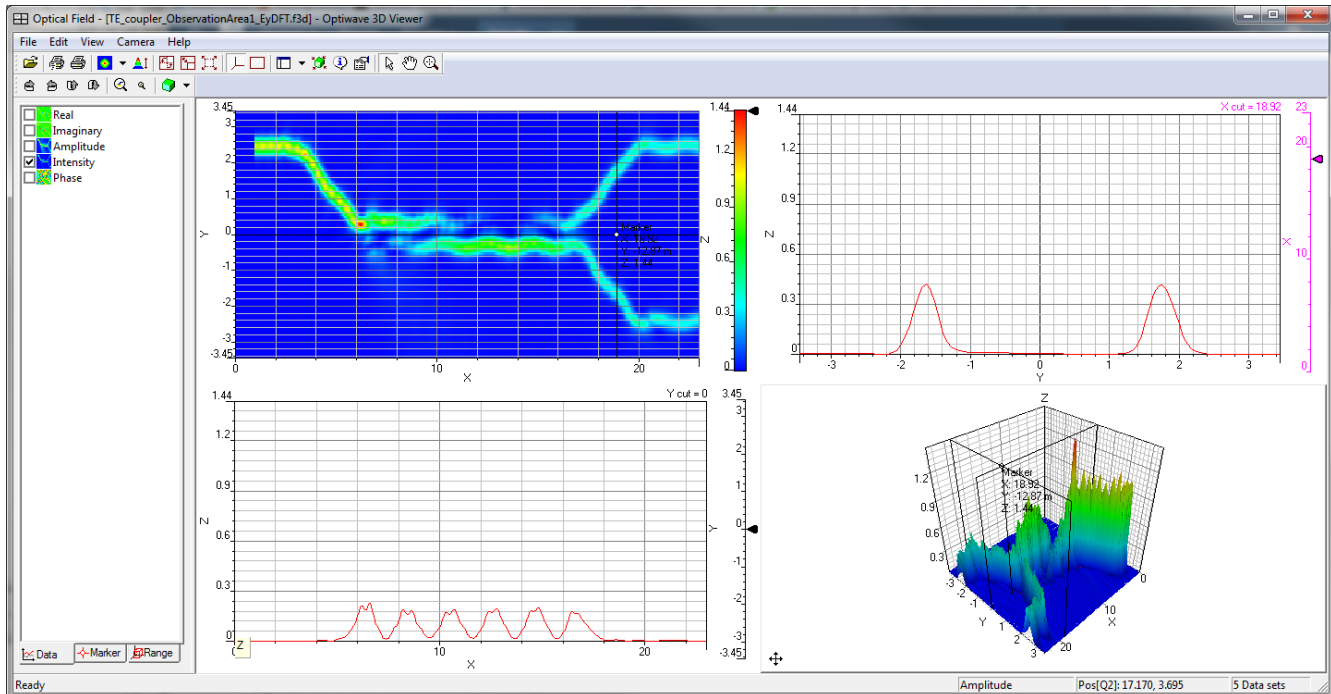
2.5.3 3D viewer application

Additional information about the field distribution at the output ports can be extracted from the 3D viewer application. In order to access this too the field data must be exported.

1. **Select** the *Export data* function under the *Tools* menu bar
2. Ensure that *ObservationArea1* is **checked**
3. **Click** *Export*

The result will be that a *.f3d* file will be created in the directory where the project is saved. An *.f3d* file can be opened using Optiwave's *3D viewer*. Simply **double-click** the *Ey* file in *Windows Explorer* to launch the application.

² <http://confluence:8090/display/OD/Technical+Background>



17 3D viewer

The 3D viewer window consists in 4 quadrants:

- **Top-left:** the XZ observation area exported by OptiFDTD Analyzer
- **Top-right:** a vertical line profile corresponding to the vertical red line in the top-left quadrant
- **Bottom-left:** a horizontal line profile corresponding to the horizontal red line in the top-left quadrant
- **Bottom-right:** a surface representation of the observation area

For more details, please consult the [User's Reference](#)³ manual.

The coupler's output can be seen by dragging the vertical red line in the observation area graph (top-left quadrant) to the right to show a line profile. It can be seen from the line profile (top-right) that the output waveguides are balanced with approximately 40% transmission in each branch (**thus the Coupler has roughly a 1 dB loss - 20% of the incident power has been lost**)


2.5.4 Movie

A movie of the propagating optical field has been saved in the directory that the project file was saved in. You can play it using *Windows Media Player* or any third party player such as *VLC*.

³ <http://confluence:8090/display/OD/User%27s+Reference>

3 Photonic Crystal Band Diagram

In this example, an FDTD project is made to analyse a photonic crystal by the Plane Wave Expansion (PWE) method. This example is also an implementation of Figure 1 of Chapter 5 in reference

 Unknown macro: 'footnote'

. The structure is an array of cylindrical rods with permittivity 8.9 arranged in a square pattern. A trajectory of K vectors is specified in K space and optical frequencies for Bloch waves are found for each point on the trajectory. These frequencies are plotted in Figure 2 of this reference to identify frequencies for which no optical propagation is possible, the so-called band diagram.

To reproduce these results in OptiFDTD, follow these steps:

- **Open** the OptiFDTD Designer
- Go to *File > New* to create a new project
- Specify the size of the simulation space by going to *Edit > Wafer Properties...* . In this dialog box, specify a length and width of 10 μm .
- In the 2D Wafer Properties tab of the Wafer Properties dialog box, specify the material to be Air. This specifies that the rods are surrounded by air.
- Create the material of the rods. In the *Project Browser*, **right-click** on *Materials* and select *Add Dielectric Material*.
- **Enter** the name "rod" and specify the isotropic refractive index to be 2.983287 ($= \sqrt{8.9}$)
- **Open** the WG_Fiber_Example profile from the *Project Browser* by **right-clicking** and **selecting** *Properties*.
- This simulation will be done in 2D, so the 2D profile definition is the only relevant one in this project. In the 2D profile definition panel, **select** the new material "rod" from the combo box. Save the change using apply or cancel.

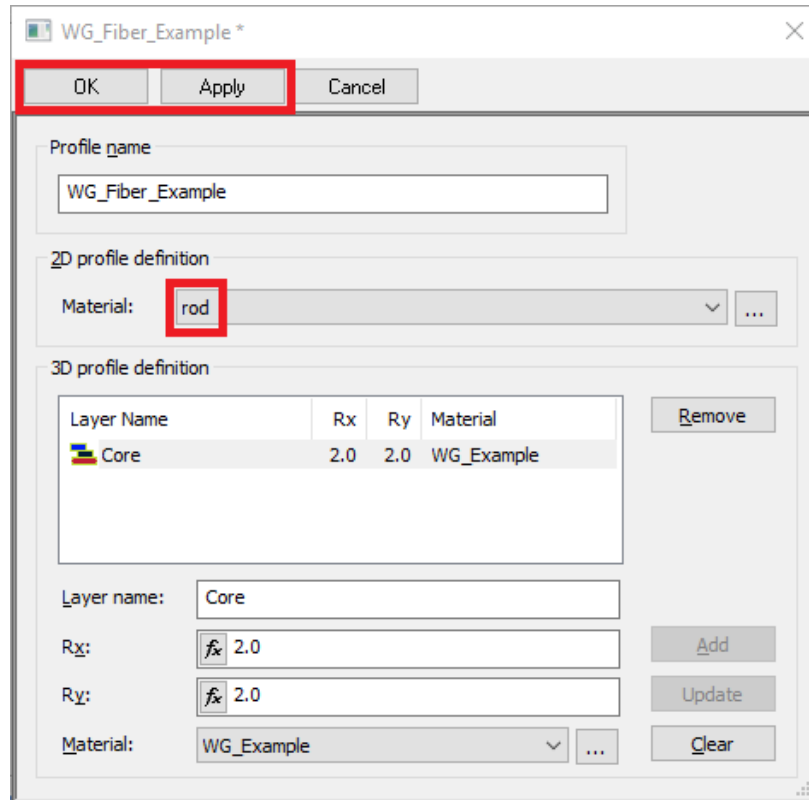


Figure 1, WG_Fiber_Example dialog box

- **Click** on the layout to activate draw functions. **Select** *Draw > PBG Crystal* and click in the layout space to place the crystal structure there.
- In the *Project Browser*, **open** the new component *PBGCrystalStruct1* to get the *Crystal Lattice Properties* dialog box.
- In the *Atom Waveguides* in *Unit Cell* panel, **click** *New* to get the *Elliptic Waveguide Properties* dialog box
- For the *Major* and *Minor radius*, **enter** 0.2 For the *Profile*, **select** *WG_Fiber_Example* from the combo box.

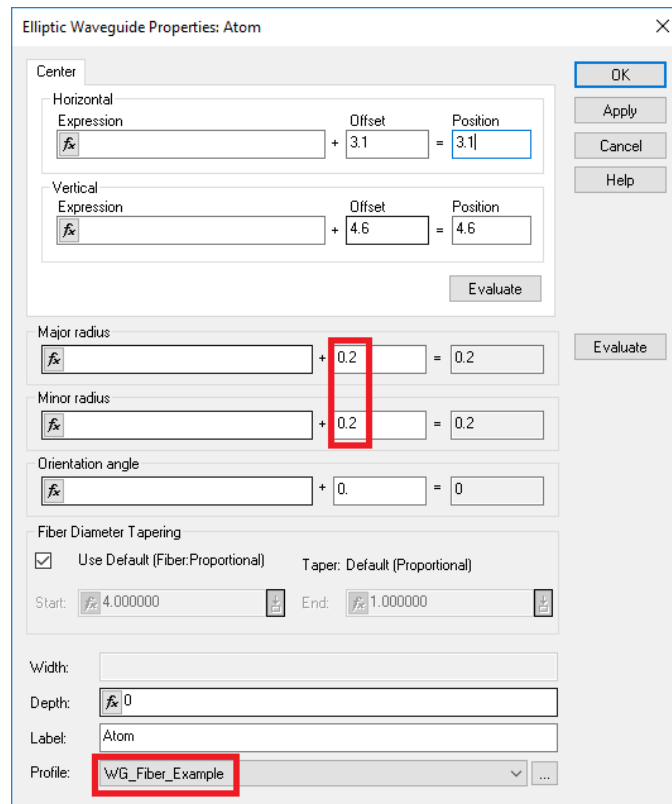


Figure 2, Elliptic Waveguide Properties dialog box

- **Click OK** to return to the Crystal Lattice Properties dialog box.
- In the Crystal Lattice Properties dialog box, **specify** the origin coordinates to be 1 for Horizontal, -3.5 for Vertical
- For *Lattice dimensions*, **enter** 8 for #A and #C. **Click OK**.

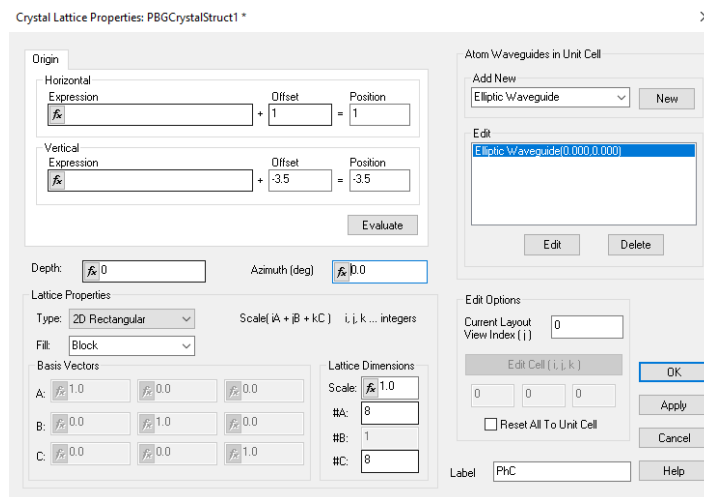
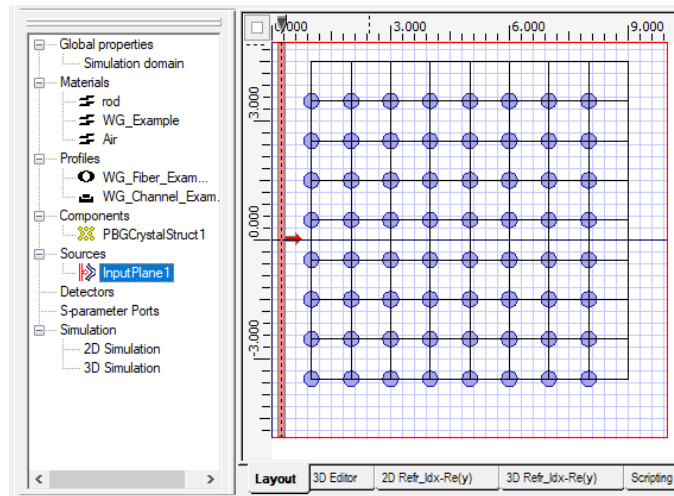
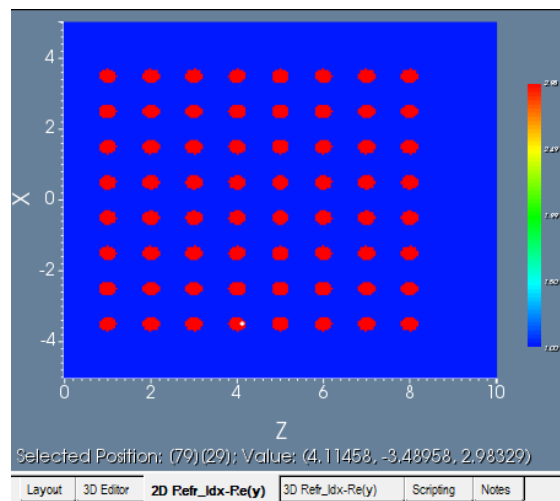


Figure 3, Crystal Lattice dialog box


- The Input Plane is not used in the PWE simulation. However, a wavelength must be defined for OptiFDTD, which is done through the input plane. **Click** on the *Select Tool* button to cancel PBGCrystal creation mode and to activate the draw functions.
- **Select Draw > Z-direction input plane**, and draw this somewhere on the layout away from the photonic crystal.
- Once created, the Input Plane will appear in the *Project Browser*. **Right-click** on it, **select Properties**, and **enter** the wavelength 1.55 in the *Wavelength* field. This wavelength is not used in PWE simulations, but a value is required to define the wavelength to display the refractive index of dispersive materials.

**Figure 4, Input Plane. (Not used in PWE simulations)**

- Observe the resulting 2D refractive index distribution in the 2D Refr tab. Check that there is a periodic array of circles, radius 0.2, in a square pattern, with refractive index 2.983287, and surrounded by air.

**Figure 5, Refractive Index Distribution**

1. Save and name the project

2. **Select** *Simulation > PWE Band Solver Parameters ...* to launch the *PWE Parameters* dialog box
3. **Select** the *2D Solver*.
4. **Select** the *TE Polarization*. Optiwave follows the convention that transverse electric (TE) means that the electric field in the simulation is parallel to the rods (i.e. the electric field is perpendicular to the 2D space of the simulation). Optiwave TM polarization will find waves in which the magnetic field is parallel to the rods. The convention used in Ref  is the opposite. Therefore selecting TE polarization in OptiFDTD will find the curves labelled TM in the reference, and Optiwave TM simulations will find the curves labelled TE in the reference.
5. **Select** the # *bands* to be 4, and the *Tolerance* to be 1.e-6.
6. **Define** the K-vector path by **selecting** Square ZX from the combo box. This pre-defined path labels the vertices G, X, M. The path will start at G, continue to X, then to M, and then back to G. The path is a triangle in K space as shown in Figure 2 of the reference
7. For better results, **change** the Div numbers at X, M, and the last G to 10. This will make the simulator calculate the optical frequencies at 10 points while going from one vertex to the next. Check that the Mesh specifies 16 steps in a and c, and that the domain origin is at (0,0).

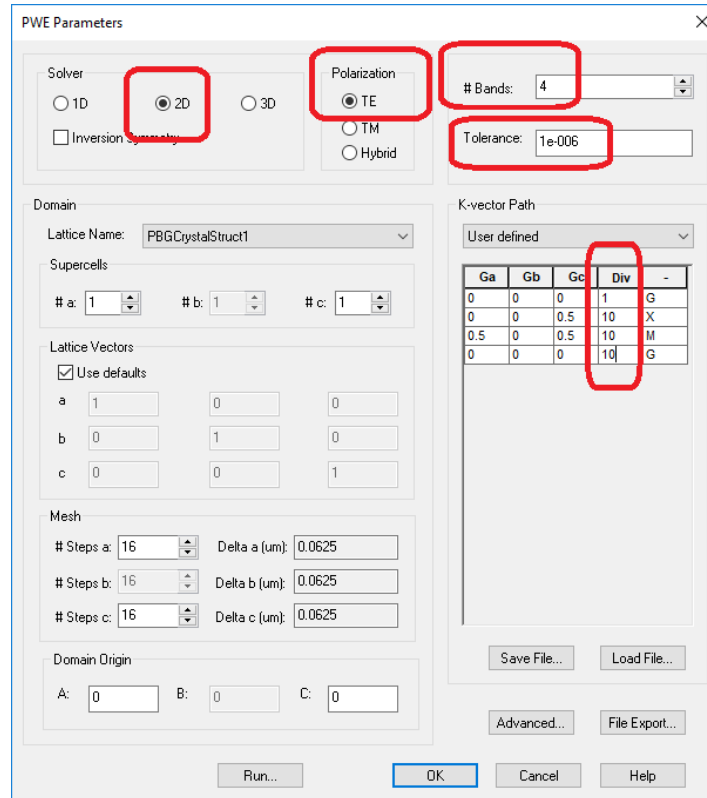


Figure 6, The PWE parameters

- **Click** *Run...* The PWE Band Solver Simulator will appear and display the result, as in Figure 7. **Click** on the *Connect* and *Locate Band Gaps* buttons to connect the dots and locate the band gaps.

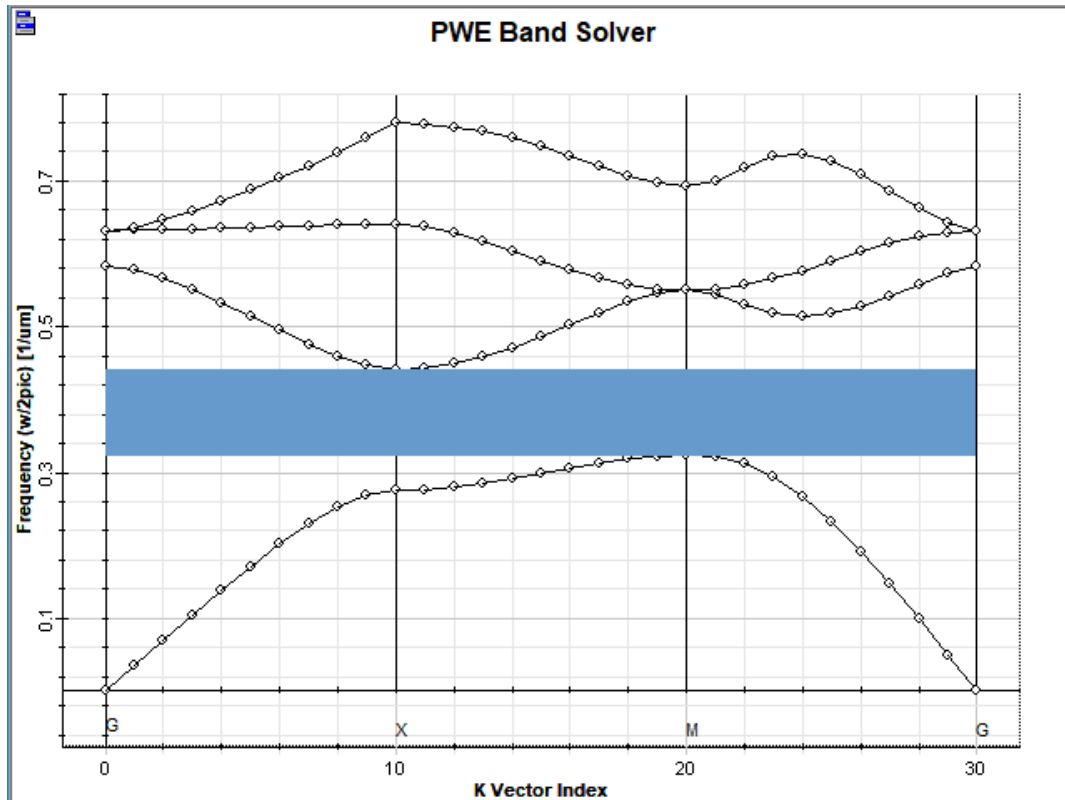



Figure 7, Frequencies of optical Bloch waves in the photonic crystal

Figure 7 reproduces the result shown in Figure 2 of Chapter 5 in the reference, recall that while these are TE results for OptiFDTD these are labelled TM in the reference. To calculate the band diagram for the other polarization, go back to the PWE parameters dialog box (*Simulation > PWE Band Solver Parameters ...*) and change the radio button in the Polarization panel to TM. Then **click Run...** again.

Reference

 Unknown macro: 'display-footnotes'