

# Analysis of Photonic Crystal Fibers (PCF) in OptiFDTD

## Introduction

In the past 10 years, photonic crystal fibers (PCF) have attracted much scientific and commercial interest. The research and design work for PCF starts from accurate modal analysis of the fiber. Once the modes are found, all of the PCF properties such as loss, dispersion, and cutoff can be determined.

There are general three types of PCF: holey fiber (HF), field-confined holey fiber (FCHF) and Hole-Assisted Light-guide Fiber (HALF), as shown in figure 1.

In OptiFDTD, the mode solver is combined with the FDTD engine so that FDTD simulation can use the modal field as input directly. In addition to this integrated mode solver, OptiFDTD also provides an independent mode solver to allow user to solve and study the mode outside of an FDTD calculation. The PCF mode solver will mainly be used as an independent mode solver. Modal analysis requires two main steps:

- (1) Create the layout
- (2) Calculate the modes

We will use waveguide layout designer to define the PCF layout, and then convert the layout to a refractive index distribution file. The mode solver will load in the index distribution file and perform the mode analysis.

Some users may want to scan the wavelength to find out the cutoff wavelength, or want to scan the layout parameters. All of this is possible with OptiFDTD

The following section will demonstrate how a photonic crystal fiber can be analyzed in OptiFDTD.

## Sample 1---Photonic Crystal Fiber---Holey Fiber

The most common PCF is Holey Fiber as shown in [figure 1 \(a\)](#), the cladding region consists of a hexagonal array of air-holes; One missing air-hole defines the core region. The real layout may have more layers of air holes, but it is sufficient to model only three layers since the mode is well confined <sup>[2]</sup>. To be safe, here we take five layers for the mode solving. We use the PBG array editor to define the hexagonal air hole. To make the core center in the simulation domain's center, we need to draw a graph ([see figure 2](#)) so that the detail dimension information can be marked. Such a graph can make the layout creation easier, so it is highly recommended.

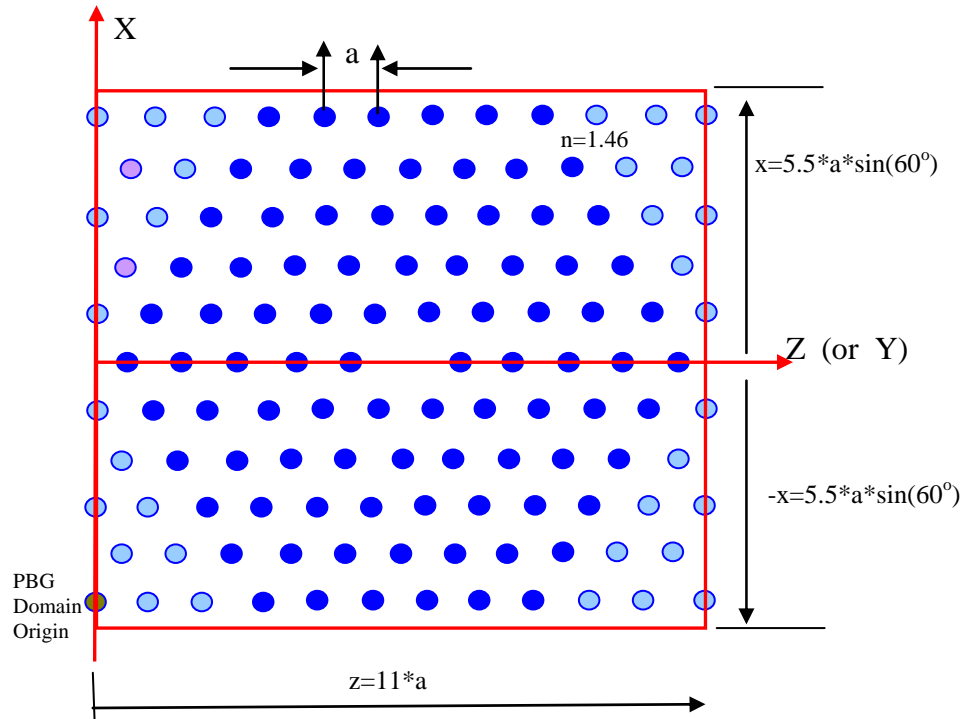


Figure 2 Five layers holey fiber

Please note:

For this five layer holey fiber

- The background material refractive index  $n = 1.46$
- Hexagonal Air hole, periodic  $a = 2.3\mu\text{m}$ ,  $n = 1.0$ , radius  $R = 0.6\mu\text{m}$
- Simulation domain,  $z = 11*a$ ,  $x = 11*a*\sin(60^\circ)$
- The green dot is the hexagonal lattice original point  $x = -5*a*\sin(60^\circ)$ ;  $z = 0$

## i. Create the layout

To design a holey fiber structure, perform the following procedures.

- | Step | Action   |
|------|--|
| 1    | Start <b>Waveguide Layout Designer</b> .   |
| 2    | To create a new project, select <b>File &gt; New</b> .<br><i>The Initial Properties dialog box appears.</i>  |
| 3    | Click <b>Profiles and Materials</b> .<br><i>The Profile Designer window appears.</i>   |
| 4    | Under the <b>Materials</b> folder, right-click the <b>FDTD-Dielectric</b> folder and select <b>New</b> .<br><i>A new Dielectric material dialog box appears.</i> |
| 5    | Type the following information:<br>Name: <b>n=1.46</b><br>Refractive index (Re): 1.46  |
| 6    | To save the material, click <b>Store</b> .<br><b>N=1.46</b> appears in the <b>FDTD-Dielectric</b> folder in the directory and in the dialog box title bar.       |
| 7    | Under the <b>Profiles</b> folder, right-click the <b>Channel</b> folder and select <b>New</b> .  |

- The **ChannelPro1** dialog box appears.*
- 8 Create the following channel profile:  
 Profile name: **Channel\_Air**  
 2D profile definition  
 Material: **Air**
- 9 Click **Store**.  
 (Close the **Profile Designer**.or leave it open)
- 10 In the **Initial Properties** dialog box, , type/select the following:  
**Waveguide Properties**  
 Width [ $\mu\text{m}$ ]: **0.6**  
 Profile: **Channel\_Air**  
**Wafer Dimensions**  
 Length: **length**  
 Width: width  
**2D Wafer Properties**  
 Material: **n=1.46**
- 10 Click **OK** in the initial dialog box. Click **Yes** in the question dialog box to start define the variable. **Variables and Functions dialog box appears**  
*Because variable (length, width) is used in the initial dialog box .Click OK will promote the parameter editor dialog box*
- 11 In the **Variables and Functions** dialog box appears, define the following variable in sequence then click **OK** to close the dialog box

Name	value	
a	2.3um	~lattice constant
b	$a \cdot \sin(\pi/3)$	~height of triangle cell
length	$11 \cdot a$	~length of domain
width	$11 \cdot b$	~width of domain
c	$-5 \cdot b$	~X original point of lattice
R	0.6um	~Radius of air hole

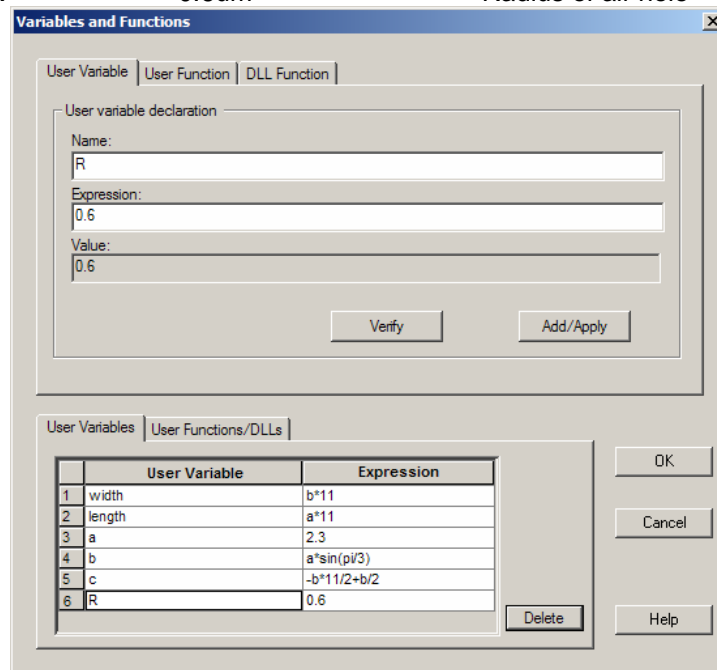


Figure 3 Variable and function dialog box

**Notes:** you can access Variables and functions dialog box in anytime by Select "Edit Parameters" under Simulation menu of layout designer

12. From the **Draw** menu, select **PBG Crystal Structure**.
13. Click mouse once in the layout window, *The **PBG Crystal Structure** appears in the layout window (see [Figure 4](#))*. Click select tool ( the arrow shortcut toolbar) to release the PBG selection.

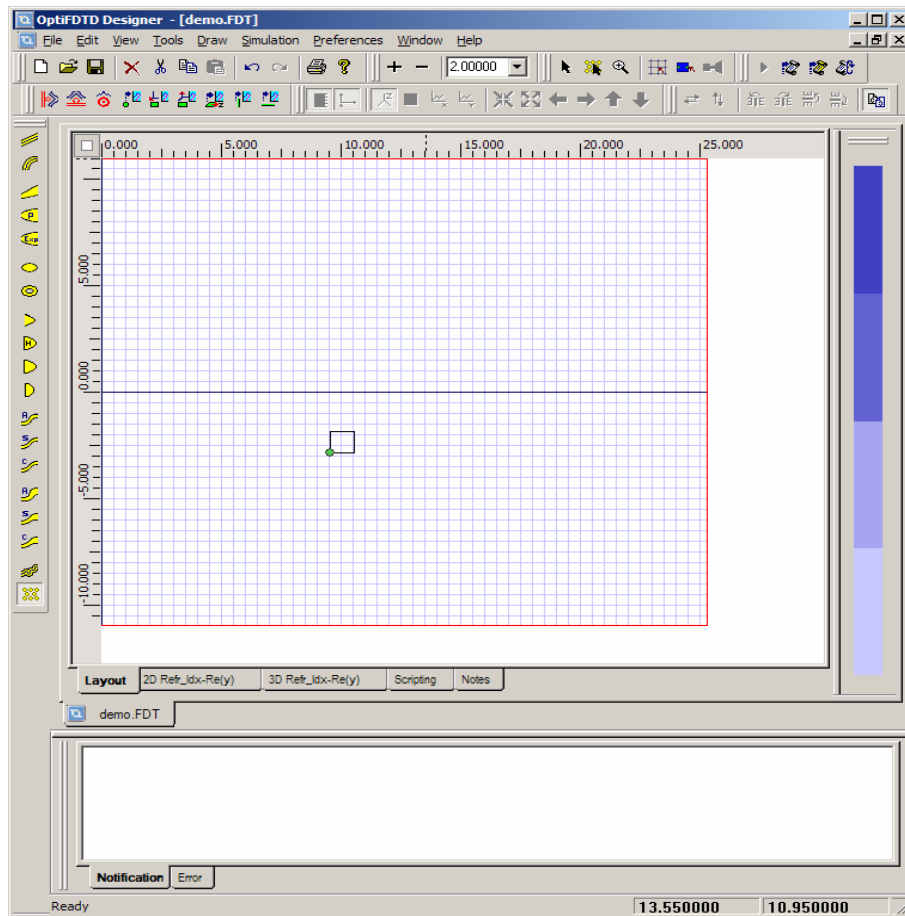


Figure 4 Layout window with initial PBG layout

14. To edit the crystal structure, double-click on the PBG structure (a rectangular shape) in the layout. *The **Crystal Lattice Properties** dialog box appears ([Figure 5](#))*.

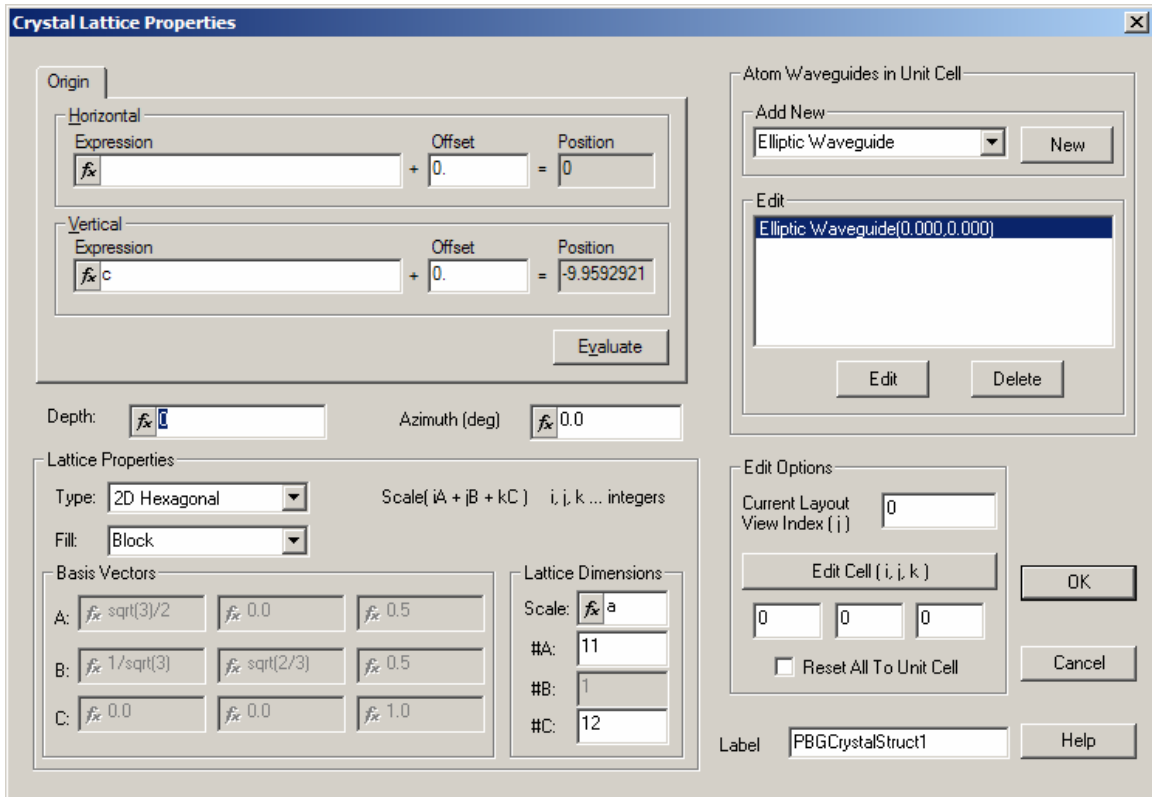


Figure 5 Crystal Lattice properties dialog box

- 15 In the Crystal Lattice Properties dialog box, Set the following parameter

**Origin**

**Horizontal, expression:** 0  
**Horizontal Offset:** 0  
**Vertical Expression:** c  
**Vertical Offset:** 0

**Depth:** 0.0  
**Azimuth:** 0.0

**Lattice Properties:**

**Type:** 2D Hexagonal  
**Fill:** Block

**Lattice Dimension:**

**Scale:** a  
**#A** 11  
**#C** 12

- 16 In **Atom Waveguide in Unit Cell, Add New**, select **Elliptic Waveguide** from the drop-down menu and click **New**. The **Elliptic Waveguide Properties dialog box** appears (see [Figure 6](#)). In **Elliptic Waveguide Properties dialog box** set following value

In **Center, Offset**, type/select the following:

Horizontal: **0.0**  
Vertical: **0.0**  
Type/select the following:  
Major radius: **0.6**  
Minor radius: **0.6**  
Orientation angle: **0.0**  
Channel thickness tapering: **Use Default (Channel: None)**  
Depth: **0.0**  
Label: **Atom**  
Profile: **Channel\_Air**.

Click **OK**.  
*The **Elliptic Waveguide Properties** dialog box closes.*

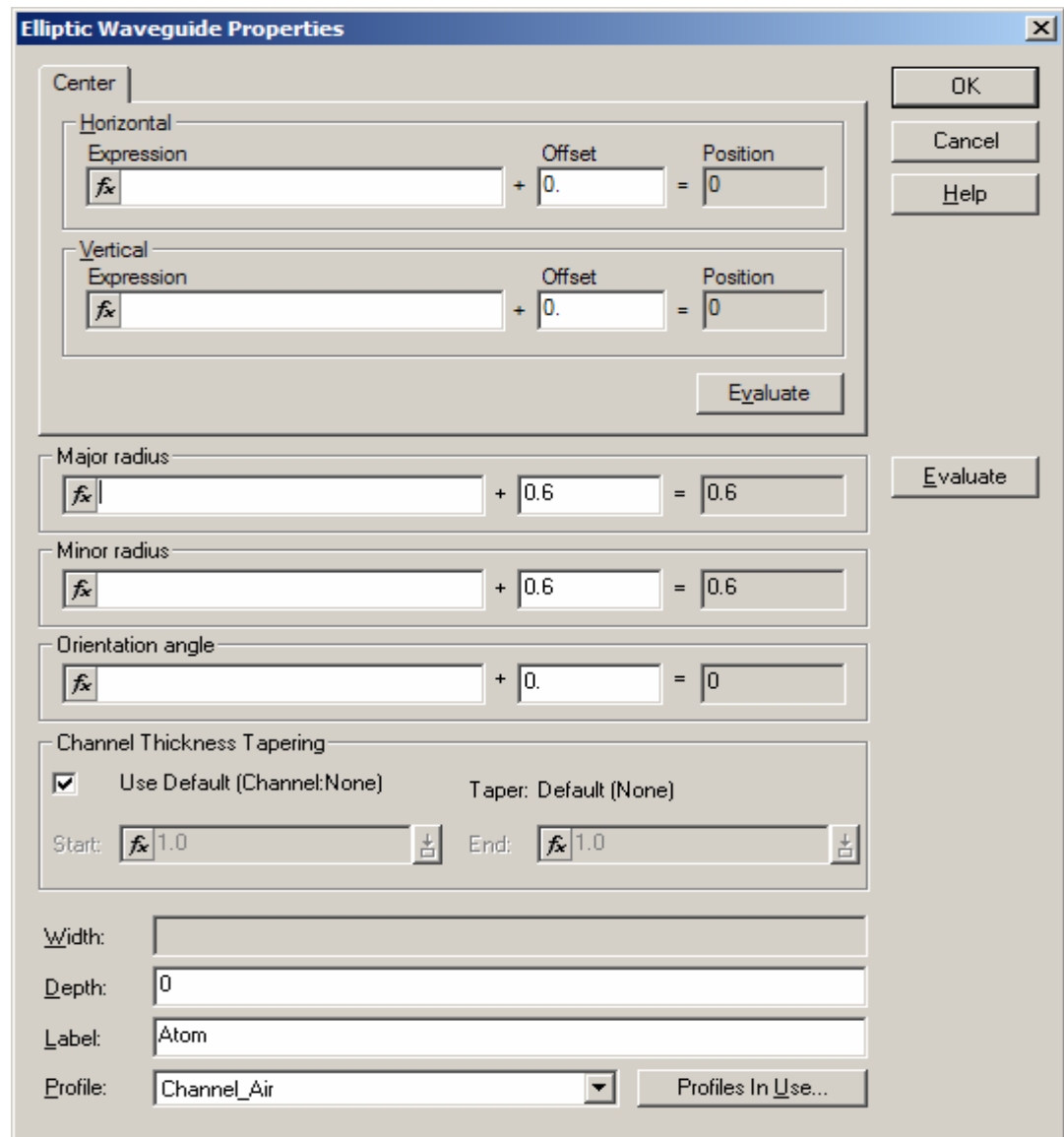


Figure 6 Elliptic Waveguide Properties dialog box

**Note:** When you return to the **Crystal Lattice Properties** dialog box, you will see the defined elliptic waveguide listed in **Atom Waveguide in Unit Cell**.

**Note:** If you close the **Crystal Lattice Properties** dialog box, you will see the defined PBG structure in the layout window (see [Figure 7](#)).

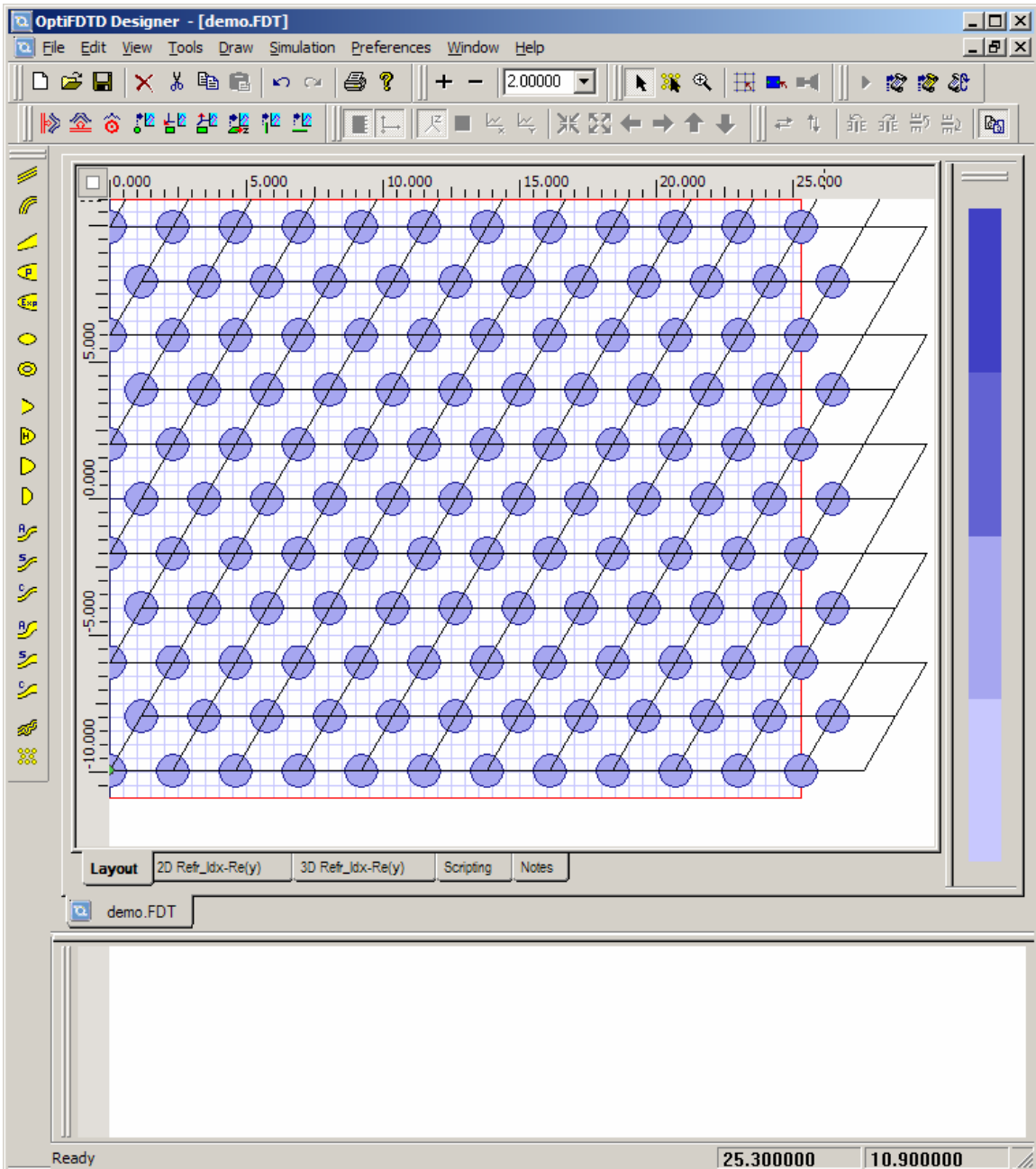


Figure 7 PBG layout in designer window

- 17 In the layout designer, to select the PBG lattice, click the PBG area. The **PBG Crystal Structure Cell Editing Tool** (a shortcut toolbar beside the Arrow select tool as shown in figure 8) becomes active. Select this shortcut tool bar., When this tool

is selected, right click on the photonic cell and click "Cells Off" to disable the cells. With this step, a five layer Holey fiber can be realized (see [figure 9.](#))

18



Figure 8 shortcut toolbar for PBG Crystal Structure Cell Editing Tool

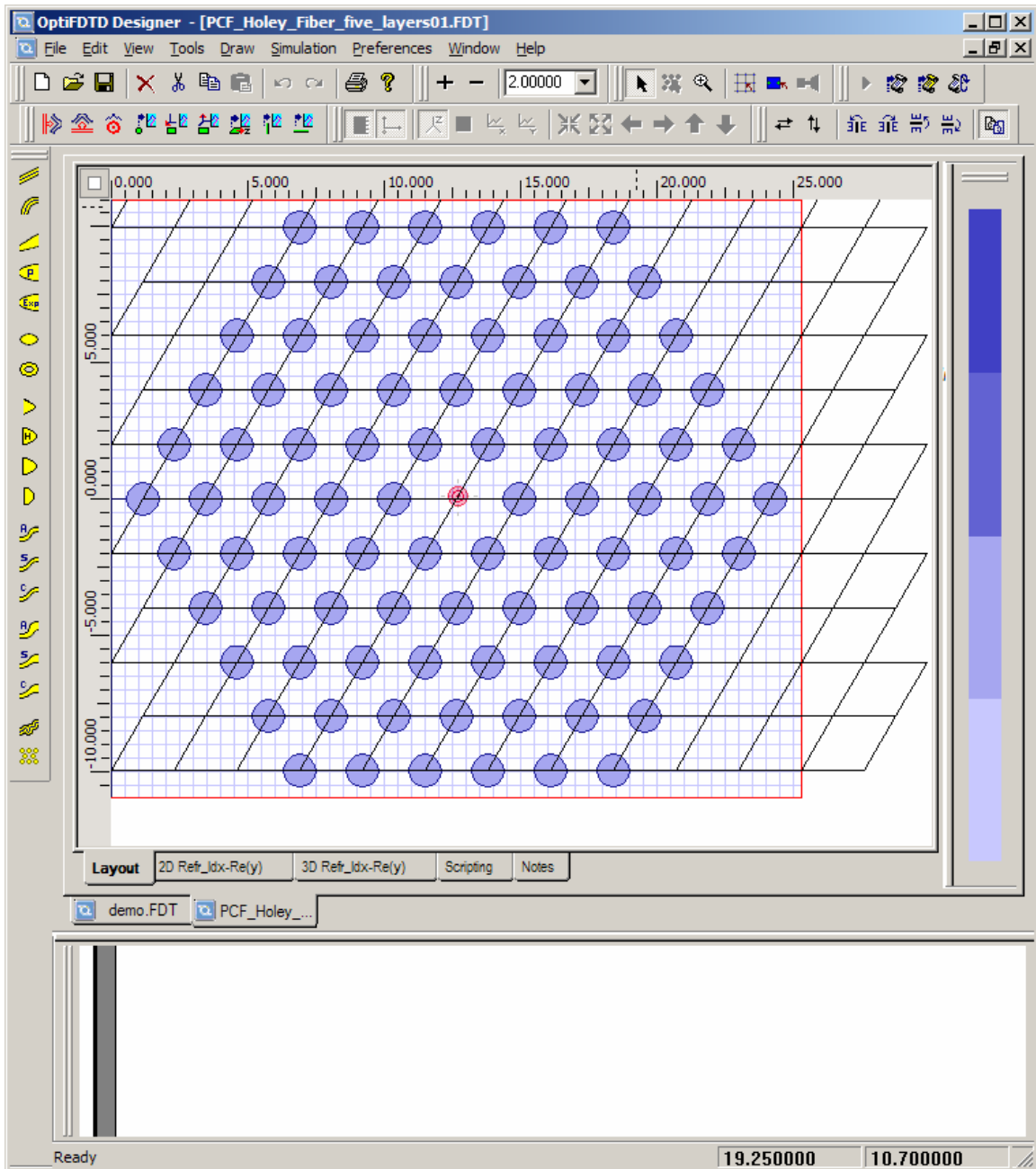


Figure 9 Five layers holey fiber

- 19 To observe the index distribution, please set a point source in the layout. The center **Wavelength** is 1.3um with Gaussian **Modulated Continuous Wave** as time domain waveform
- 20 Click **2D Simulation Parameters** under **Simulation** menu, set following parameters and Click OK.
- Mesh Delta X: 0.08um  
Mesh Delta Z: 0.08um  
**Run for 1 time steps**
- 21 Click **2D Refr\_Idx\_Re(y)** to observe the refractive index distribution. ( see figure 10)

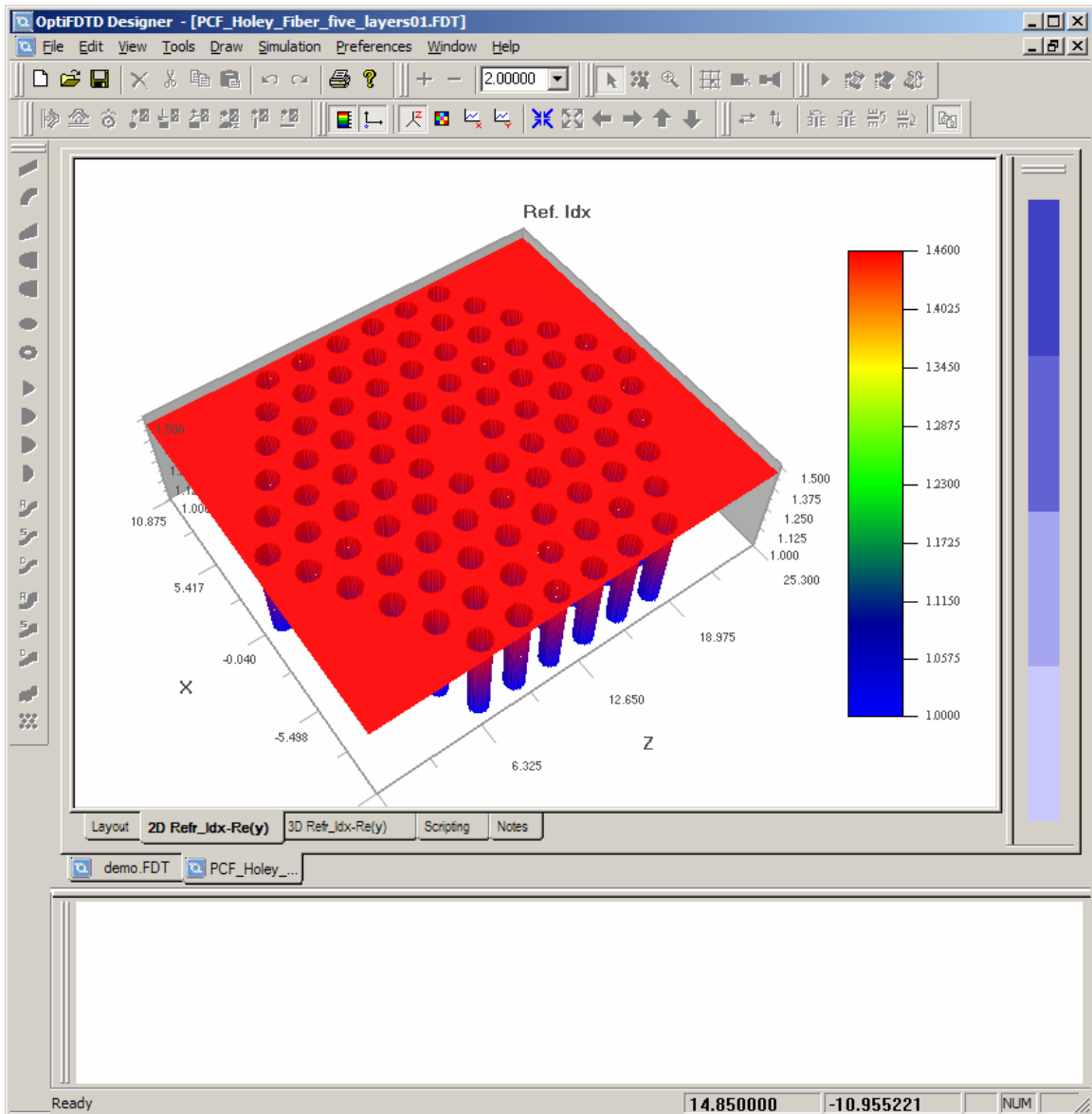


Figure 10 Refractive index distribution

## ii. Convert Refractive index distribution to a file

Because we will use the independent mode solver to solve the mode, we have to save the refractive index distribution to a file. In the current version of OptiFDTD, the option to save the refractive index distribution to file only exists in the analyzer. We have to perform a short simulation so that all the data is written to an analyzer file. (In the next release, there will be the option to save the refractive index distribution directly from the layout designer).

- | Step | Action  |
|------|---|
| 1    | Set an input wave (with Gaussian Modulated Continuous Wave as time domain waveform) in the layout, (This work was done in step 19 in a previous section). |
| 2    | Set proper mesh size for mode solving (This work was done in step 19 in a previous section)   |
| 3    | Set time domain simulation iteration number as 1. (This work was done in step 19 in a previous section)   |
| 4    | Perform the 2D simulation. After the simulation, open the analyzer.   |
| 5    | In the Analyzer, Click Export under file menu.<br>Export data dialog box appears (see figure 11)  |

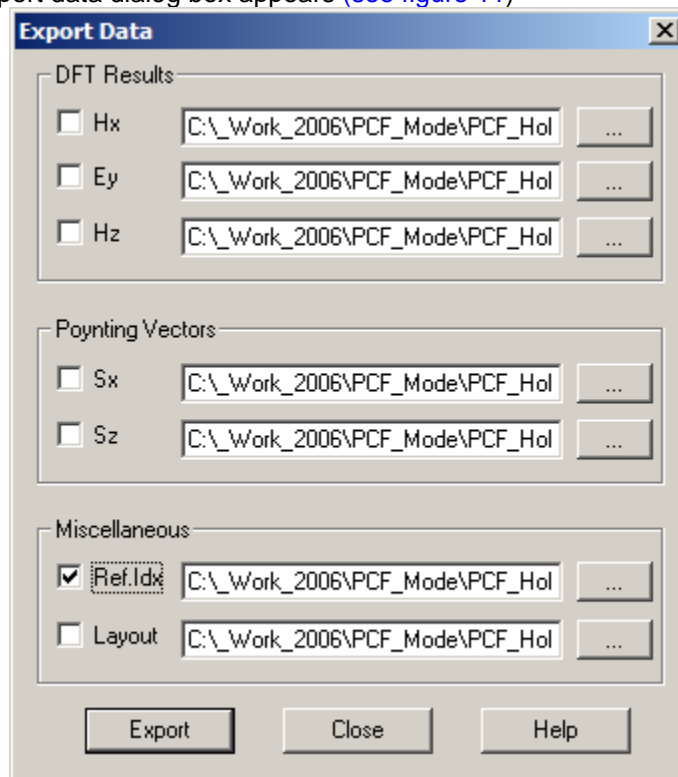


Figure 11 Export data dialog box

- 6 Check **Ref.Idx** Box and click “...” button to set the file name and folder  
The file name is [PCF\\_Holey\\_Fiber\\_five\\_layers01.rid](#) in this tutorial. Click Export to export data. Click **Close** button to close the dialog box

## iii. Perform the Mode analysis

Step Action

- 1 To open Independent 3D Mode Solver, from the **Start** menu, select **Programs > Optiwave Software > OptiFDTD 6.0 > Tools > Mode 3D**. *3D Mode solver appears (see [Figure 12](#))*

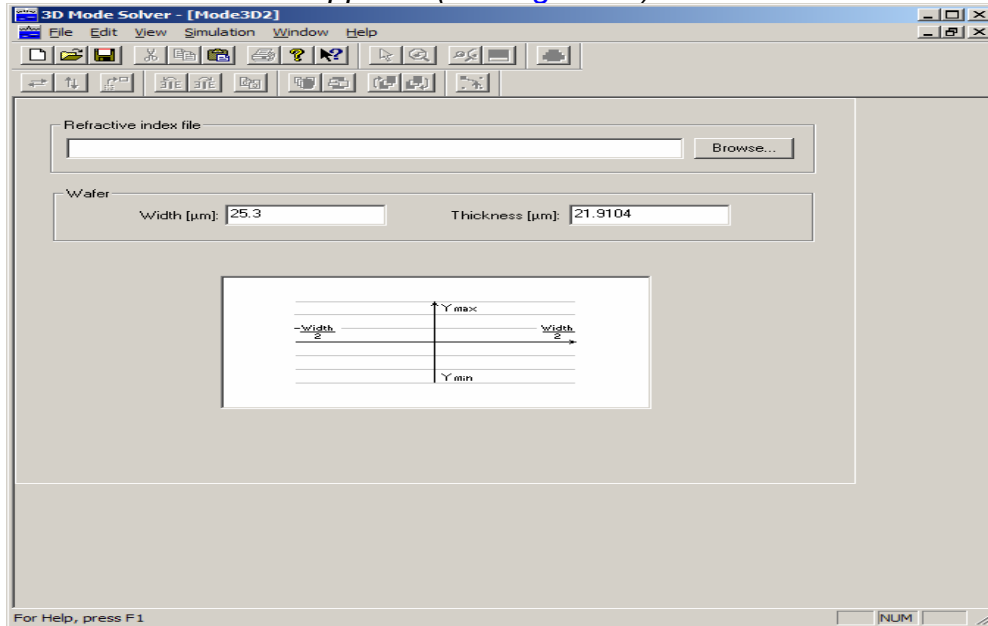


Figure 12 3D Mode Solver

- 2 Click "**New**" under file menu and select "**User Defined File**" and click OK in the pop up window.
- 3 Click Brose button to load in the refractive index file ([PCF\\_Holey\\_Fiber\\_five\\_layers01.rid---](#) in this tutorial )
- 4 Click **Simulation** menu and select **ADI Method** under **Global Parameters**. **Global Data:ADI Method** dialog appears ( see [Figure 13](#))

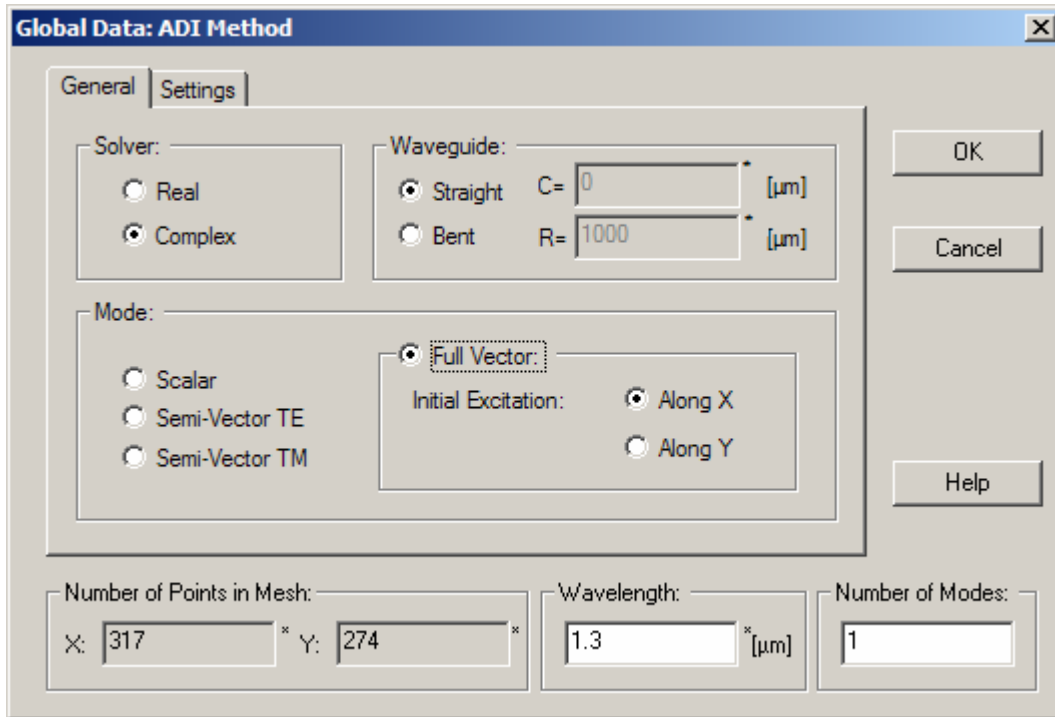


Figure 13 Global data: ADI Method dialog box

- 5 In the Global data: ADI Method dialog box, type the following value, then click **OK** button to close the dialog box

**Wavelength:** 1.3um  
**Number of Modes:** 1

**General:**

**Solver:** Complex  
**Mode:** **Full Vector**  
**Initial Excitation:** Along X  
**Waveguide:** Straight

**Setting:**

**Start Field:** Gaussian  
**Accuracy:**  
**Index Tolerance:** 1E-007  
**Field Tolerance:** 1E-005  
**Boundary Condition:** TBC

Notes: Other setting follows the default

- 6 Select “**Calculate ADI**” under simulation menu. Click **Run** button to start the mode analysis. 3D mode solver resulting window appears (see [figure 14](#)).

The layout will be first displayed, once the mode is solved (after a while), the display will be modal filed. (Click Display under Structure View will display the layout again)

Notes:

- (1) save the field pattern by click **Save As** button
- (2) Save the modal index by click “**Save Table**”
- (3) Click **Close** in the file menu to close this window

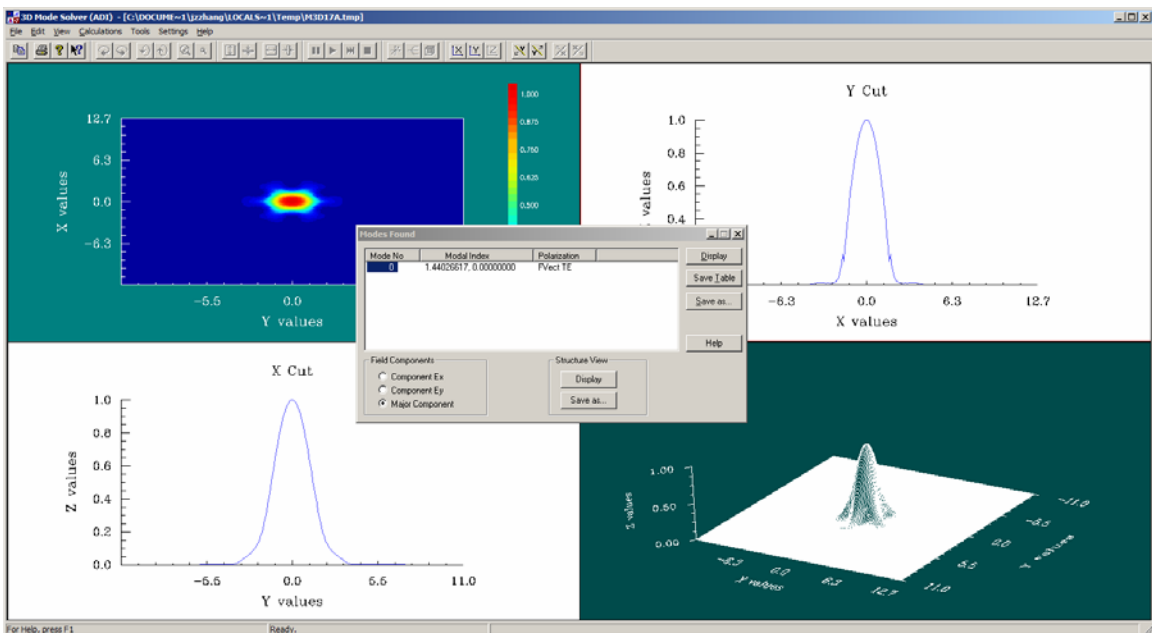


Figure 14 3D mode solver results window

#### iv. Perform the wavelength scanning mode analysis

##### Step Action

- 1 In the **3D Mode Solver** Window, click **Edit Parameters** under **Simulation** menu. **Edit Parameter** dialog box appears (see [figure 15](#)).

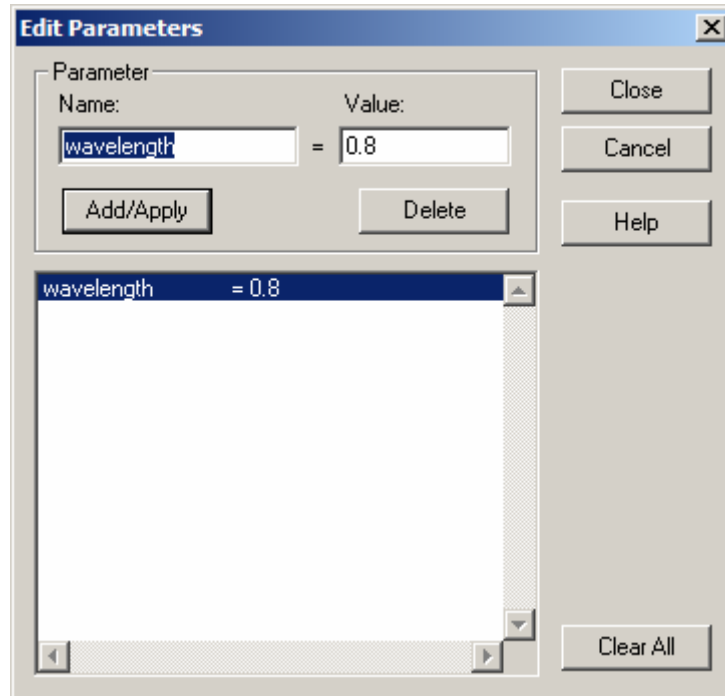


Figure 15 Edit Parameters dialog box

- 2 In the **Edit Parameter** dialog box, in **Name** box, input “**wavelength**” and set value as **0.8**, Click **Add/Apply** button to make the defined variable to be listed. Click **Close** to close the dialog box
- 3 Click **Scan Parameters** under **Simulation** menu. **Scan parameters** dialog box appears (see [figure 16](#)).

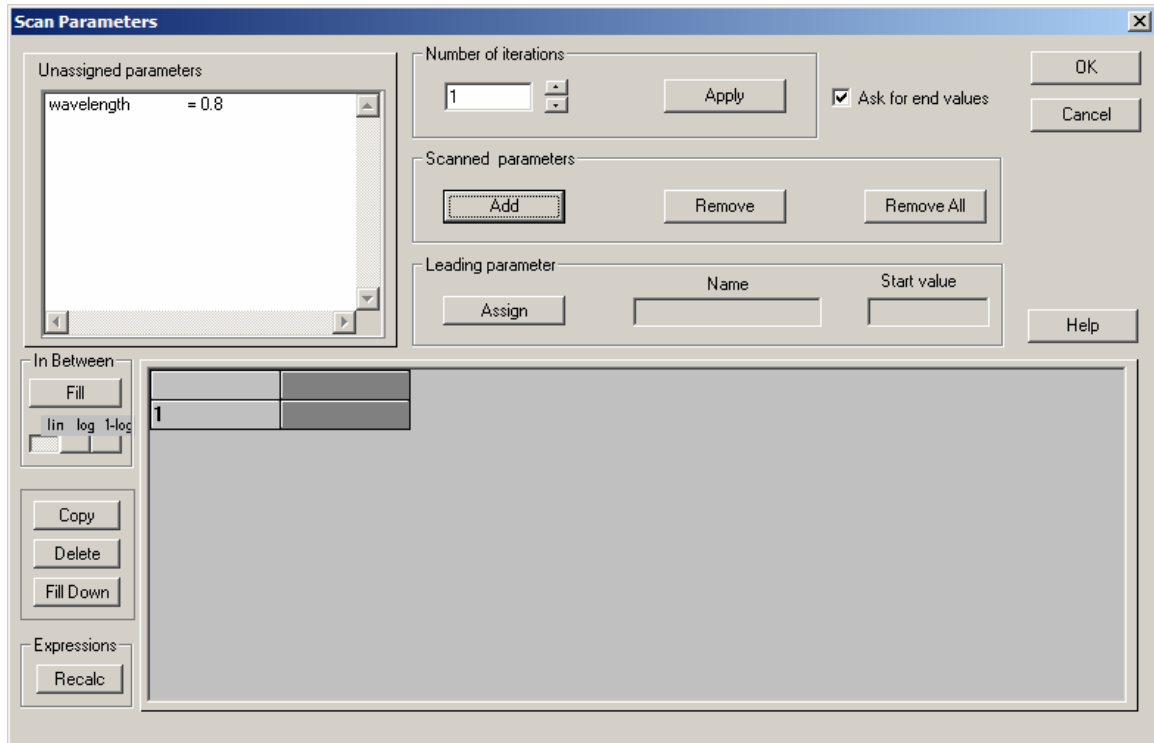


Figure 16 Scan Parameters dialog box

- 4 Select “wavelength” under **Unassigned Parameters** table, Click Add button to add variable wavelength as the scan parameter. Parameters wavelength will be listed in the bottom table.
- 5 Set **Number of Iteration** as 31 and click apply button. The wavelength table will be extent to 31 lines.
- 6 Click **Assign** button, parameter wavelength will be set as **Leading Parameter**
- 7 In the wavelength table, click “wavelength”, the whole wavelength column will be selected. Under wavelength column, in the line 31, Double click the box. And input iteration end value 1.4
- 8 Re-select the wavelength column, click **Fill** button will automatic generate the sampling point for each sweep wavelength value.
- 9 Click OK to close this **Scan Parameter** dialog box
- 10 Click **Calculate ADI** under simulation menu. In the Wavelength box input parameters name “wavelength”, select Full vector with initial excitation along Y. Set Complex Solver with boundary condition as TBC. Run the simulation will start the wavelength scan simulation, the cutoff wavelength can be obtained in this manner ( see figure 17)

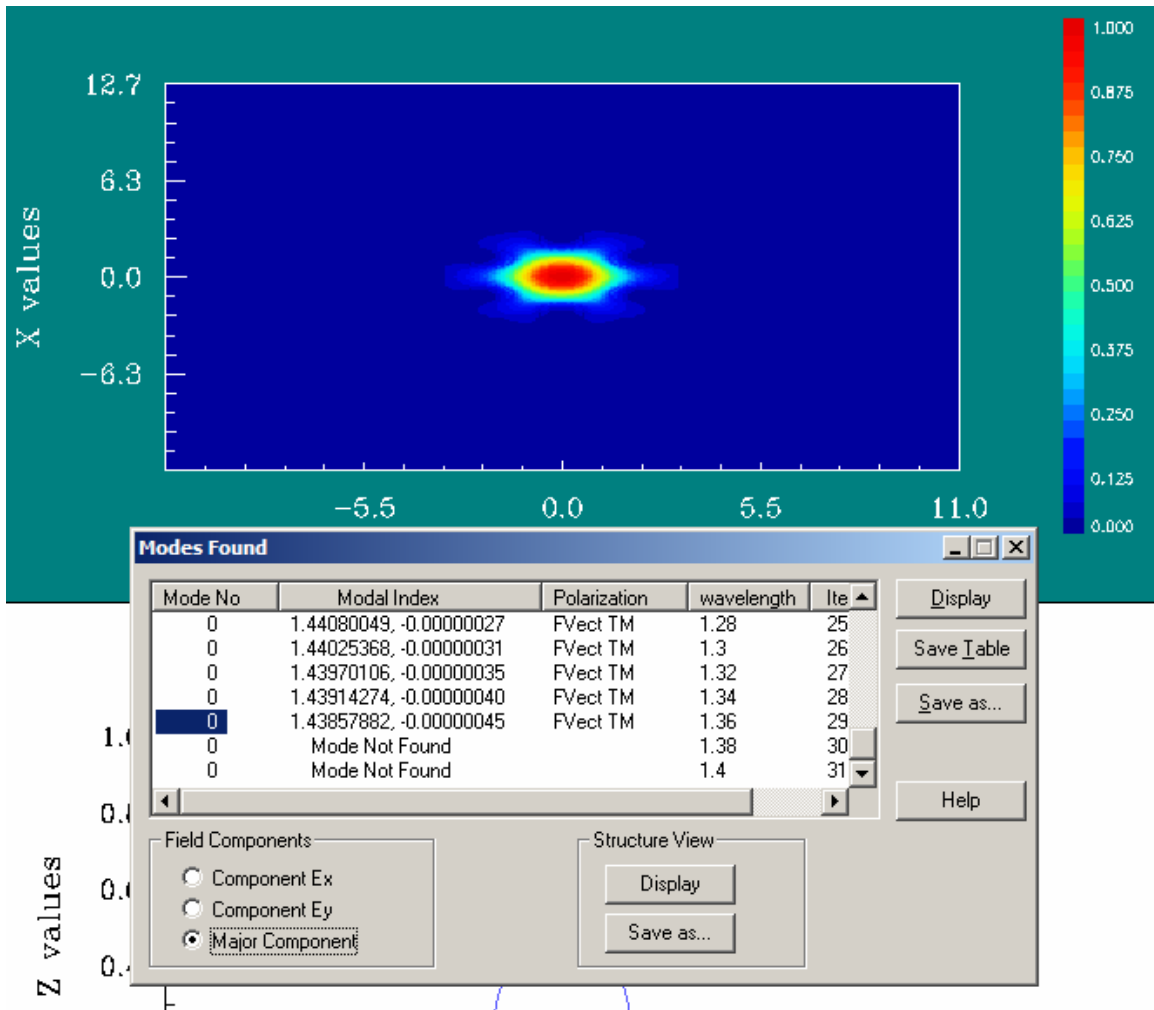


Figure 17 monitor the wavelength scan simulation