

# FAQ

FIMMWAVE Version 5.3

© 1997-2011 Copyright Photon Design



*propagating ideas*

34 Leopold St  
Oxford, OX4 1TW  
United Kingdom  
Phone + 44 1865 324990  
Fax + 44 1865 324991  
[info@photond.com](mailto:info@photond.com)  
[www.photond.com](http://www.photond.com)

.....	1
<b>1 Introduction - Structure of this Document.....</b>	<b>4</b>
<b>2 List of Keywords.....</b>	<b>4</b>
<b>3 Installation Problems .....</b>	<b>4</b>
3.1 On Windows 2000, I install the USB driver but FIMMWAVE still fails to work. What shall I do? .....	4
3.2 I get the error "freopen stderr" when trying to install Python. ....	5
<b>4 Mode Solving (FIMMWAVE) Problems .....</b>	<b>5</b>
4.1 Can FIMMWAVE estimate mode attenuation? What is the difference between modeLossOV and modeLossEV? .....	5
4.2 Does the WG scanner work with complex waveguides? .....	6
4.3 Can one study diffraction effects using FIMMWAVE? .....	6
4.4 When evaluating the mode profiles, what is the exact definition of field intensity? ..	6
4.5 When manual scanning for modes, what is the difference between the blue line and the red line? .....	6
4.6 How does increasing the size of the computational window affect the calculation of the effective index of a mode? .....	6
4.7 Can FIMMWAVE model Thermal-Optical and Electro-Optical Effects?.....	7
4.8 In FIMMWAVE, how do I know if my mode is guided or radiative? .....	7
4.9 Is there a method to tell the difference between real modes (i.e. modes that truly exist and are index-confined) and spurious modes (i.e. modes that are confined by the boundary of the calculation window)? .....	7
4.10 When using the FMM Solver, how does the time it take for the MOLAB to find a 2D mode vary with respect to increasing the number of 1d modes per slice? .....	8
4.11 How do I know if the modes I have built are accurate? .....	8
4.12 How can I speed up the time it takes for the MOLAB to build my basis set of modes? ..	8
4.13 How do I choose the size of my waveguide cross-section? Where shall I locate the boundaries.....	9
4.14 I am having trouble find bend modes of a curved waveguide. Any suggestions?..	9
4.15 When I use PMLS on the top (or bottom) boundary with the FMM Solver I get incorrect results. Why? .....	10
4.16 When should I use the electric walls, magnetic walls, and impedance walls .....	10
<b>5 Propagation (FIMMPROP) Problems .....</b>	<b>10</b>
5.1 In FIMMPROP, is it possible to plot the power in a mode Vs the propagation distance? .....	10
5.2 How do I copy a component from one project to another? .....	11
5.3 What is the difference between the "Paste" and the "Paste As Reference" Operations? .....	11
5.4 Does a zero-length taper behave in exactly the same manner as a butt joint? .....	11
5.5 Why can I not propagate through a taper when "Autorun" is off in the taper's MOLAB options? .....	11
5.6 How does the calculation time of a propagation calculation increase when the number of sections and joints increases? .....	11

5.7	How does the calculation time of a propagation calculation increase when increasing the number of modes used? .....	11
5.8	If I increase the number of modes in my propagation calculation, will my result be more accurate?.....	12
5.9	Can I exploit the symmetry in devices that are symmetrical in the propagation direction? .....	12
5.10	If I increase the number of components in a project, how is the memory affected? .....	13
5.11	Which solvers should I use to model tapers? .....	13
5.12	Is radiation loss taken into account in FIMMPROP? .....	13
5.13	How is radiation loss calculated?.....	14
5.14	Can FIMMPROP model polarisation effects in fibers? .....	14
5.15	What is the difference between the Normal Fresnel, the Oblique Fresnel and the Complete and Special Complete overlap methods? .....	14
5.16	Which is the most accurate overlap method?.....	14
5.17	When can I use the “special complete” method? .....	15
5.18	How do “Hard wall” Boundary Conditions affect my calculation? .....	15
5.19	How Can I prevent Reflection at the “Hard walls” from affecting my Result?.....	16
5.20	If I change a parameter in one element of my device, does FIMMPROP have to recalculate the fields in all of the elements of the device?.....	16
5.21	How can I control the phase of my input modes?.....	16
5.22	I have a structure with multiple input branches. How can I inject power in each branch with a phase difference? .....	17
<b>6</b>	<b>Miscellaneous Problems .....</b>	<b>17</b>
6.1	To use FIMMWAVE’s TCP/IP interface, do I need to use C++ or will any client based code work? .....	17
6.2	Can I display an S-matrix, then save it to a file via the command line (TCP/IP) interface?.....	18
6.3	If I have measured data of the refractive index profile of a device, either in ASCII format or in EXCEL format, can I import the information into FIMMWAVE?.....	18
6.4	Can I purchase FIMMPROP without purchasing FIMMWAVE? .....	18
6.5	Where do I find FIMMPROP?.....	18
6.6	How do I start FIMMWAVE on a port number so that I can communicate with it via TCP/IP? .....	18

# 1 Introduction - Structure of this Document

This document contains an easy-to-use list of answers to questions that are frequently asked by the application user. Two main features allow the reader of this document to find an answer to their question:

- Questions are grouped into broad topics. A list of the topics as well as the questions within that topic is given in section 3.
- Each question in this document has a set of associated keywords that are usually written next to the question. These keywords are written in **red**. Thus allowing you to take advantage of the advanced search facilities Adobe Acrobat, to find the answer required. Section 2 contains a list of keywords that can be used in this searching process.

Sections 3 onwards contains the questions and their answers grouped under the relevant headings.

## 2 List of Keywords

1-E

1d modes, 2d modes, Accuracy, Bi-directionality, Boundary Conditions, Calculation Speed, Client/Server, Command Line, Complete Method,

F-L

Farfield, Free Space Joint, freopen stderr, Hard-Wall, impedance walls, Impedance Factor, Joint,

M-Q

metal wall, magnetic wall, Normal Fresnel Method, MOLAB, Polarisation, PML, Propagation, PythonWin

R-Z

Radiation, Reflection, Symmetry, Taper, USB,

## 3 Installation Problems

### 3.1 *On Windows 2000, I install the USB driver but FIMMWAVE still fails to work. What shall I do?*

**Keywords:** **USB**

The following is an extract from an email regarding this matter which was sent to Photon Design by the company that manufactures both our parallel port and USB keys.

“...With regards to the problem of the USB drivers not being installed correctly. With Windows 2000, it is recommended that all USB devices must be removed from the PC ports, before the system drivers are installed. In addition, it is found that if you have previously installed another USB driver on your machine, then the Rainbow USB drivers may clash with these drivers. The best way to overcome this problem is to go into device manager, making sure that you are not getting the yellow question or exclamation mark next to the USB drivers. If you do, you will need to un-install the Rainbow USB system drivers. Now sometimes the Rainbow un-installation program claims the drivers have been removed. We have found that this is not the case, and you need to manually remove the drivers yourself. Then re-install the drivers from beginning.

We have reported this problem to our engineers who are looking into this matter and working on a fix.”

If you have problems uninstalling the USB driver such that upon reinstallation, your system still believes that a USB driver still exists, then it has been reported that the following may solve your problem:

- In Windows Explorer, navigate to %root%\WINNT\system32\drivers
- Remove the file SNTNLUSB.SYS
- Reboot your machine
- Reinstall your USB driver.

### 3.2 *I get the error “freopen stderr” when trying to install Python.*

**Keywords:** `freopen stderr` PythonWin

This is a problem with the PythonWin version 201 installation system (pywin32-201.win32.py2.3.exe) PythonWin is a Windows environment for writing and running Python scripts that can be used to automate FIMMWAVE. It is not a Photon Design product and is distributed for your convenience on the installation CD.

During PythonWin installation the install system needs to write some temporary files. The following suggestions should help:

Ensure the TEMP environment variable points to a directory that is a) writeable, b) is not full, c) does not have any spaces in its path or name. We use “c:\temp” here.

Sometimes if the TEMP directory contains temporary files from a previous PythonWin install then the new install may fail - delete all temporary files from the TEMP directory. Temporary files typically start with a “~” and or have a .tmp suffix – you can safely delete both sorts of file. (In fact you can safely delete all files from a TEMP directory once all your applications are closed – as by definition they should be temporary anyway!)

Set the TMP environment variable to the same as the TEMP variable.

No try to re-install PythonWin (run the pywin32-xxx.win32-pyxxx.exe install file in the PythonInstall directory on your CD).

If you wish, you can point TEMP and TMP back to their original locations once the PythonWin installation was successful.

If these measures fail, copy the pywin32-201.win32.py2.3.exe file to a writeable directory on your hard disk and run the install from there.

Note that this installation problem is fixed in the latest version of PythonWin (ver204) – with the installation file pywin32-204.win32.py2.3.exe. You can download this from <http://sourceforge.net/projects/pywin32>.

## 4 Mode Solving (FIMMWAVE) Problems

### 4.1 *Can FIMMWAVE estimate mode attenuation? What is the difference between modeLossOV and modeLossEV?*

**Keywords:** FIMMWAVE

There are two ways that FIMMWAVE can model modal attenuation.

The parameter **modeLossOV** can be used to approximate mode losses due to material absorption. The mode loss for this parameter is calculated by performing an overlap between the loss profile and the field profile, which allows you to obtain mode loss values even with real solvers. The accuracy of **modeLossOV** depends on the number of grid points (nx, ny) used to perform the overlap calculation. When using a real solver it also depends on how well the real field profile can represent the lossy mode (which should have a complex field profile).

The formula used for the calculation of **modeLossOV** is given in section 13.2.1 "Mode loss" of the FIMMWAVE manual.

The parameter **modeLossEV** gives you a more rigorous value for the mode loss, which is calculated directly from the imaginary part of the effective index. The parameter

**modeLossEV** includes losses due to material absorption and optical leakage (when working with PMLs or transparent boundary conditions).

In summary:

**modeLossOV** = loss due to material absorption, with real and complex solvers

**modeLossEV** = loss due to material absorption and optical leakage, with complex solvers

When using a complex solver in absence of PMLs or transparent boundaries, you should find that **modeLossOV** converges towards **modeLossEV** when nx and ny are increased.

## 4.2 Does the WG scanner work with complex waveguides?

Keywords: FIMMWAVE

Yes it does.

## 4.3 Can one study diffraction effects using FIMMWAVE?

Keywords: FIMMWAVE, Farfield

Yes! FIMMWAVE provides a rigorous solution of Maxwell's Equations. Diffraction effects are modelled by including higher order modes in your structure. You can plot the farfield of these modes to ascertain their angular deviation. This can be done via the *Inspect Mode Panel*.

## 4.4 When evaluating the mode profiles, what is the exact definition of field intensity?

$$I(x, y) = \text{Re}\{\epsilon(x, y)\} \frac{1}{4} \mathbf{E}(x, y) \cdot \mathbf{E}(x, y)^* + \frac{\mu_0}{4} \mathbf{H}(x, y) \cdot \mathbf{H}(x, y)^*$$

This is the local *energy density* and has units nJ/m<sup>3</sup>

In contrast, the power is defined by the Poynting vector:

$$\underline{P}_z(x, y) = (\underline{E} \wedge \underline{H}) \cdot \underline{z}$$

## 4.5 When manual scanning for modes, what is the difference between the blue line and the red line?

Keywords: FIMMWAVE

The FMM Solver algorithm generates a *characteristic matrix*, which is a function of the guess eigenvalue. A mode is located when this matrix is singular – i.e. when the determinant becomes zero. The blue and green lines respectively represent the real and complex part of the determinant and go to zero at a mode. The red line is the inverse of the blue line and goes to infinity at a mode. It is a matter of preference which line you use to work with.

## 4.6 How does increasing the size of the computational window affect the calculation of the effective index of a mode?

Keywords: FIMMWAVE, Accuracy, 2d modes, 1d modes, Hard-Wall, Boundary Conditions

If the walls were initially placed at a distance where the Ex and Ey fields of the guided mode and their gradients are zero then the effective index should not change. If the mode is radiative, then (in general) the difference between the effective indices of consecutive modes becomes smaller with increasing computational window size.

If it is found that the effective index of a guided mode changes when increasing the computational window size, there are two possible explanations:

1. Before the change, the close proximity of the hard-wall boundaries to the guiding layer(s) was having an adverse effect on the calculation of the mode's effective index.
2. The increase in computational window size may require you to increase the calculation parameters: typically you may need to increase (1d)nmodes for the FMM Solver, nx and ny for the FDM Solver, nxgrid and nygrid for the FEM Solver. In the case of the FMM Solver, the number of 1d modes (**1dnmode**) needs to be increased to resolve higher spatial frequencies that have now been introduced.

#### **4.7 Can FIMMWAVE model Thermal-Optical and Electro-Optical Effects?**

**Keywords:** FIMMWAVE

Yes it can, provided you have the Thermal and Electro-Optic module.

#### **4.8 In FIMMWAVE, how do I know if my mode is guided or radiative?**

**Keywords:** FIMMWAVE, Radiation

If confinement in the waveguide is entirely due to the index contrast between the core and the cladding (as it is the case in a ridge or a buried waveguide), the easiest way is to simply look at the effective index of the mode. If it is larger than the refractive index of the cladding/substrate then the mode cannot propagate in this region. It is thus guided. If the effective index of the mode is less than the refractive index of the cladding/substrate then it is a radiation mode.

In the case of a rib waveguide, it is not possible to compare the effective index with the cladding index for the lateral radiation.

With the real FMM Solver, you can use the **Side Loss** value reported in the **Inspect Mode** panel (see the FIMMWAVE manual section 13.3, also 2.5) – this is a measure of the power incident on the left and right boundaries. A mode with a non-zero side loss is a leaky mode. But remember that the side-loss does not detect potential radiation up and down so that a zero side-loss mode might still not be a proper guided mode in principle, though in practise it would be an unusual geometry that caused radiation up and/or down but had zero sideways radiation.

In addition, if you put the left and right walls reasonably far away, you will be able to easily recognise the radiating modes - you will see the standing waves.

A more rigorous – although albeit tedious – possibility is to compare the effective index of the eigenmode with the effective index of the vertical fundamental 1D modes (TE and TM) in the side slices. This can be done rather easily with the FMM Solver by checking the effective index of the 1D modes in “Engine Info”. If the 1D modes have effective indices that are smaller than the effective index of the 2D mode, then it will be guided. If the 2D mode has an effective index smaller than the effective index of one of the 1D modes, then the field component corresponding to the polarisation of the 1D mode (Ex for TE, Ey for TM) will be leaky.

Caution: the dominant field might sometimes look bound, but you must check also the minority field - sometimes the mode can radiate via the minority field (Ey for an Ex dominant mode).

#### **4.9 Is there a method to tell the difference between real modes (i.e. modes that truly exist and are index-confined) and spurious modes (i.e. modes that are confined by the boundary of the calculation window)?**

**Keywords:** FIMMWAVE, Radiation

Yes! See question 4.8 of this document.



#### 4.10 When using the FMM Solver, how does the time it take for the MOLAB to find a 2D mode vary with respect to increasing the number of 1d modes per slice?

**Keywords:** FIMMWAVE, Accuracy, 2d modes, 1d modes, MOLAB

If you double the number of 1d modes per slice **{(1d) nmode}**, the time taken is about eight times as long (i.e. it has an  $N^3$  relationship).

#### 4.11 How do I know if the modes I have built are accurate?

**Keywords:** FIMMWAVE, Accuracy, 2d modes, 1d modes, MOLAB

There are two ways of knowing if your modes are accurate:

1. The most efficient way to check to see if the mode profiles are correct is to use the **Orth Test** button on the **Mode Finder Panel**. This calculates a matrix whose elements are the overlap integrals of the modes with each other. In principle, this should give the identity matrix. However, due to numerical errors, the off-diagonal terms may not be zero. Upon pressing the **Orth Test** button, a value is returned. This value is the maximum off-diagonal term. In practice, you should be aiming at keeping this value below  $1e-3$  (0.1%). If this is not acceptable, then you may need to increase **(1d)nmodes** (if applicable) and/or the mode profile resolution parameters **nx** and **ny**.

2. To check to see if modes built using the **FMM Solver** have the correct effective indices, (i.e. if **(1d)nmodes** is large enough), you could also use the **(1d)nmodes Error Analysis** button in the **Engine Info** panel which you can access from the **Mode Finder**. This will display a graph of the effective index vs **(1d)nmodes**. When this converges, the effective index is accurate. See section 10.9 in the FIMMWAVE manual for more details.

3. With the FEM and FDM Solvers, it is recommended that you check the convergence of the effective index of the modes when the computation mesh resolution is modified: **nxres** and **nyres** parameters with the FEM Solver and **nx** and **ny** parameters with the FDM Solver.

#### 4.12 How can I speed up the time it takes for the MOLAB to build my basis set of modes?

**Keywords:** FIMMWAVE, Calculation Speed, MOLAB, Symmetry

You should design your waveguide cross section so that it is as small as possible while keeping the hard walls far enough away so that the exponential tails of the guided mode(s) have decayed to zero before reaching them. You can check this using the Plot Section tool in the Inspect Mode panel.

You need to bear the following two principles in mind:

1. **Keep box size as small as possible**
2. **Boundary must not disturb the guided modes**

You can apply the following rule of thumb:

**The boundary is far enough away if  $\psi = 0$  and  $d\psi/dL = 0$   
at the boundary for the guided modes,**

where  $\psi$  is the main component of the electric field and  $L = x$  for the LHS and RHS boundaries,  $L = y$  for the top and bottom boundaries.



With the FMM Solver, doing this in the y direction allows you to minimise the number of 1d modes needed to build an accurate set of 2d modes. The calculation time for building these modes is proportional to this number cubed!

In addition to this, an extremely powerful way to minimise computational time even further is to take advantage of any **symmetry planes** in your cross section. If your structure has symmetry planes in either the x or y direction (or both) then you should set the mode solver options accordingly. (When setting this you also need to ask yourself if you need to model both types of polarisation – it is common that you do not). See section 5.5.5 in the FIMMWAVE manual.

With the FMM Solver, if you are able to take advantage of the symmetry in the vertical direction this will allow you to reduce the number of required 1d modes by a factor of two. Thus speeding the calculations up by a factor of eight due to the reasons described above. Taking advantage of the symmetry in the horizontal direction will also speed up the calculations up by a factor of 2. This is because we are halving the number of slices in the cross section. Thus, we are halving the number of overlap matrices (of 1d modes) at each slice interface.

With the FDM and FEM Solvers, the computation will be about three times faster per symmetry plane.

#### ***4.13 How do I choose the size of my waveguide cross-section? Where shall I locate the boundaries.***

Please see first paragraph of question 4.12 above.

#### ***4.14 I am having trouble find bend modes of a curved waveguide. Any suggestions?***

**Keywords:** FIMMWAVE, PML

There are three bend solvers in FIMMWAVE: the complex FMM Solver, the complex FDM Solver and the FEM Solver.

The complex FMM Bend Solver relies on cylindrical co-ordinates and provides the most rigorous solution. The complex FDM Bend Solver relies on an approximation to cylindrical coordiantes and provides in many cases good results. The FEM Bend Solver relies on a conformal transformation, which will provide less rigorous results than the other two but can be useful for specific geometries in which the triangular mesh is better suited.

To calculate bend losses, you can use PMLs with all three solvers or transparent boundary conditions with the FMM Solver.

For a rectangular geometry, the best results are given by the complex FMM Solver with transparent boundary conditions. The MOLAB does not work with transparent boundaries, and the complex FMM Solver is generally slow in comparison with the other ones. We therefore recommend that you follow the procedure below to calculate your modes:

- a) Use the complex FDM bend solver to find the modes of interest with Transparent boundaries. Use a relatively high spatial resolution (e.g. 200 by 200) as this will increase your chances of polishing the modes.
- b) In addition to the guided modes, you will find some spurious modes in the transparent boundary condition. Make sure that you delete these modes before proceeding further.
- c) Close the Mode Finder and save the mode list.
- d) Switch to using the Complex FMM Solver with a relatively high number of 1D modes (again to increase the chances of polishing).
- e) Use the Polish function.

If the mode found by the FDM Solver is an adequate approximation to the bend mode then this process will be much faster than using either the MOLAB (with a PML layer) or the Manual Scanner.

#### 4.15 When I use PMLs on the top (or bottom) boundary with the FMM Solver I get incorrect results. Why?

**Keywords:** FIMMWAVE, PML

PMLs may produce inaccurate results when they are present at either the top or the bottom boundary for the complex FMM Solver.

Indeed, the larger the PmlWidths, the less the basis set of 1D modes is able to accurately represent the EM fields within the structure. Thus, for example, using these absorbing layers to investigate the loss of a “leaky mode” in the vertical direction has to be undertaken with care. This is not true for pmls in the horizontal direction.

If you wish to account for absorption in the vertical direction with the complex FMM Solver, then we recommend that the whole structure is rotated 90 degrees. Thus the PML layer can be placed on either the left or right boundary and not on the top or the bottom. See section 4.3 in the Fimmwave manual for more details.

#### 4.16 When should I use the electric walls, magnetic walls, and impedance walls

**Keywords:** FIMMWAVE, boundary conditions, electric wall, magnetic wall, impedance walls, Impedance Factor.

When solving for the guided mode of a waveguide usually you want the walls to be far enough away so that the boundaries do not perturb the mode. This means in general that all the electromagnetic fields and their gradients in the direction of the wall should be zero at the wall. E.g. for a TE-like mode, where  $E_x$  and  $H_y$  are dominant, then  $E_x$ ,  $H_y$ ,  $dE_x/dx$ ,  $dH_y/dx$  should be zero at the left and right walls. If this condition is met then it does not matter whether you use metal or magnetic walls.

In contrast, the electric and magnetic walls are useful for imposing symmetry planes to reduce computational effort. E.g. if you have a waveguide that is laterally symmetric and you want to find the zero-order TE-like mode, then  $E_x$  and  $H_y$  of this mode is symmetric about the centre line. Therefore you can apply a metal wall at the centre and remove the right half of the structure – the metal wall pulls  $dE_x/dx$  and  $dH_y/dx$  to zero. The table below outlines the effect of the two boundary conditions. “n” in the gradients indicates the direction normal to the wall.

	Parallel-E	Perpendicular-E	Parallel-H	Perpendicular-H
Electric Wall	$E_{//} \rightarrow 0$	$dE_{\perp}/dn \rightarrow 0$	$dH_{//}/dn \rightarrow 0$	$H_{\perp} \rightarrow 0$
Magnetic Wall	$dE_{//}/dn \rightarrow 0$	$E_{\perp} \rightarrow 0$	$H_{//} \rightarrow 0$	$dH_{\perp}/dn \rightarrow 0$

Impedance walls: These are supported only by the FMM Solver. They provide a generalisation of the magnetic and electric walls – i.e. can provide anything in between a perfect electric and a perfect magnetic wall. In principle an impedance wall will always perturb a guided mode less than a metal or magnetic wall – you can choose an Impedance Factor that does not perturb the evanescent tail of a guided mode so that the evanescent tail appears to be unperturbed by the walls. However there is no simple way of choosing the Impedance Factor – you have to do so by trial and error.

## 5 Propagation (FIMMPROP) Problems

### 5.1 In FIMMPROP, is it possible to plot the power in a mode Vs the propagation distance?

**Keywords:** FIMMPROP

Yes, from the component window, select **View\Z Profiles\Mode Power**. You can now plot the modepower of the modes of interest with respect to the propagation distance.

## **5.2 How do I copy a component from one project to another?**

**Keywords:** FIMMPROP

Multiple projects can be loaded simultaneously, and will be displayed as such in the project tree. Each node of the project tree has copy and paste functionality. Just right-click on the node that is to be copied on the pop-up menu, select **copy**, move it to the desired project by right clicking on the project node and selecting **paste**. The mode lists of each waveguide (if already calculated) will also be copied.

## **5.3 What is the difference between the “Paste” and the “Paste As Reference” Operations?**

**Keywords:** FIMMPROP

When you use “Paste”, FIMMPROP inserts a new copy of the sections, joints, etc. that were on the clipboard into the destination device. However, when you use **“Paste As Reference”** a reference to the original component is pasted. If the original is changed, then the destination device changes. This can speed up design and calculation time you can easily split a complicated calculation into several smaller steps. You then can set parameters correctly in each of the referenced components such that when you know their accuracy is correct, combine them into the final device without the need for any recalculation.

## **5.4 Does a zero-length taper behave in exactly the same manner as a butt joint?**

**Keywords:** FIMMPROP, Accuracy, Taper

The simple answer is no! The butt joint and the taper use different algorithms for their evaluation. However, as the resolution parameters in the taper algorithm are increased, the two should converge upon the same result. The Order-Zero taper “integration order” will converge faster for short tapers than the Order-One.

## **5.5 Why can I not propagate through a taper when “Autorun” is off in the taper’s MOLAB options?**

**Keywords:** FIMMPROP, Propagation, Taper, MOLAB

The taper algorithm needs to find modes along the length of the taper – so it needs to run the MOLAB automatically as needed; switching autorun off would prevent this.

## **5.6 How does the calculation time of a propagation calculation increase when the number of sections and joints increases?**

**Keywords:** FIMMPROP, Propagation

Generally, if the modes are initially built in each section of the device, then the calculation time varies between  $N^2$  and  $N^3$  where  $N$  is the number of sections in the device. See question 6.7 for a more detailed breakdown.

## **5.7 How does the calculation time of a propagation calculation increase when increasing the number of modes used?**

**Keywords:** FIMMPROP, Propagation, Taper, 2d modes, 1d modes

There are (at most) four stages to a FIMMPROP calculation. Let  $N$  be the number of modes:

- 1.) Finding the modes in each section. This varies linearly with  $N$  as you might expect.
- 2.) Building the mode profiles. This also increases linearly with  $N$ .
- 3.) Calculating an overlap matrix at each join. This varies as  $N^2$ .
- 4.) Combining the scattering matrices to give the final result. This varies as  $N^3$ .

Clearly, different calculation methods impose different time penalties for each stage. For example, for quasi 2D propagation calculations the analytic “**complete (1d) method**” can be used and accordingly the mode profiles do not need to be built so stage 2 of the calculation takes no time at all. In addition, due to the fast analytical integration, stage 3 takes very little time. Therefore, stage 4 will quickly dominate for increasing  $N$ .

In the case when the FMM Solver is used, stages 1, 2 and 3 may take a considerable time, but 4 will be negligible in comparison (unless  $N$  were very high).

These rules are general guidelines – the different algorithms for bends, periodic elements etc all have computational overheads of their own.

### **5.8 If I increase the number of modes in my propagation calculation, will my result be more accurate?**

**Keywords:** FIMMPROP, Propagation, Accuracy, 2d modes, 1d modes

In many cases, the answer is yes, however there are a few points worth mentioning:

- 1.) Any propagation calculation will not be accurate if the modes in each section of the device are not accurate. See question 4.11 for more details on how the accuracy of your modes can be checked.
- 2.) Joint calculation accuracy: When using the **Complete overlap** joint method, increasing the number of modes in your device will increase the accuracy of the joint calculation. (Note that this is strictly only true if adjacent waveguides have identical computational windows and there are no offsets or tilts.) The **Fresnel** methods, on the other hand, are approximate methods, so convergence to the correct answer is not guaranteed. See question 5.16 for more details on joint calculation methods.
- 3.) If you are simulating a taper, you need to include all modes to which light is coupling into in reality – the faster your taper (less adiabatic), the more modes you will need. (Though, typically only a few modes of each polarisation are needed for near adiabatic tapers.)

Sometimes, adding too many modes can make matters worse – higher order modes represent radiation modes. In theory these radiate away from the device of interest, however, due to the hard walls surrounding the computational window these modes may be reflected off the boundaries back into the device adversely affecting the results. If these modes are excluded the radiation will simply be lost from the calculation. Always think about the physics – what would the light do in reality? Would energy in a particular mode be important for your simulation, or would it radiate away anyway.

### **5.9 Can I exploit the symmetry in devices that are symmetrical in the propagation direction?**

**Keywords:** FIMMPROP, Symmetry

The answer to this question depends on the type of device you are simulating. If your device merely uses just simple sections and simple joints then you should use the same waveguide

files on either side of the symmetry plane. Thus, the basis set of modes for these sections will not have to be re-built.

If your device is periodic in the propagation direction, then it is advisable to use the **FIMMPROP Periodic Section** to define your device.

If your device is a bend or a taper, then at present there is no way of exploiting symmetries in the propagation direction (unless of course you define your device section by section). Future versions of FIMMWAVE / FIMMPROP will address this issue.

### **5.10 If I increase the number of components in a project, how is the memory affected?**

**Keywords:** FIMMPROP

Upon the evaluation of a component, everything remains in memory until you either close the project or reset the component (/Utils/Reset...). This can include mode profiles, S-matrices for every joint, field data, etc. Therefore, there is a huge memory penalty if many components are evaluated in the same project.

### **5.11 Which solvers should I use to model tapers?**

**Keywords:** FIMMPROP, Taper

For 1D+Z modelling, you can use the Quasi-2D (XZ) or (YZ) Effective Index Solver.

For fully 3D problems, we recommend that you start by using the FDM Solver as it will generally calculate large numbers of modes quickly. Besides, it guarantees not to miss any modes, which can be very useful when modes need to be calculated at many different positions. Another benefit of the FDM Solver is that it can use the **special complete** overlap method, which provides a very fast and accurate calculation of the overlaps in the taper at a low cost in memory.

Once you have obtained a result with the FDM Solver, you can try and see whether the result could be improved by using the FMM or FEM Solver.

If the FMM Solver provides you with more accurate mode calculations, you can also use it for taper modelling. Please note that in the case of the FMM Solver you can use the **EVScan** to polish the modes from one end of the taper to the other – this is usually faster than calculating the modes at each discretisation step. It is recommended that you check how easy it is to polish your modes using a WG Scanner beforehand – please see section 7.2 of the FIMMPROP manual for more details.

### **5.12 Is radiation loss taken into account in FIMMPROP?**

**Keywords:** FIMMPROP, Propagation, Radiation, Hard-Wall Boundary Conditions

In FIMMPROP, “hard-wall” boundary conditions are implemented, thus radiation is modelled as a discrete set of radiation modes. In your calculation, you have two options (in general) to account for radiation loss.

- 1.) In the basis set of modes for each section of the device, you include only the guided modes and from the calculation diagnostics, you can evaluate how much power is lost from these guided modes.
- 2.) In the basis set of modes for each section of the device you include both the guided and radiation modes in your basis set, and then you can measure the power in the radiation modes to ascertain the radiation loss. (**/View/z-profiles/mode-power**)

Method 1 has the feature that when power is coupled out of the guided modes, it is lost from the calculation and so cannot be reflected from the “hard-wall” boundary conditions and then interfere with the rest of the calculation. This can sometimes be a disadvantage if those radiation modes are required to describe the interesting features of the device (e.g. if radiation modes are needed to describe the coupling between one guided region of a device and another).

When using “hard-wall” boundaries, method 2 may lead to the reflection of the radiating field by the boundaries: please see sections 5.18 and 5.19 below.

See also section 7.6.3 of the FIMMPROP manual for more details.

### **5.13 How is radiation loss calculated?**

**Keywords:** FIMMPROP, Propagation, Radation

The hard wall boundaries used in FIMMPROP cause the continuum of radiation modes to be broken up into a discrete set of equivalent “box” modes.

The computation does not distinguish between guided and radiation modes – to include the radiating light in your calculation, you just increase the number of modes used. The computation will automatically compute the coupling to the equivalent “box” modes and you can inspect the amount of power in these modes (e.g. **/View/z-profiles/mode-power**).

See question 5.12 of this document for more details.

### **5.14 Can FIMMPROP model polarisation effects in fibers?**

**Keywords:** FIMMPROP, Polarisation

Yes it can! Since FIMMPROP rigorously calculates the coupling between one mode with another using a vectorial formulation. All that is needed is the inclusion of modes of the desired polarisations.

### **5.15 What is the difference between the Normal Fresnel, the Oblique Fresnel and the Complete and Special Complete overlap methods?**

**Keywords:** FIMMPROP, Propagation, Accuracy, Joint, Normal Fresnel Method, complete Method

The **Complete** and **Special Complete** methods provide a rigorous solution of Maxwell’s equations. If an infinite set of modes were used on each side of the join then the calculation would be exact. In practice, it gives very good accuracy even for a modest number of modes. It is also more carefully formulated for large joint tilts.

The **Complete** method can be used at a joint or in a taper for any mode solver. The **Special Complete** method is engine-specific and can only be used in specific cases – please see section 4.2.2 of the FIMMPROP manual or question 5.17 below for more details.

The **Fresnel** methods use an approximate formula that can give the coupling between two modes independently of all others. They do not have the pre-requisite of needing a complete basis set like the **Complete Method**. They are therefore often found to give better accuracy when you are only considering one or two modes. On the other hand, it is not so rigorous in the treatment of large joint tilts.

### **5.16 Which is the most accurate overlap method?**

**Keywords:** FIMMPROP, Propagation, Accuracy, Joint, Normal Fresnel Method, complete Method



The **complete** method is the most rigorous overlap method for fully 3D structures. However, for it to be accurate, it requires a complete (i.e. infinite) basis set.

Of course, no calculation can use an infinite set of modes so this basis set is truncated in practice. Typically you should get reasonably good accuracy for the coupling between low order modes even when you have only a small number of modes in your basis set. For strongly confined waveguides, with large miss-match at the join, you will probably need rather more – you will have to check this for your own structures.

The **Special Complete** method is equivalent to the **Complete Method**. It is much faster, and has much lower numerical errors. Note that for quasi-2D structures (i.e. those that have their refractive index varying in the z-direction and only one other direction) calculated with the Effective Index Solver, the **Special Complete** method calculates the overlaps analytically. Please see section 5.17 below to find out when to use the **Special Complete** method.

The **Complete Methods** will only converge to the accurate solution if the computational windows of the adjacent waveguides have the same size, and if there are no offsets and tilts. In practice though, the complete methods will give satisfactory solutions even if these conditions are not met.

The **Fresnel methods** although only approximate, do not have the requirement of a near-complete basis set and so will (reasonably) accurately calculate a coupling coefficient between one mode and another. The Fresnel approximations break down when the two modes in question have high spatial frequencies.

In general, you will want to use the **Complete Methods**, as they guarantee convergence to the correct solution, however Fresnel methods are the only available option when using reflective layers. In addition, the oblique Fresnel method might converge with relatively few modes when large tilts are involved.

See also section 4.2.2 of the FIMMPROP manual for more details.

### 5.17 When can I use the “special complete” method?

**Keywords:** FIMMPROP, Propagation, 1dmodes, Joint

You can use the **special complete method** in a joint or a taper when the following solvers are used:

- the Effective Index Solver
- the FDM Solver except when HSymmetry or VSymmetry is set to both
- the FDM Fibre Solver
- the GFS Solver.

In all cases, you will need to ensure that the waveguide cross-sections on both sides (or throughout the taper) have the same dimensions and that the mode solver parameters are identical on both sides of the joint.

Please note that the special complete overlaps are not compatible with tilt angles at the joints.

See section 4.2.2 of the FIMMPROP manual for more details.

### 5.18 How do “Hard wall” Boundary Conditions affect my calculation?

**Keywords:** FIMMPROP, Propagation, Radiation, Reflection, Hard-Wall Boundary Conditions

Hard wall boundary conditions may affect your calculation if power is reflected from them such that there is spurious coupling of this power back into the guiding regions of your device. You should try to think about the physics – what would happen in reality with mirrors near your structure?



### **5.19 How Can I prevent Reflection at the “Hard walls” from affecting my Result?**

**Keywords:** FIMMPROP, Propagation, Radiation, Reflection, Hard-Wall Boundary Conditions

There are several things that you can do to prevent spurious reflections affecting your result.

If your structure is quasi-2D then a good solution is to implement absorbing boundary conditions (**pmls**) at the hard wall boundaries. These are set within in the Waveguide Editor (**Pml width (um)**), see FIMMPROP Manual section 4.9 for more details.

If your structure is 3D then the most obvious solution is to increase the size of the computational window such that the reflected light does not have time to propagate far enough to interfere with that in the core. However, this can inflict the following penalties:

- 1.) If you double the width of the device, to keep the same resolution of spatial frequencies, you have to not only double the x component of the mode profile resolution (**nx**) but also double the number of 2D modes (**maxNmodes**).
- 2.) If you double the height of the device, you have to double the y component of the mode profile resolution (**ny**) to keep the same resolution of spatial frequencies. In addition, if you are using the **FMM Solver**, you must double the number of 1d modes per slice for every section of your device.

An alternative solution for a 3D structure is to include only the guided modes in the basis set of modes for each section of the device.

### **5.20 If I change a parameter in one element of my device, does FIMMPROP have to recalculate the fields in all of the elements of the device?**

**Keywords:** FIMMPROP

No, FIMMPROP does not have to recalculate everything from the start. The scattering matrix formalism permits the following hierarchy to be implemented:

1. Calculate local modes
2. Calculate S-matrices of each section and joint
3. Calculate the S-matrix of the device.

If you change one parameter in one element, only the parts of the calculation that depend on that parameter need to be recalculated. For example, if you change the length of a section, then the modes are still valid and so do not need to be re-calculated. Likewise, if you change the parameters of a joint, the computations of the sections are not affected.

FIMMPROP takes full advantage of this hierarchy and will always try to use any part of a previous calculation that is still valid – this happens automatically. See section 4.3.1 “the automatic update scheme” in the FIMMPROP manual for more details.

### **5.21 How can I control the phase of my input modes?**

You can assign a specific phase to each input mode via the “vector” mode of the “Input Field Profile” panel. This will allow you to define your input field in terms of a vector of the input modes, the vectors being expressed in terms of real and imaginary parts.

A bit of maths is needed to get a relationship between the vector coefficients and the phase shift.

For each mode, you can for instance express the real and imag parts as a function of the input phase as follows:

$$\begin{cases} input\_phase = \arctan\left(\frac{imag}{real}\right), \\ imag^2 + real^2 = 1 \end{cases}$$

which leads to :

$$\begin{cases} real = \frac{1}{\sqrt{1 + \tan(input\_phase)^2}} \\ imag = \frac{\tan(input\_phase)}{\sqrt{1 + \tan(input\_phase)^2}} \end{cases}.$$

Please note that this formulation assumes that *input\_phase* belongs to the interval ]-90,90[ degrees. The power in the mode is arbitrarily chosen to be 1.

Please see also section 4.6 "Defining the input field" in the FIMMPROP manual.

## 5.22 I have a structure with multiple input branches. How can I inject power in each branch with a phase difference?

If your set of modes is fully decoupled and each mode corresponds to the excitation in a different branch then you can use the "vector" directly, using the equations given in question 5.21 above.

For instance, if you want to inject the first two modes in phase quadrature:

$$input\_phase(A) - input\_phase(B) = 90,$$

you can use the following input vector:

$$\begin{aligned} \text{mode A} &: \left( \frac{1}{\sqrt{2}}, \frac{1}{\sqrt{2}} \right) \\ \text{mode B} &: \left( \frac{1}{\sqrt{2}}, \frac{-1}{\sqrt{2}} \right) \end{aligned}$$

If you want to inject the first two modes in phase inversion:

$$input\_phase(A) - input\_phase(B) = 180,$$

you will not be able to use the equations above as you can no longer express both the input phases in the interval ]-90,90[ degrees. Instead you can simply use the following vector:

$$\begin{aligned} \text{mode A} &: (1, 0) \\ \text{mode B} &: (-1, 0) \end{aligned}$$

If your modes are decoupled but expressed as supermodes (i.e. you have symmetric and anti-symmetric modes but their effective indices are identical), you can use an I/O Port Section to inject power in each branch independently and then use the "vector" input as explained above.

## 6 Miscellaneous Problems

### 6.1 To use FIMMWAVE's TCP/IP interface, do I need to use C++ or will any client based code work?

**Keywords:** Client/Server, Command Line

You can use any language that can implement a TCP/IP connection. However, FIMMWAVE is shipped with PYTHON ([www.python.org](http://www.python.org)) and contains a library of useful functions for connecting to FIMMWAVE. This makes it incredibly easy for you. See the PYTHON section of the FIMMWAVE manual for more details.

## ***6.2 Can I display an S-matrix, then save it to a file via the command line (TCP/IP) interface?***

**Keywords:** Client/Server, Command Line

Yes you can! The TCP/IP interface can return whole matrices. An example of a command that returns a matrix is:

```
app.subnodes[1].subnodes[1].cdev.smat.lr
```

This returns the left to right transmission S-Matrix for a component that is referenced by its position in the project tree. If you use the PYTHON client, you can obtain this S-matrix as a PYTHON list, which can then be saved easily to file.

## ***6.3 If I have measured data of the refractive index profile of a device, either in ASCII format or in EXCEL format, can I import the information into FIMMWAVE?***

**Keywords:** Client/Server, Command Line

The MWG and FWG editors allow you to import refractive index profiles from your measurement data. Please see sections 3.3.5 (FWG) and 3.4.7 (MWG) of the FIMMWAVE manual for more details.

## ***6.4 Can I purchase FIMMPROP without purchasing FIMMWAVE?***

**Keywords:** FIMMPROP

No – FIMMPROP is an extension module to FIMMWAVE. You must have FIMMWAVE to use FIMMPROP.

## ***6.5 Where do I find FIMMPROP?***

**Keywords:** FIMMPROP

If you have the FIMMPROP module, then when you run FIMMWAVE, you are already running FIMMPROP. You merely have to open or create a new project. See the FIMMPROP manual for more details.

## ***6.6 How do I start FIMMWAVE on a port number so that I can communicate with it via TCP/IP?***

**Keywords:** Client/Server

If you use a Shortcut Icon to start FIMMWAVE then you can edit the target properties accordingly so that FIMMWAVE is started on a port number. E.g. Target = ....fimmwave.exe - pt 5101. Upon starting, you should see confirmation from the messages window that FIMMWAVE has started on your chosen port number.