Università di Parma – Dipartimento di Ingegneria dell'Informazione

COURSE OF "PHOTONIC DEVICES"

GUIDED MODES OF A STEP-INDEX OPTICAL FIBER
COMSOL MULTIPHYSICS 4.3

Enrico Coscelli (enrico.coscelli@unipr.it)

Fiber modeling

The definition of the model to be solved with COMSOL Multiphyisics consists in 4 basic steps:

1) Definition of the geometry. The cross-section of the fiber is draw or imported, to replicate the geometric characteristics of the real optical fiber.
2) Definition of the physical properties. Physical properties (i.e. refractive index) are set to each model subdomain and boundary conditions are applied.
3) Definition of the general simulation parameters, such as the wavelength and the number of solutions to be calculated.
4) Meshing. The computational window is discretized into a set of triangular elements. The discretization allows the PDE to be solved as a linear equation set. The distribution and density of the mesh elements are crucial for the precision of the results, especially in complex models. COMSOL provides automatic routines for mesh generation, with several parameters that can be changed to get an optimal discretization.

Solving

The problem is solved by the FEM solver integrated in COMSOL. A number of parameters can be set, such as the eigenvalue around which look for solutions.

Post-processing

Several quantities relative to the calculated solutions can be plotted. In our case, the most relevant are the electric field **E**, magnetic field **H**, Poynting vector **S**, electric displacement **D** and magnetic flux density **B**. Further processing is possible, such as calculation of the integral of these quantities over chosen subdomains.

Step-by-step simulation of a step-index optical fiber

1) Start COMSOL Multiphysics 4.3 (Tutti i programmi -> COMSOL 4.3-> COMSOL Multiphysics 4.3) and select `New` to open a new model.
2) From the model wizard, set `select space dimension` to 2D and press ⇨ to confirm.
3) Choose the `electromagnetic waves` module, `frequency domain (emw)` and confirm with ⇨.
4) In `select study type`, choose `mode analysis`. 🏁 to confirm.

The main GUI window will open, which is used to create the model and visualize post-processing data. Few options require configuration:

5) In the model builder, select geometry 1 and set unit to µm.
6) Expand node electromagnetic waves, frequency domain and select wave equation, electric 1. Set electric displacement field model to refractive index. This step allows to define the refractive index of each subdomain instead of relative electric permittivity.

It is now possible to draw the geometry of the step index fiber:

7) Right-click the node geometry 1 under the model builder. Select circle in the pop-up menu. In the settings window, edit radius and set it to **4.5 µm**.
8) Repeat step 7, and draw a new circle with a radius of **60 µm**.
9) Right-click on the node materials in the model builder and select material from the pop-up menu. This will tell COMSOL to define a new material to be used in the model.
10) Right-click on the node material 1 that we have just created. Select rename from the pop-up menu and change its name to "Silica Glass".
11) From the menu material -> geometric entity selection choose only the subdomain corresponding to the core of the fiber. Set the refractive index to **1.45**. Select add to material to apply the material "Silica Glass" to the core.
12) Repeat steps 9) – 11) to create a new material "Cladding Silica" with refractive index **1.442** and assign it to the cladding.

NOTE: The order of the nodes in the model builder is important! In case of multiple definitions COMSOL will consider only the last value ("override" flag will be shown).

The model of the fiber is now completely set. To solve it, we need to generate a mesh.

13) Select node mesh 1 from the model builder. The window mesh settings will open. Set element size to finer and press build all to generate the mesh.

It is now possible to start the solver to get the modes

14) Select the node study 1 -> step 1: Mode analysis from the model builder.
15) In the mode analysis settings window, find the study settings menu. Set desired number of modes to **10** and search for modes around to **1.448**. Set also mode analysis frequency to **c_const/1.55[µm]**, in order to obtain the solutions at λ = 1550 nm.
16) Select the node study 1 and click on compute to start the solver.

After calculations are finished, it is possible to choose the data to plot with model builder -> results.

Further steps

- Solve again at λ = 1030 nm and λ = 1850 nm. Do you notice any difference in the solutions? If so, why?
- Change the cladding refractive index and set it to 1.446. What do you expect to happen? Solve again at λ = 1030 nm and λ = 1550 nm.
- Which is the minimum cladding refractive index that allows propagation of the sole LP_{11} mode at λ = 1550 nm? Apply it to the model and solve again at λ = 1550 nm.
- Edit core radius and set it to 12.5 µm. Solve the model again at λ = 1550 nm, looking for 25 modes. Try to classify them into $LP_{\nu\mu}$ families. Which cladding index you need to obtain single-mode propagation?