%%%
%%%
%%%
%%%

Part I

I used COMSOL to simulate a simple 2D model, in which a plane wave propagates from air into a dielectric medium (n = 1.5). Here are some details:

1. Three subdomain ass shown in Figure 1.
2. Boundary conditions: Periodical condition (Floquet periodicity) at two sides. Scattering boundary conditions at the top and bottom. The plane wave is excited at the top.
3. Definition of the transmission: **-(Poyav_rfwh)/(188.5*cos(theta))**. The transmission variables ‘Transsy’ and ‘Transby’ are averaged in subdomain 2 and on line marked in Figure 1 respectively.

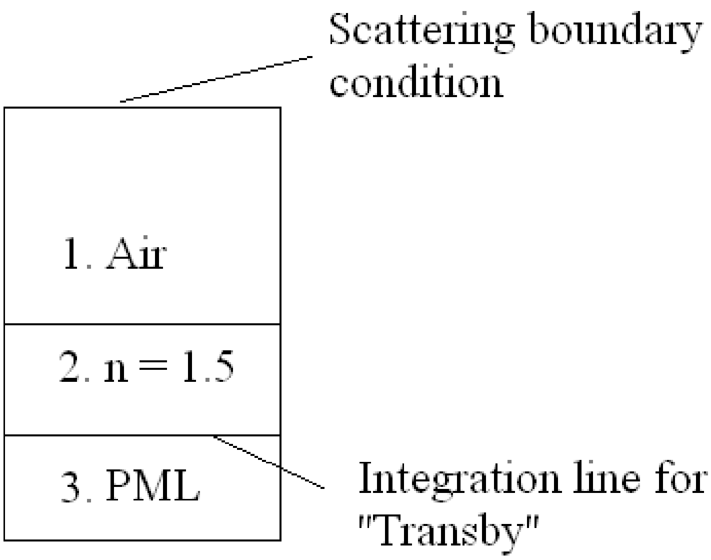
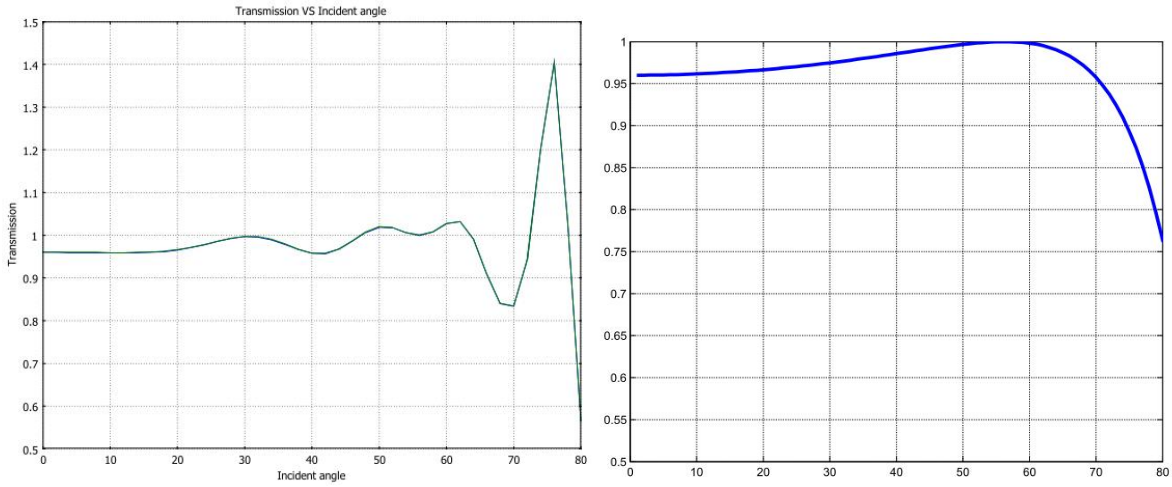


Figure 1. The model

Figure 2(a) shows the transmission changing with the incident angle, while figure 2(b) gives the theoretical results.



(a) (b)

Figure 2. Transmission curve calculated by (a) COMSOL; (b) formula.

I have two following questions:

1. There is much difference between COMSOL result and theoretical result at some incident angles (such as 76 degree). The difference is small only at small angles. Why does this happen?
2. There are also small difference between 'Transsy' and 'Transby'. They should be exactly the same. Why?

Do not suggest me to use 1D model or change the periodical condition on the two sides.

Looking forward to your reply!

%%
%%
%%
%%

Part II

There is another way to check the results.
I added two new variables 'Eaveb' and 'Eaves' in Options>Integration coupled variables>Bandary variables and Options>Integration coupled variables>Subdomain variables. (You can also find the two variables 'Transsy' and 'Transby' there). The definitions are as following

Eaves = normE_rfwh/377/A   Eaveb = normE_rfwh/377/L

Where A is the area of subdomain 2 and L is the length of the line between subdomain 2,3. The amplitude of the incident E-field is 377.

Eaves (Eaveb) are actually a normalized average values of electric field in subdomain 2 (on the line between subdomain 2,3).

Figure 3 shows the difference between COMSOL's result and the theoretic one. (For TM light, $E_T/E_0 = 2*n_1*cosi_1 / (n_1*cosi_1 + n_2*cosi_2)$)

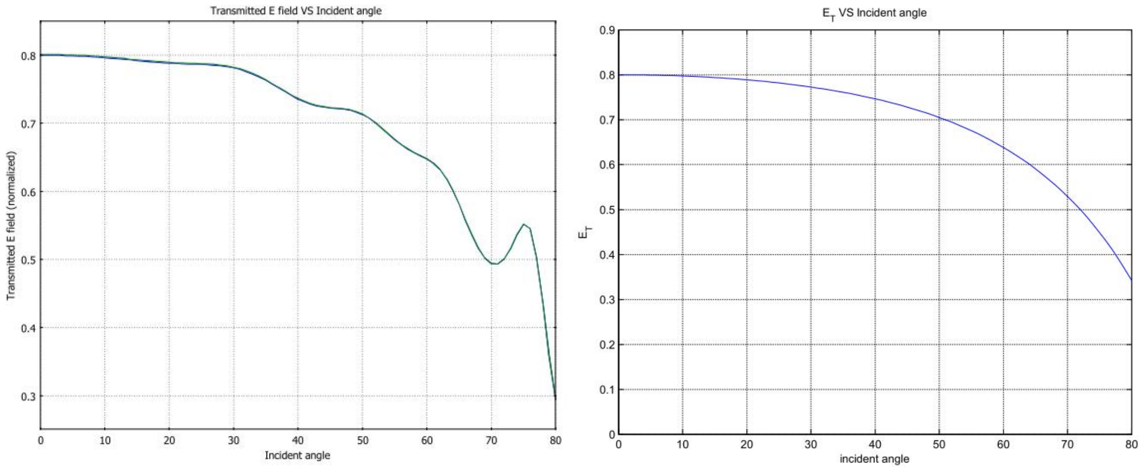


Figure 3. Transmitted E-field calculated by (a) COMSOL; (b) formula.

How to get the curve in Figure 3.

1. Open my mph file.
2. Refine mesh.
3. Solve> Solver parameters> Harmonic propagation>Parametric>General>Parameter values: change 0:40:80 into 0:2:80.
4. Solve
5. Postprocessing> Global Variables Plot> Quantities to plot: Eaves, Eaveb.

%%
%%
%%
%%

Part III

The simplest way to check the result.

Consider a light incident on the interface at incident angle of 76 degree.

1. Open my mph file.
2. Refine mesh.
3. Solve> Solver parameters> Harmonic propagation> Stationary
4. Solve
5. Postprocessing> Cross-Section Plot Parameters > Line/Extrusion > Apply

Then you will find that the amplitude of E-field in sobdomain 2 is around 205 (i.e. E_T=205). The incident plane wave has a amplitude of 377 (Because we set H-field is 1 in the boundary condition, $E_0=Z_0*H_0$, Z0 is the impedance of air and the value is around 377).

Using the formula , $E_T/E_0 = 2*n_1*cosi_1 / (n_1*cosi_1+ n_2*cosi_2)$ and set E_0=377, you will get E_T=162.

Why?

%%%
%%%
%%%
%%%