Case Study

Design and Simulation of a Parallel Plate Waveguide using Ansys HFSS

Mrinmoy Bharadwaj

Edited by Madhumita Saravana Kumar, Ansys Academic Development Team

education@ansys.com

Summary

This case study demonstrates basic steps involved in the design and simulation of a basic parallel plate waveguide and visualizations of the electric and magnetic fields inside the waveguide.

The Ansys Electronics Desktop (AEDT) is a platform that enables true electronics system design. AEDT provides access to the Ansys industry-standard electromagnetics simulation solutions such as Ansys HFSS, Ansys Maxwell, Ansys Q3D Extractor, Ansys SIwave, and Ansys Icepak using electrical CAD (ECAD) and mechanical CAD (MCAD) workflows. In addition, it also includes direct links to the complete Ansys portfolio of thermal, fluid, and mechanical solvers for comprehensive Multiphysics analysis. Tight integration among these solutions provides the user with unprecedented ease of use for setup and faster resolution of complex simulations for design and optimization.

Table of Contents

1. Introduction

A waveguide is used to transfer electromagnetic energy from one point to another. There are several different types of waveguides. The parallel plate waveguide (PPW) is the most basic type of waveguide. It has a very simple geometry formed by two parallel flat conducting metal plates with hollow space in the middle, as illustrated in Figure 1. PPW can support transverse magnetic (TM) and transverse electric (TE) modes and can also support a transverse electro-magnetic (TEM) mode due to its geometry. The understanding of the analysis of a parallel plate waveguide can be useful for modeling many other waveguides and also for modeling the propagation of higher order modes in a strip line.

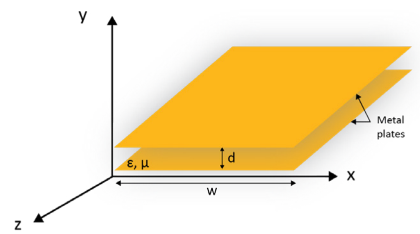

Figure 1: Basic geometry of a PPW

The width 'W' is assumed to be much greater than the separation 'd' so that the effect of fringing fields and any variations along x-direction can be ignored in the analysis. ε_r and μ_r are assumed to be the relative permittivity and relative permeability of the medium filling the region between the two metal plates.

2. PPW Model Design

For simulation, High Frequency Structure Simulator (HFSS) inside the Ansys Electronic Desktop framework shall be used. For building up the simulation model of a basic parallel plate waveguide in HFSS, we have used two copper sheets with some separation as shown in Figure 2.

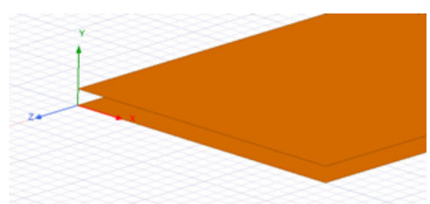

Figure 2: Simulation model of a PPW

The length, width and separation of these copper plates have been kept as variables, namely L, W and d respectively as shown in Figure 3. Making these dimensions variables would enable design iterations during simulations.

Name	Value	Unit
d	20	mm
W	300	mm
L	300	mm

Figure 3: Variables to denote PPW dimensions in Simulation model

The region between the plates has been assumed to be vacuum in the model. However, any dielectric material or combination of materials can be assumed. In fact, in this study results for a combination of dielectric materials will also be shown at a later stage.

For feeding electromagnetic energy into the model, a plane on one of the edge apertures of the structure has been used as shown in Figure 4.

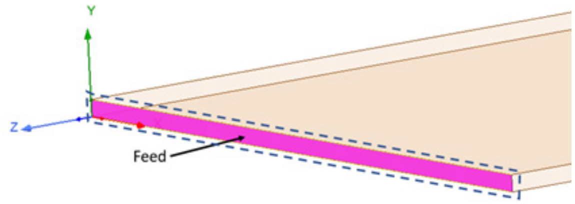

Figure 4: Feed to the PPW structure

The number of modes and mode polarities can be chosen to excite the PPW structure. In this study, a single mode shall be used for simulations.

3. HFSS Solution Process

The HFSS solver utilizes Finite Element Method (FEM) for solving geometry. FEM is a volumetric meshing based numerical solution technique. This method creates an initial mesh of the entire solution domain and then refines this initial mesh iteratively to improve the mesh quality. Once the metric for the mesh quality is met, the solver starts solving the frequency points, defined by the user, at each mesh node. In Figure 5, a view of the meshing applied onto the parallel plates of the PPW is shown for better understanding.

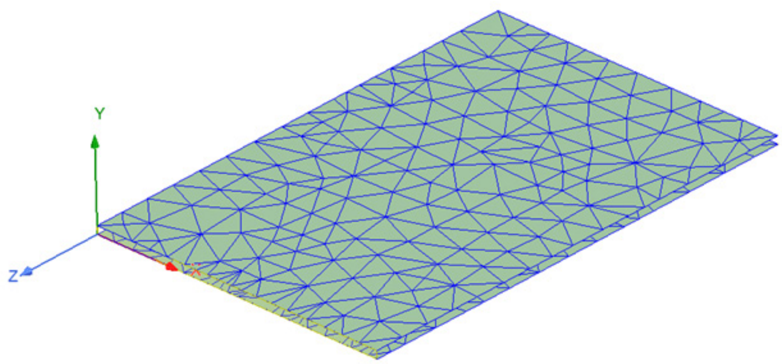

Figure 5: FEM applied to PPW

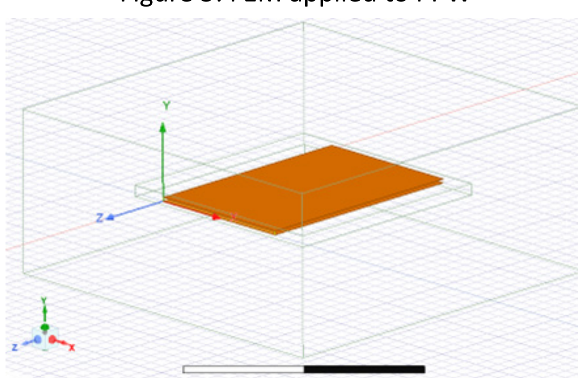

Figure 6: Simulation model

The entire simulation process consists of the following steps:

- **Geometry and Material:** In this first step the initial physical geometry and the materials to be used are defined. Figure 6 shows the initial geometry used in this case study.
- **Preprocessing:** This step involves defining the feed and the boundary conditions required for the simulations.
- **Solution:** In this step, a solution set-up and an associated frequency sweep is defined. The solution set-up involves specifying the meshing frequency and mesh quality metric.
- **Post-processing:** Once the model is solved, required outputs like s-parameters, electric and magnetic fields, surface currents *etc.* can be evaluated and visualized.
- **Validation:** The last and most important step is to validate the results and outputs and check if they are in line with what is expected.

It is important to note that the simulation process shown above might be an iterative process which may require several cycles of repetition before an optimal design can be found. At the end of each iteration, the engineer or designer can see if the changes in geometry and other aspects resulted in an improvement in the overall design or not. Based on this knowledge, further changes can be made to move the results in the required direction.

4. Simulation of a basic PPW with Ansys HFSS

In this section the electric and magnetic fields in the parallel plate waveguide shall be presented. From the basic analytical model of a PPW, we know that PPW can support TEM, TE, and TM modes due to its geometry. The simulation of a PPW with d=20mm, W=300mm and L=300 mm had been carried out at 1GHz. The electric and magnetic fields in top and cross-sectional views are shown in Figures 7 and 8.

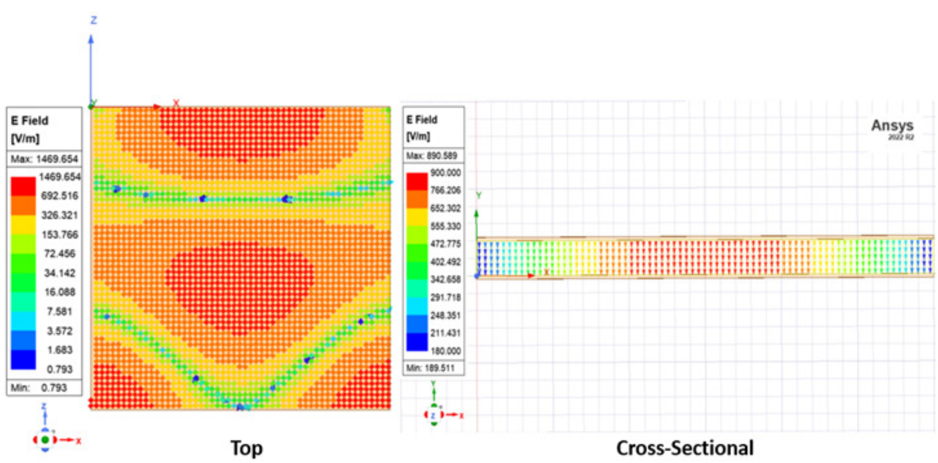

Figure 7: Top and cross-sectional plots of electric fields

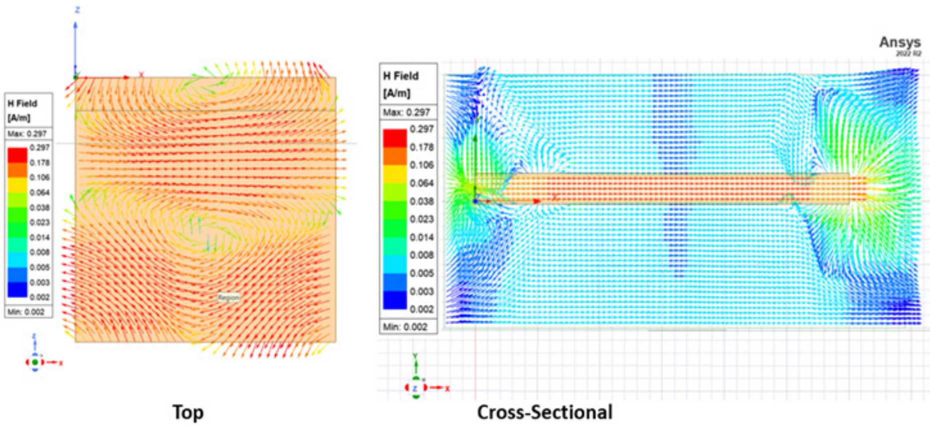

Figure. 8: Magnetic field plot in a PPW

If you are new to HFSS please enroll for this introductory self-learning course on Introduction to Ansys HFSS. To learn more about using Ansys HFSS for simulations, please visit the Ansys Innovation Courses website.

5. PPW with multiple fill materials

To satiate curiosity, simulation with partially filled dielectric materials have been performed for the parallel plate waveguide as well. Half of the empty space is filled with FR4 epoxy material (ε_r=4.4, μ_r =1). The geometry of the PPW is shown in Figure 9.

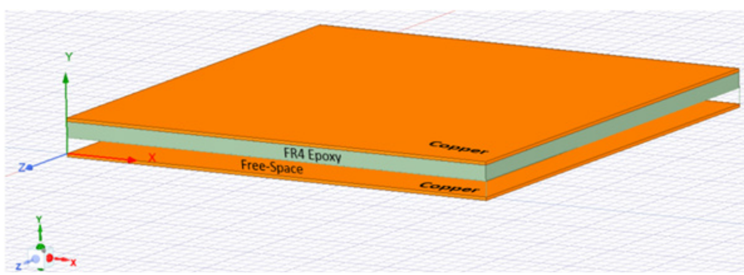

Figure 9: Geometry of partially filled PPW

The electric and magnetic field plots are shown in vs 10 and 11.

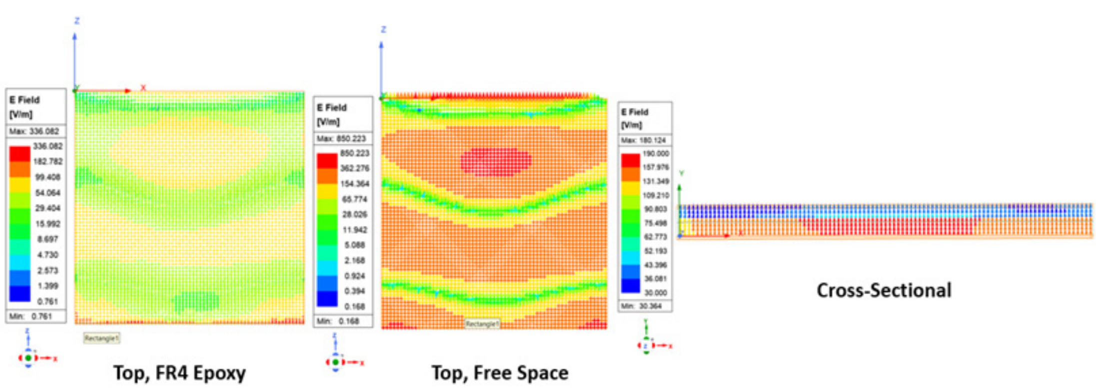

Figure. 10: Electric field plots for partially filled PPW

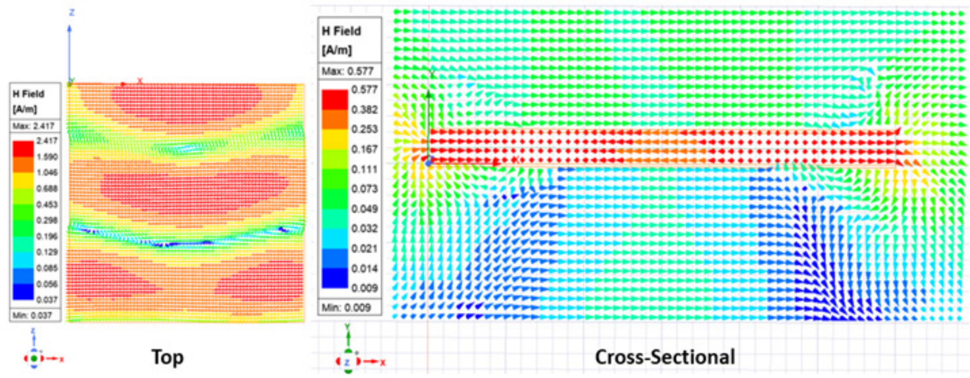

Figure. 11: Magnetic field plots for partially filled PPW

As evident in the field plots, the introduction of the partially filled dielectric material into the region between the metal plates causes significant variations in the electromagnetic field distributions inside the PPW. A good understanding of the behavior of the PPW will help in easier understanding of more complex waveguide structures such a rectangular, ridged, coaxial waveguides *etc.*

6. Further possibilities of investigations

The simulation-driven design and analysis process is an iterative one, often involving iterative modifications to the existing design to arrive at a more optimized design. In the case of the PPW, one can alter several variables in the design such as:

- Width and separation of the metal plates to see its impact on the electric and magnetic field distributions.
- Dielectric material and its properties, and the variations in the filling geometries to understand the impact caused on the fields as a result of the variations.
- Number of modes excited into the structure to see which mode is sustained and which is attenuated.
- Variations in the geometrical shape of the parallel plates to identify the impact on the field distributions inside the structure.

7. What does Ansys HFSS bring to the understanding?

In this case study, Ansys HFSS helps illustrate the design and simulation process of the most basic form of a waveguide, namely the parallel plate waveguide. The FEM based solver allows for a highly accurate solution of the electric and magnetic fields inside the PPW geometry, and also carry out several other simulations which would be rather impossible to describe with analytical models. The Ansys HFSS solution process can be used to explain how the numerical solution of the Maxwell's equations are carried out using FEM. The design process is iterative and can take up to several cycles to reach the required optimal design. From a more practical perspective, the fundamental benefits of a simulation-driven design process can be understood by illustrating how simulation helps speed up the design process and decrease cost as it lowers the need for expensive experiments and prototypes.

References

[1] Pozar, David M. (2012), "Microwave engineering," John Wiley & Sons, 2011. NJ :Wiley, Ch. 3, pg. 102

Use and Reproduction

Document Information

This case study is part of a set of teaching resources to help introduce students to designing and simulating high-frequency structures.

Ansys Education Resources

To access more undergraduate education resources, including lecture presentations with notes, exercises with worked solutions, microprojects, real life examples and more, visit www.ansys.com/education-resources.

Feedback

If you notice any errors in this resource or need to get in contact with the authors, please email us at education@ansys.com.

Acknowledgments

Robert O'Rourke and Vladimir Litunv

ANSYS, Inc.
Southepointe
2600 Ansys Drive
Canonsburg, PA 15317
U.S.A.
724.746.3304
ansysinfo@ansys.com