

Semester S1

-

Foundations of electromagnetic wave propagation

Practical Work PW3

-

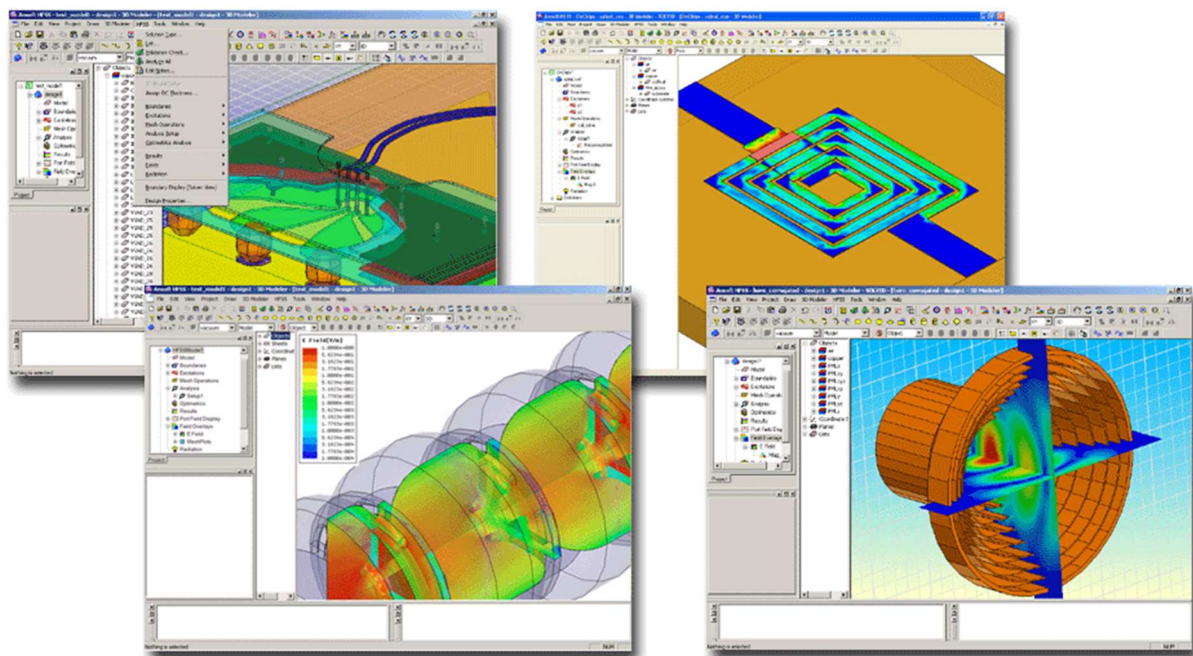
Electromagnetic analysis of planar microwave circuits: introduction to “HFSS”

I INTRODUCTION

I.1 CONTEXT

As seen in momentum PWs, simulation is an essential step since it allows predicting the behavior of the electromagnetic and electrical parameters of the devices used. For many applications, the approximate analytics methods do not allow to obtain, with sufficient precision, these parameters.

We therefore propose to use the 3D High Frequency Structure Simulator (HFSS), based on the two-dimensional and three-dimensional finite element method in the frequency domain. This powerful tool is widely used for the characterization and computer-aided design of complex planar or volume components, circuits and microwave modules (interconnection, antennas, RF and microwave components, packaging, etc.). Compared to Momentum, HFSS manages structures and 3D phenomena and more "fair" (multi-) planar. For example, a waveguide cannot be simulated under Momentum but is perfectly simulated under HFSS. This is notably due to the use of FEM (finite element method) but other methods for 3D structures also exist (FDTD, ...).



– Figure 1 –

I.2 SHORT PRESENTATION OF THE METHOD OF FINISHED ELEMENTS

The studied structure can consist of several environments Ω_i of any size and geometry, homogeneous, linear, isotropic, with or without loss, characterized by their permittivity ϵ_i , their permeability μ_i and their conductivity σ_i . **It must be necessarily limited by a closed surface** composed of electric or magnetic walls or conditions (impedance ...) simulating infinity.

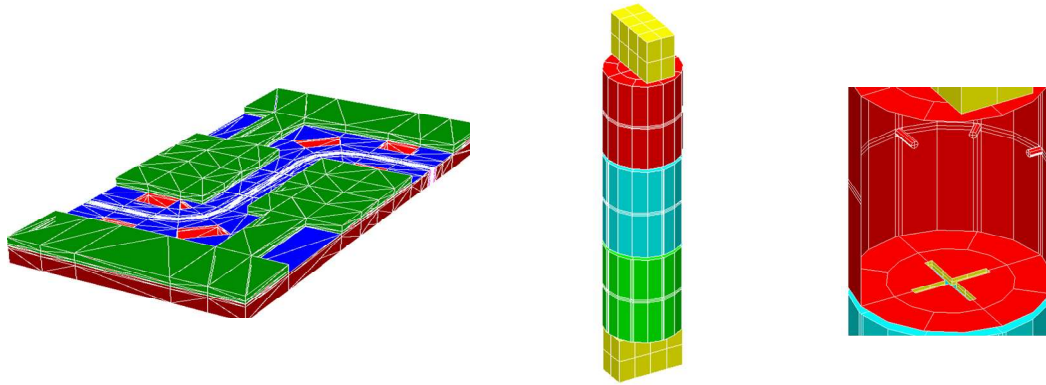
Note: **in HFSS, a wall / face in contact with the outside is equivalent to a face with a perfect conductor condition.** This is HFSS specific and may vary from one simulator to another.

Solving the problem requires writing that the electromagnetic field in the structure satisfies Maxwell's equations. This method can therefore be applied to the characterization and calculation of the electromagnetic field distribution of all passive structures in three dimensions:

- armored structures in free oscillations (called **Eigenmode** in HFSS): calculation of resonance frequencies and associated electromagnetic fields.
- shielded structures or not (ie with radiation), "maintained" by transmission lines, waveguides, microstrip lines ... (called **Driven Modal**): determination of impedances and constants of propagation of access, calculation of generalized S parameters and associated electromagnetic fields (radiated field, far field, ...)
- structures with localized access (called **Driven Terminal** or **Driven Modal**).
- ...

In forced or "maintained" oscillations, the structure has at least one access and we look for the linear relationship existing between the outgoing and incoming waves, i.e. the distribution matrix [S]. We then obtain an equation that will be solved by the numerical finite element method.

This resolution first requires that the structure studied be broken down into simple elements (3D tetrahedra). This is called the meshing of the structure (Figure 1). HFSS uses an adaptive mesher, which means that the mesh is iteratively refined by the software.



– **Figure 2** – Mesh of several volume structures

The next step consists, on each of the elements, to approximate the unknown vector functions \vec{E} and \vec{H} using polynomials. The approximate function is then introduced into the general equation. This digital resolution leads to matrix systems that will differ according to the dimensions and the type of problem.

I.3 THE DIFFERENT STEPS OF A PROJECT

This paragraph describe the different basic steps to draw, define, solve and analyze a problem.

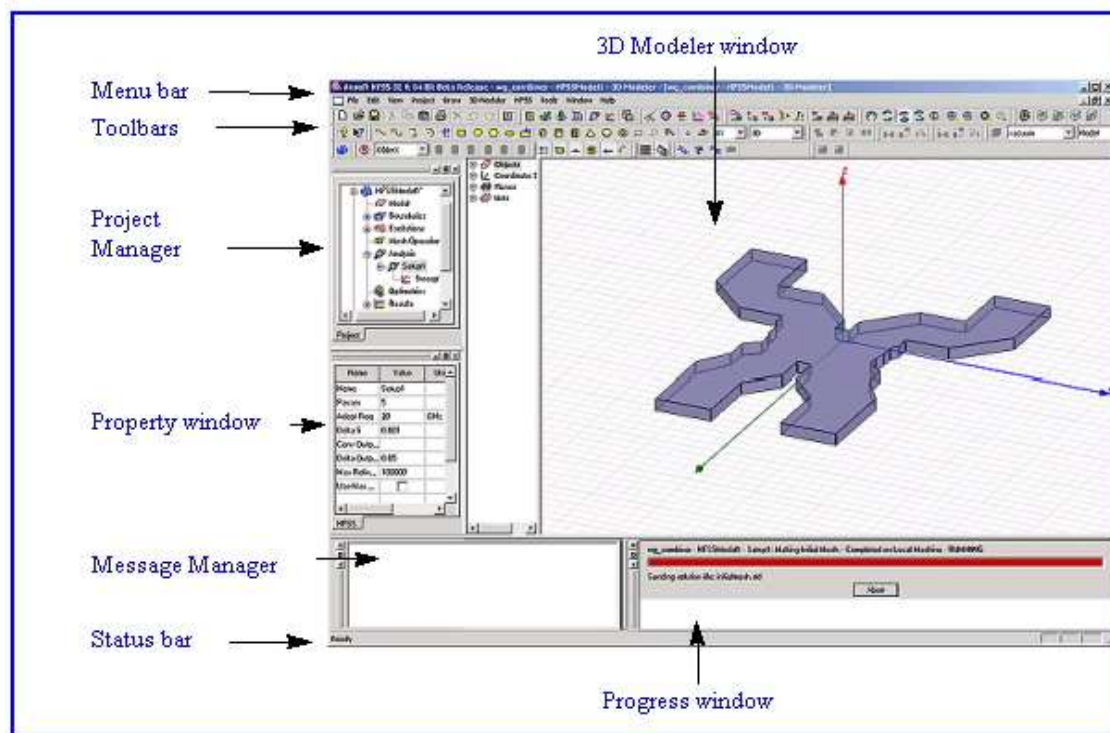
1. Creating or opening a project. A project can contain multiple designs (= structures).
2. Inserting an "HFSS Design" model into a project.
3. Selecting a solution type:
 - Driven Modal
 - Driven Terminal
 - Eigenmode
4. Definition of units (mm, μm , ...).
5. Drawing the geometry of the structure and assigning a material to **each object**.
6. Attribution of the boundary conditions (Perfect E, Perfect H, plane of symmetry, radiation condition, ...) on the surfaces of the structure requiring it.
7. For "Driven" type solutions, definition of accesses (excitation sources and loads).
8. Definition of the parameters of the simulation (working frequency, precision on the solution, number of iterations to be carried out, ...) then launching the computation.
9. Post-processing: visualization and analysis of simulation results.

1.4 HFSS GUI

The **HFSS** GUI contains several windows (cf. - Figure 3 -):

- **3D Modeler window**: the objects will be drawn in this window
- **Project Manager** describes in detail the different open projects. Each modeling contains the geometry of the structure (**Model**), the boundary conditions (**Boundaries**), the excitation systems (**Excitations**), the optional mesh operations (**Mesh Operations**), the calculations to be carried out (**Analysis**), the results (fields, S-parameter, ...), ...
- **Property window** contains the information of the object selected in the **3D Modeler window** or the information concerning a step of the modeling selected in **Project Manager** (excitations, analyzes, results ...).
- **Message Manager** is a control window that gives information on the progress of the description of modeling and analysis (error messages, ...)
- **Progress Window** allows you to monitor the progress of the simulation run.

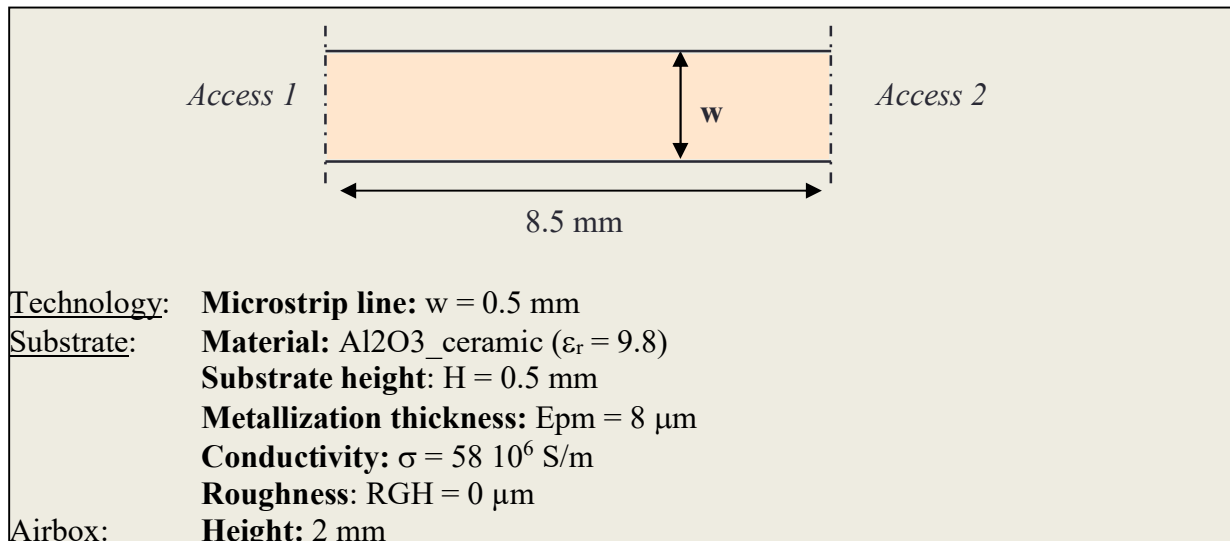
The interface also contains a menu bar, toolbars (shortcuts for the various commands contained in the menu bar) and a **Status bar** (information on the command being executed).





– Figure 3 – Ansoft/Ansys HFSS GUI

II MICROSTRIP LINE STUDY

In order to point out the differences between HFSS and Momentum, the first structure will be the same as in the first practical work: a 50 Ω line.



– Figure 4 –

- Log on to the PC (under Windows) using your account
- Start HFSS by double clicking on the icon :
- A new project called "Projectn" is automatically created when the software is launched.
- Click on the Insert HFSS design icon : a model named "HFSSDesign1" is created in the project.
- Save this new project by clicking File then Save As. Then type "PW3_hfss_lines" in the file name tab and save. The project is then saved in the file "PW3_hfss_lines.hfss" in the directory "Ansoft".

II.1 SETTING UP THE PROJECT

II.1.1 Solution type

The structure is analyzed in forced oscillations. To specify the type of solution:

- Click **HFSS** and then **Solution Type**. In the **Solution Type** window select **Modal** and **Network Analysis**.

II.1.2 Units


To check the units for drawing the geometry of the structure to study:

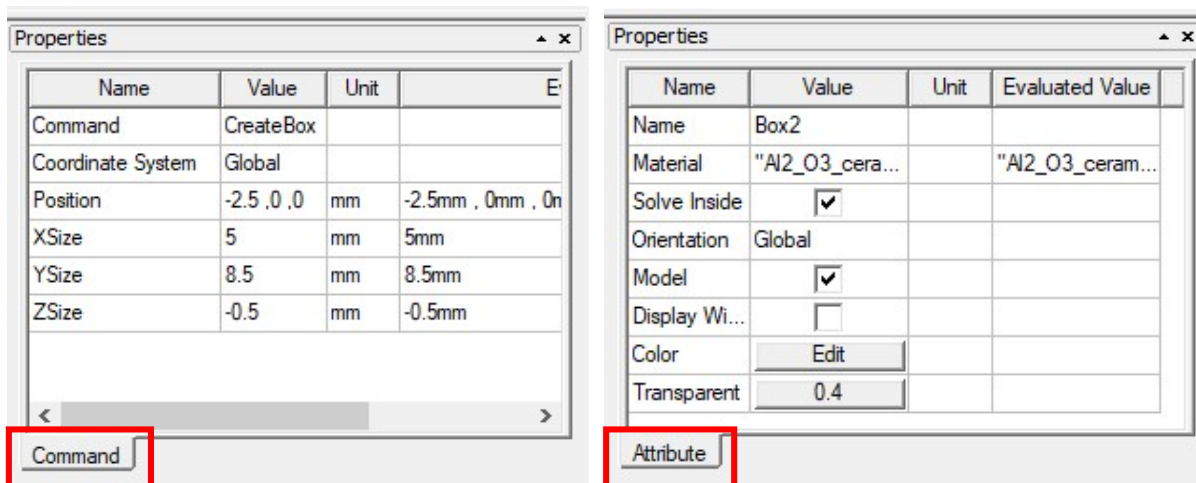
- Click **Modeler** and then **Units**. Select **mm** in the **Set Model Units** window and click **OK**.

II.1.3 Structure drawing

You have to draw 3 boxes: the first for the air box, the second for the line and the third one for the substrate below the air box.

For the air box :

- Click **DRAW** then **Box** (or click on  in the toolbar)
- Enter the position of a corner of the box in the **Status bar** field:
 $X = -2.5$ $Y = 0$ $Z = 0$ then press *Enter*
- Specify the dimensions following x, following y and following z of the box
 $dX = 5$ $dY = 8.5$ $dZ = 2$ then press *Enter*
- In the **Properties** window select the **Command** tag. It is then possible to control and modify the dimensions and the position of the 3D box.
- By selecting the **Attribute** label, it is then possible to give a name to the box (for example: guide), to confirm or modify the material inside the 3D box (here: **air** or **vacuum**), to modify the transparency of the box. Increasing the transparency of the box makes it easier to distinguish the interfaces between different objects if necessary.



– Figure 5 –

- Click on  to visualize the full structure.

Now, do the same thing for the signal line (0.5 mm x 8.5 mm x 0.008 mm) and for the substrate (5 mm x 8.5 mm x 0.5 mm) under the air box.

For the line, you need to select the material named **copper**

- In the modeling window, click on the element
- In the **Properties** window, **Attribute** label, in front of **Material**, click on "vacuum"
- Choose **Edit ...**, a new window appears.
- Choose **copper** in the listed materials.

For the substrate, you will need to change the material associated to it and to create a new one.

- Go in the **Materials** window following the same procedure as above.
- Click on **Add Material**

Name: "alumina"

Relative Permittivity: 9.8

Dielectric Loss Tangent: 0.0001

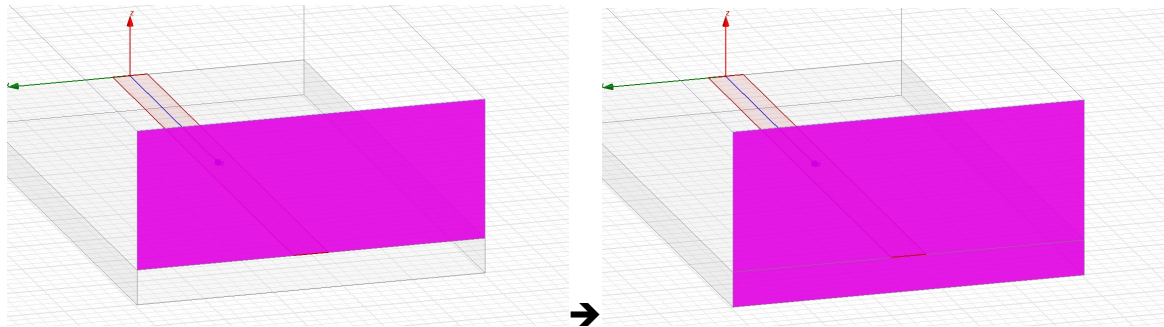
Then click on **OK**

II.1.4 Accesses and boundaries

Accesses


The structure we have just built has two accesses, which we will now define.

- Right click in the **3D Modeler** section and click on **Select Faces**.
- Select the faces to be associated with Port 1 by clicking on them (hold CTRL down between each click).



– Figure 6 –

- Click on **HFSS, Excitations, Assign, Wave Port**.
Wave ports are used in the case of transmission lines and waveguides.
- The **Wave Port** window appears: Type **Port1** in the **Name** field and click **Next**.
The mode number is set to 1.

Repeat the same operation for Port 2 on the other side. Use the icon  to rotate the structure and have access to the corresponding side of port 2. Press Esc to exit the selected mode.

Accesses visualization

- Click on + in the **Excitations** subdirectory in the **Project Manager** window and on **Port1** to view access 1 and **Port2** to view access 2.

Boundaries conditions

By default, all other external surfaces that limit the structure are considered as electrical short circuits (walls with infinite conductivity). So we have to change the lateral faces and the top one to air boundaries.

- Select faces as you did for accesses.
- Click on **HFSS, Boundaries, Assign, Radiation**.
- Click OK.

The drawing is now done. Before moving to the next part, save your project.

II.2 ANALYSIS AND EXPLOITATION OF THE RESULTS

The objective here is to define the parameters of the different simulations envisaged.

The software calculates the parameters S of the structure at a given frequency or on a frequency band, on the ports only or in the whole volume.

II.2.1 Determination of the electromagnetic field and S parameters.

In order to run any analysis we need to define some simulation setup:

- Click **HFSS, Analysis Setup, Add Solution Setup**
- Type "msline" in the **Setup Name** field
- Type 16 GHz in the **Solution Frequency** field
- Set the maximum number of iteration is 6
- Set the maximum difference between two parameter values S to 0.01 (precision)
- Then click on **OK**

To add a frequency sweep:

- In the **Project manager** window, right click on the setup called **msline**
- Click Add **frequency sweep**
- Choose:

Sweep type: interpolating

Start: 1 GHz

Stop: 16 GHz

Step size: 0.1 GHz



The accuracy of the solution depends on the mesh. At first, the software designs a coarse mesh and performs a calculation, then refines the mesh and a second calculation is done. The difference of these two results gives the precision of the solution. If this accuracy is less than or equal to that set by the user in the simulation parameters, the process stops.

The software then considers that the last matrix $[S]$ obtained is the solution of the problem. If this is not the case, a new mesh is done and the previously described approach is repeated.

The other criterion to stop the resolution is the number of "passes" always set by the user in the parameters of the simulation (one pass is associated with a mesh).

When one of the two previous criteria is satisfied, the simulation ends.

Before running the previously defined simulations make sure that all the necessary steps have been validated:

- Click **HFSS, Validation Check** (or click the button .
- To run the simulations, click **HFSS, Analyze All** (or click the button .

II.2.2 Check fields at the accesses

The first step after simulation is to check if the excitations are in conform to what we expect:

- Go to port field display and for show for each port the fields for the current excitation mode
 - Comment on the fields you observe.

II.2.3 Getting the parameters of the line

Before plotting the response, look to the line parameters that calculated by HFSS:

- Click **HFSS, Results, Solution Data**. A new window should appear.
- Tick Zo
 - Comment the value.
- Tick Gamma and choose Real/imaginary in the *combobox*
 - What the 2 fields of gamma stand for.

II.2.4 Plotting the response

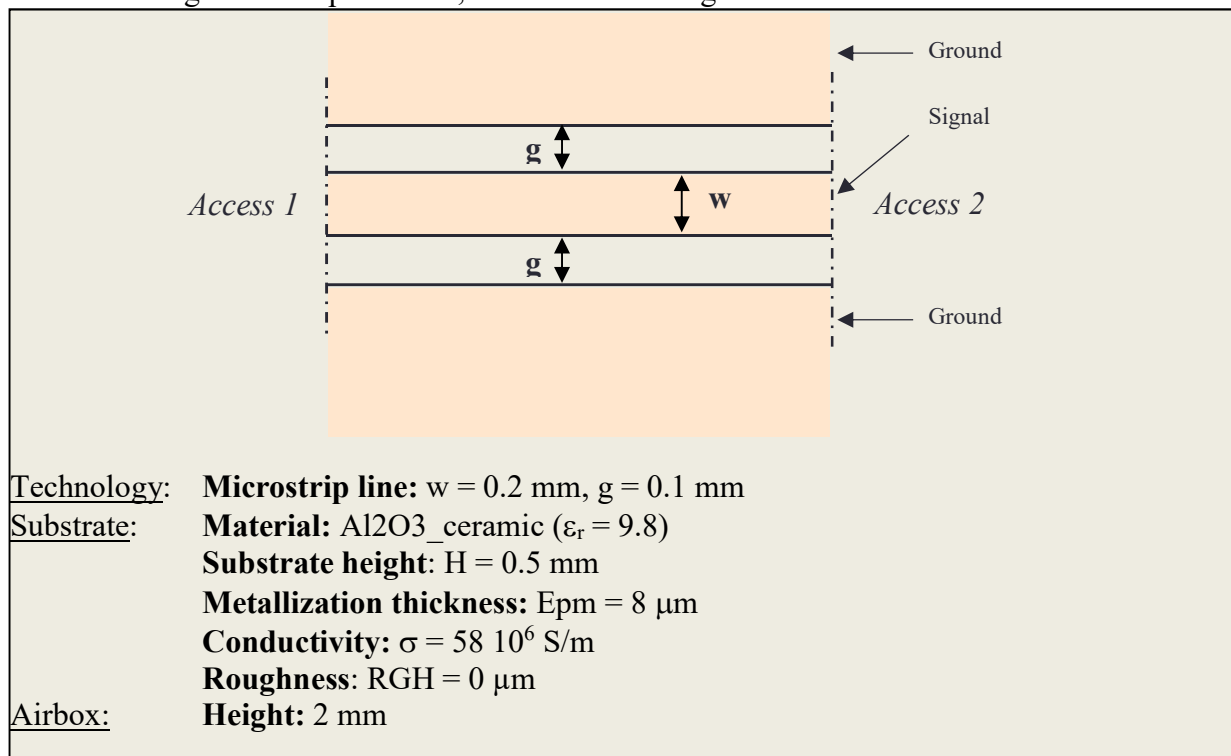
In order to see the S-parameters response:

- Click **HFSS, Results, Create Modal Solution Data Report, Rectangular Plot**.
- Choose **S Parameter, S(1,1), dB** and click **New Report**.
- Choose **S Parameter, S(2,1), dB** and click **Add Trace**.
 - Comment the curves obtained. Do they match the momentum ones ?

III COPLANAR LINE

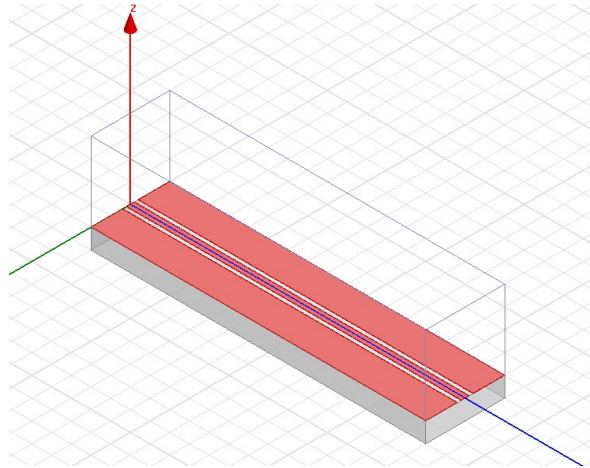
III.1 DRAWING A COPLANAR LINE

Using the same procedure, draw the following line:



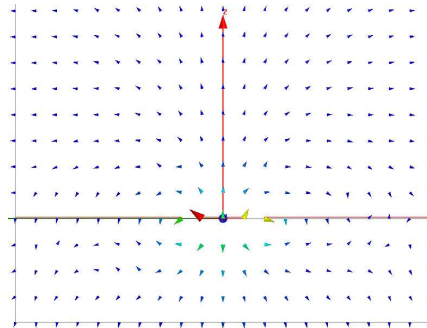
– Figure 7 –

- Do a copy of the previous design and name it **coplanar** (right click on design, **copy**, right click on project, **paste**).
- Reduce the substrate and air box widths from 5mm to 2 mm.
- Do the drawing and all the steps, including simulation.
- Check the fields at the accesses, check the line parameters and plot the responses.
 - What are your comments on these three items?



– Figure 8 –

These are the fields you should obtain by clicking on **Port Field Display**:



– Figure 9 –

III.2 USING VARIABLES IN HFSS

Variables in HFSS are a powerful way to parametrize your structure and to modify it in a simple and fast way. We will now use them to control the size of the coplanar line.

- Do a copy of the previous design and name it **coplanar_var**.
- Edit the properties of the central line (double click on its command **CreateBox**).
- For **xSize**, write “w”.
- A new window should open. Type 0.2 for the **Value** field and click **OK**. You have just created a variable.
- Now, in **Position**, write $-w/2, 0, 0$
- Go back to your design and click on the design. Variable w is now available.
 - What happen when you change the value?
- Do the full parametrization of the structure along **x-axis**: add a gap g of 0.1 mm and a width w_{sub} of 2 mm.
- Now, set the width of the substrate to 5 mm and simulate the structure.
 - Any comment on the excitation mode?