Semester S1

-

Foundations of electromagnetic wave propagation

Practical Work PW4

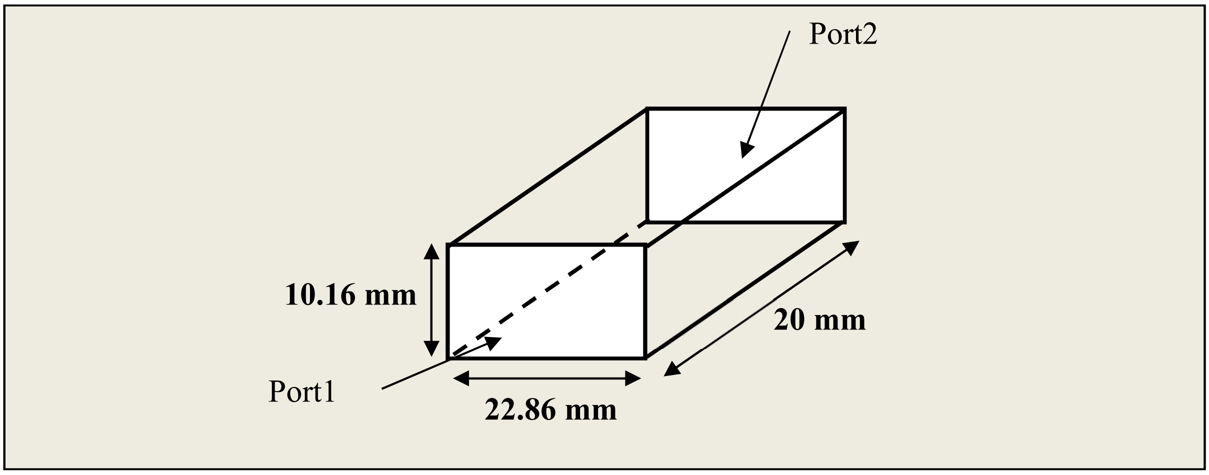
-

Electromagnetic analysis of 3D microwave circuits: waveguide analysis

Aurélien Périgaud, Olivier Tantot, Claire Dalmay

I RECTANGULAR WAVEGUIDE IN TRANSMISSION

We will study in this paragraph a rectangular guide with two accesses (Port 1 and Port 2) (- Figure 1 -).



– **Figure** 1 – Rectangular waveguide

- Launch HFSS and create a new project, call it « PW4_hfss_waveguide », and a new design called « WG2ports ».

I.1 SETTING UP THE PROJECT

I.1.1 Drawing

- Do all the steps of the previous PW in order to have the structure above :
  Size X = 22.86 mm
  Size Y = 10.16 mm
  Size Z = 20.00 mm

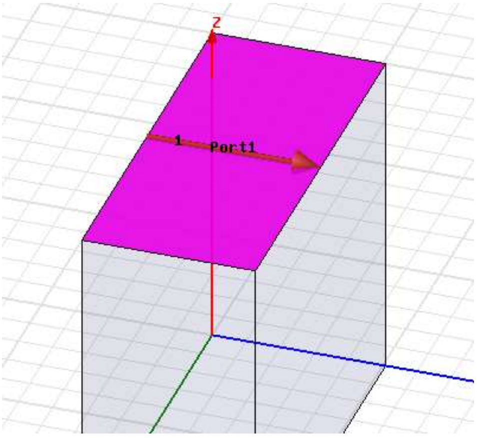
I.1.2 Accesses

The structure we have just built has two accesses, which we will now define. The faces parallel to the xOy plane in z = 20 mm and z = 0 mm will therefore be respectively Port1 and Port2 of the structure.

- Right click in the **3D Modeler** section and click on **Select Faces**.
- Select the face to be associated with Port1 by clicking on it.
- Click on **HFSS**, **Excitations**, **Assign**, **Wave Port**.
- The **Wave Port** window appears. Type **Port1** in the **Name** field and click **Next**.
- By default, the mode number is set to 1. Select **New Line** ... from the **Integration Line** drop-down list.

An **Integration Line** is a vector indicating the direction of the electric field in the access plane.

- Enter the position of the starting point of the vector
  X = 11.43 Y = 0 Z = 20 then *Enter*
- Enter the length of the vector
  dX = 0 dY = 10.16 dZ = 0 then *Enter*
- The **Wave Port** window appears again: if port1 is correctly set click **Next** then **Finish**.



– **Figure** 2 – Visualization of Port1

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

To visualize the accesses, click on the + sign of the **Excitation** section in the **Project Manager** window and then click on Port1 to see the first access and on Port2 to see the second one.

I.1.3 Boundary conditions

By default, all other external surfaces that limit the structure are considered as metallic walls with infinite conductivity. Since we do not want to imply any losses, there is no need to specify particular surface conditions.

- Save the project.

I.2 RUNNING ANALYSIS AND DISCUSSING THE RESULTS

Three different setups will be defined: two for analysis of the accesses themselves (at different frequencies) and one for the simulation of the whole structure.

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

The software allows a 2D analysis in the accesses of the structure:

- Click **HFSS**, **Analysis Setup**, **Add Solution Setup**
- Type "2D_1" in the **Setup Name** field.
- Type 11.5 GHz in the **Solution Frequency** field.
- Select **Solve Ports Only**
- Then **OK**

Create a new **Setup**, with identical parameters except for the **Solution Frequency** that is set to 6 GHz. Name this setup "2D_2".

To add a 3D resolution:

- Click **HFSS**, **Analysis Setup**, **Add Solution Setup**
- Type **3D** in the **Setup Name** field
- Type 11.5 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 **OK**

Then, add a frequency sweep:

- In the **Project Manager** window, right click on the setup called **3D**
- Click **Add frequency sweep**
- Choose:

  Sweep type: **interpolating**

  Start: 6 GHz

  Stop: 13 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 made. 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 made 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 reached, the resolution 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 start simulations, click **HFSS**, **Analyze All** (or click the button).

I.2.2 Analysis 2D_1

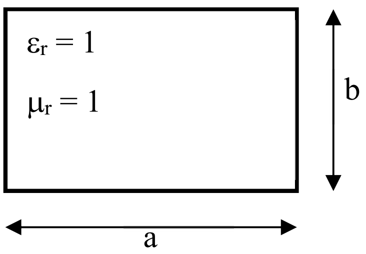
In the **Project Manager** window appears a subdirectory **Port Field Display** which allows to visualize the electric field in the accesses of the considered mode.

- Click on the + in the sub-directory **Port Field Display**
- Click on the + of the sub-directory Port1 then on **Mode1**
- Click on the + of the sub-directory Port2 then on **Mode1**
  - ➢ Comment on the visualizations at both ports and identify the excited mode in the rectangular waveguide.

By default, HFSS provide a lot of information about ports:

- Click on **HFSS**, **Results**, **Solution Data**
- Select the simulation "2D_1"
- Select the **Matrix Data** tab and then **Gamma** (the propagation constant)
- Display the real part and the imaginary part of the propagation constant.
- Note the values of the phase constant and the attenuation constant.
  - ➢ Compare these values to those you will theoretically determine with the following reminders:

Reminder:



$$\beta=\sqrt{\mu_r \varepsilon_r k_0^2 - k_c^2} \qquad \text{with } k_0=\frac{\omega}{C}$$

And for TE_{nm} mode, $k_c=\sqrt{\left(\frac{n\pi}{a}\right)^2+\left(\frac{m\pi}{b}\right)^2}$

- ➢ What **Lambda** stands for ?

- Recall the relationship between Beta and this **Lambda** when Alpha << Beta. Do you get the same value of Lambda as the one displayed by the software.

I.2.3 Analysis 2D_2

- Select the simulation «2D_2»
- Display the gamma values for the "2D_2" solution.
  - Comment on these values.

- Calculate the cut-off frequency of the TE_{nm} excited in the rectangular waveguide.

Reminder:

$$f_{cTE_{nm}} = \frac{C}{2\pi\sqrt{\varepsilon_r \mu_r}}\sqrt{\left(\frac{n\pi}{a}\right)^2 + \left(\frac{m\pi}{b}\right)^2}$$

  - Conclude.

I.2.4 Analysis 3D

I.2.4.1 Visualization of parameter values S in module and in phase in the Solution Data window at 11.5 GHz:

- Select the "3D" simulation
- Select the **Convergence** tab
  - Note the accuracy of the calculation, ie **Max Mag. Delta S**. Conclusion.

- Select the **Matrix Data** tab
- View **Gamma**, **Lambda**
  - Take the values and compare them to those obtained during the 2D calculation. Conclusions.

- View **S Matrix**
- Display the module and phase of the S parameters
- Go to **lastAdaptive** then read the S parameters in module and phase.
  - Comment on these values.

I.2.4.2 Visualization of parameter values S in module and in phase as a function of frequency:

- In the **Project Manager** window, right-click Results.
- Click **Create Modal Solution Data** and then **Rectangular Plot**
- Select "S (1,1)" and press **New report**
- Select "S (2,1)" then press **Add Trace**
    - ➢ Comment the results.

I.2.4.3 Mesh visualization:

In the **Modeling** window select the 3D object then:

- Click on **HFSS**, **Fields**, **Plot Mesh**
- Select **3D**: **LastAdaptive** from the drop-down list to the right of **Solution** and click **Done**
- To remove the visualization of the mesh, click **HFSS**, **Fields**, **Delete Plot** ... then select the mesh and do **OK**.

I.2.4.4 Visualization of the electric field throughout the structure

Select the 3D object. In the **Property** window, click on the tab corresponding to **Transparent** and place the cursor so that it has a value equal to 1. The structure is then completely transparent.

- Click **HFSS**, **Fields**, **Fields**, **E**, **Mag_E**
- Select from the drop-down list to the right of **Solution 3D**: **LastAdaptive**
- Select **Mag_E** in **Quantity** and **All** in **In Volume** then click **Done**
- Modify the visualization by clicking **HFSS**, **Fields**, **Modify Plot Attributes** ... then **OK** in the **Select Plot Folder** window
- Click the **Plots** tab and select **IsoValSurface** in **Scalar plot**
- Click the **Color map** tab, then in the drop-down list to the right of **Spectrum** choose **Rainbow** then click on **Close**

I.2.4.5 Visualization of the electric field throughout the structure as a function of time

- Click on **HFSS, Fields, Animate** and **OK**
- Click **Close** to exit animation mode.

I.2.4.6 Visualization of the electric field on one face of the structure

- Choose **Face** from the drop-down list to the right of the icon

- Select a side parallel to the xOz plane
- Click on **HFSS, Fields, Fields, E, Mag_E** then **Done**
- View the electric field module as a function of time.

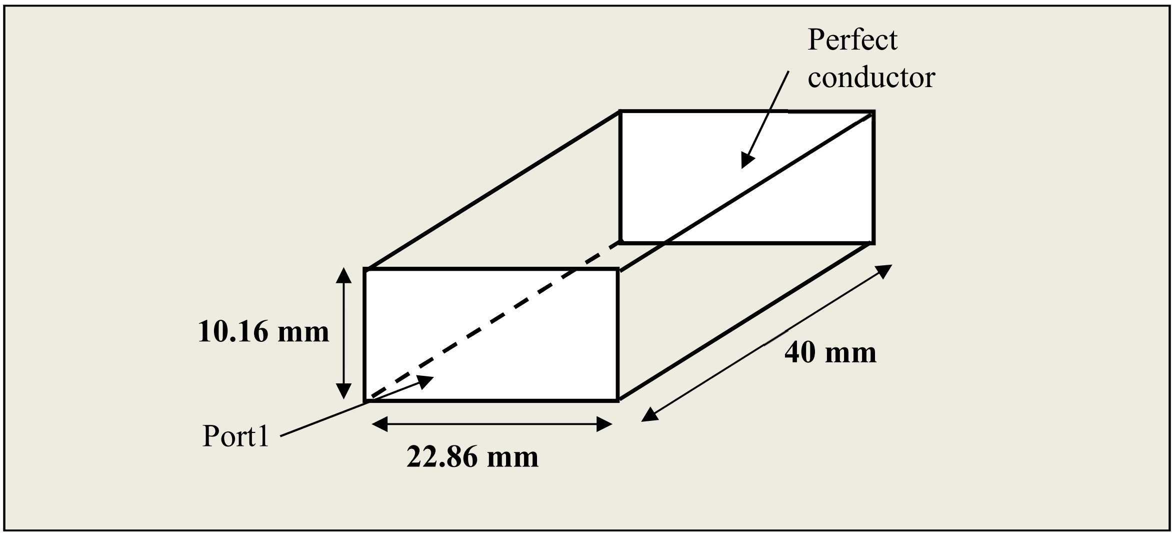
I.2.4.7 Visualization of the electric field vector on one face of the structure as a function of time

- Select a side parallel to the xOz plane
- Click on **HFSS, Fields, Fields, E, Vector_E** then **Done**
- The visualization of the arrows (dimensions, spacing, ...) can be modified by clicking on **HFSS, Fields, Modify Plot Attributes** ...
- Visualize the vector of the electric field as a function of time

II RECTANGULAR WAVEGUIDE WITH ONE SHORT-CIRCUITED END

Copy the previous design in a new one, called "WG1port".

We will now study the following structure:



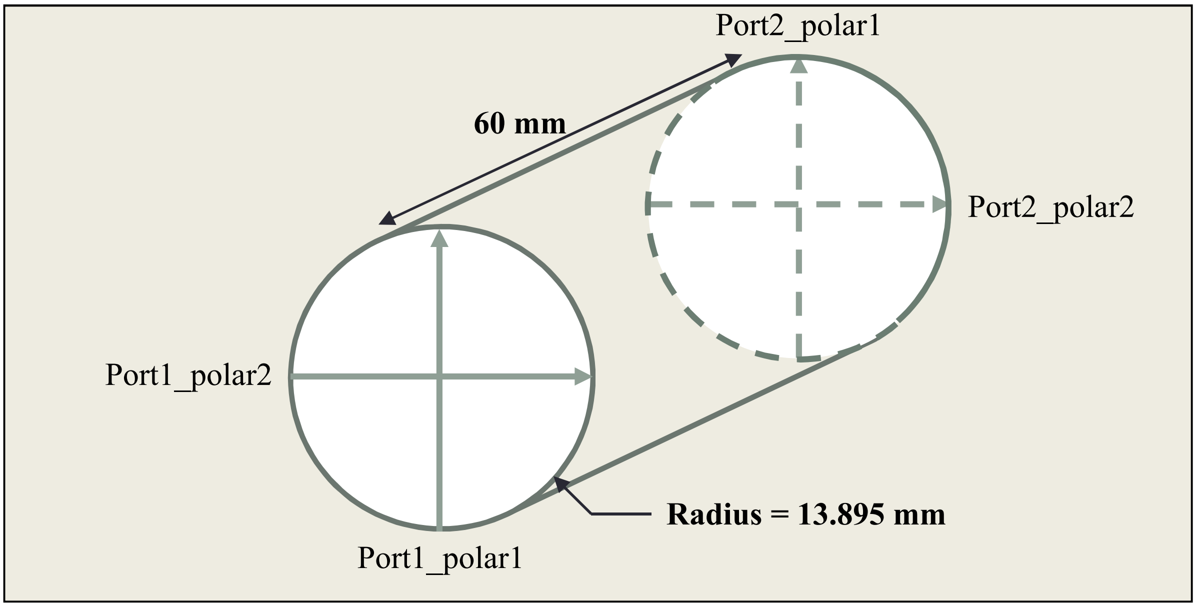
– **Figure** 3 – Rectangular waveguide with 1 access

- Perform the electromagnetic analysis at the f = 11.5 GHz frequency of this structure to access using the HFSS software.
- View the propagation constant, the parameter values S in module and in phase, the modulus of the electric field as a function of time on a face parallel to the plane xOz.
    - Conclusions

III CYLINDRICAL WAVEGUIDE IN TRANSMISSION

Create a new design and call it "Circ2ports".

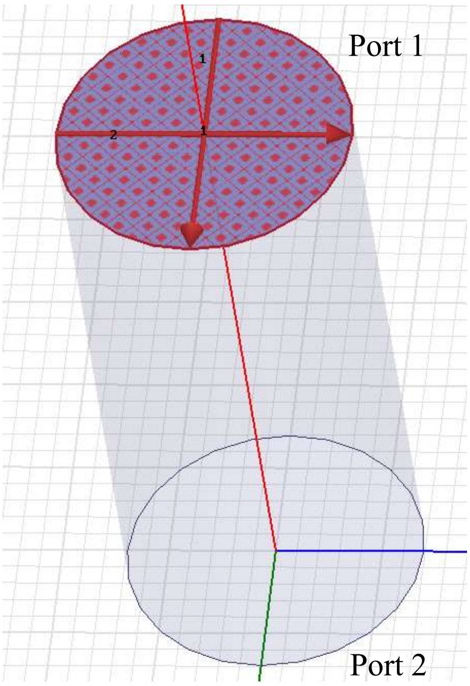
The case studied here is about the excitation of a circular waveguide:



– **Figure** 4 – Circular waveguide with 2 accesses

- Create a cylinder with the following dimensions:
  Radius = 13.895 mm Height = 60 mm Axis : Z
- Create waveports for the circular faces as before, but, in the **Modes** tab, set the **Number of Modes** to 2. The different polarizations in figure 4 correspond to the modes that HFSS will excite (so, 2 modes – or polarizations – per access port).
- Define then 2 perpendicular integration lines:
  X = -13.895 mm Y = 0 mm Z = 0 mm (or 60 mm)
  dX = 27.79 mm dY = 0 mm dZ = 0 mm
  And
  X = 0 mm Y = -13.895 mm Z = 0 mm (or 60 mm)
  dX = 0 mm dY = 27.79 mm dZ = 0 mm
- Tick **Align modes using integration lines**. This option enforce the mode to follow the integration line which is exactly what we want (this option should always be used for circular waveguide were 2 polarization can exist).

When clicking on **Excitations**, the port should look like this:



– **Figure** 5 –

- Create **Setup** and **Frequency Sweep :**
  The setup settings are the followings:

  Solution frequency: 10 GHz

  Maximum Number of Passes: 10

  Maximum Delta S: 0.01

  The frequency sweep settings are:

  Sweep type: Fast

  Start: 6 GHz

  Stop: 12 GHz

  Step Size: 0.1 GHz

- Run the analysis and plot magnitudes (in dB) for port1 and 2 according to the table besides:

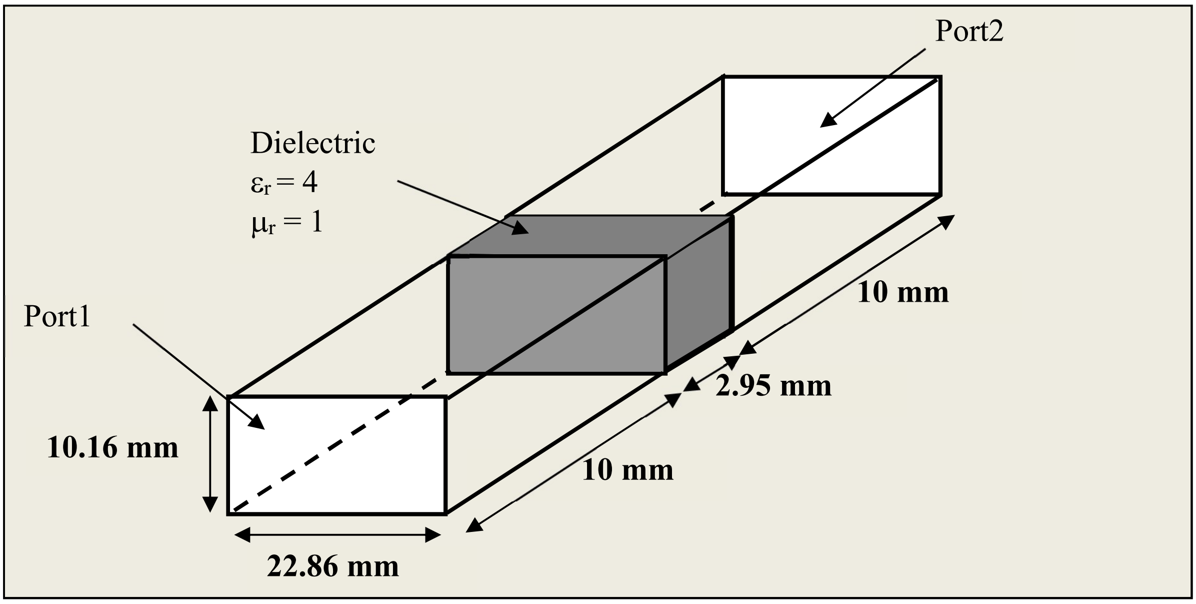
	Port1_polar1	Port1_polar2	Port2_polar1	Port2_polar2
$S1_{1}1_{2}$	x	**x**		
$S1_{1}2_{1}$	x		**x**	
$S1_{1}2_{2}$	x			**x**
$S1_{1}1_{1}$	x x			

➢ Comment those results.

IV RECTANGULAR WAVEGUIDE, PARTIALLY LOADED

Create a new design and call it "WGloaded".

We propose to study in this section, a section of rectangular guide loaded by a dielectric sample:



– **Figure** 6 – Loaded waveguide

IV.1.1 Drawing the structure

Perform the electromagnetic analysis of this structure using the HFSS software.

Calculate the parameters S in module and in phase between 8 and 12 GHz and visualize the modulus of the electric field as a function of time on a face parallel to the plane xOz.

To define the geometry, it is necessary to create three different juxtaposed boxes respectively containing air, dielectric, air.

It is therefore necessary to add in the list of materials the dielectric of relative permittivity 4 and relative permeability 1: proceed as in the previous PW.

Define **a new setup at 12 GHz** and a frequency sweep with a range from **8** to **12 GHz**, with a step of **0.1 GHz**, and select **Interpolating** method

Visualize the values of the parameters S in module according to the frequency by creating a new plot with S21 and S11.

- Comment on the results obtained. Conclusions.

Visualize the electric field vector on one face of the structure as a function of time for f = 12 GHz:

- Select the three faces parallel to the xOz plane, press *Ctrl* and click on the different surfaces.
    - ➢ Interpret the results. Conclusions.

IV.1.2 Material characterization

The previous structure simulates a material characterization experiment used at the lab. In this experiment, the material sample to characterize is place between two waveguides. The S-parameters are then analyzed in order to deduce the permittivity and permeability of the material.

We propose to do the same thing in the last part of the PW and check if we obtain the correct results.

First, we will de-embed the ports in order to set the reference planes to the planes of the dielectric object:

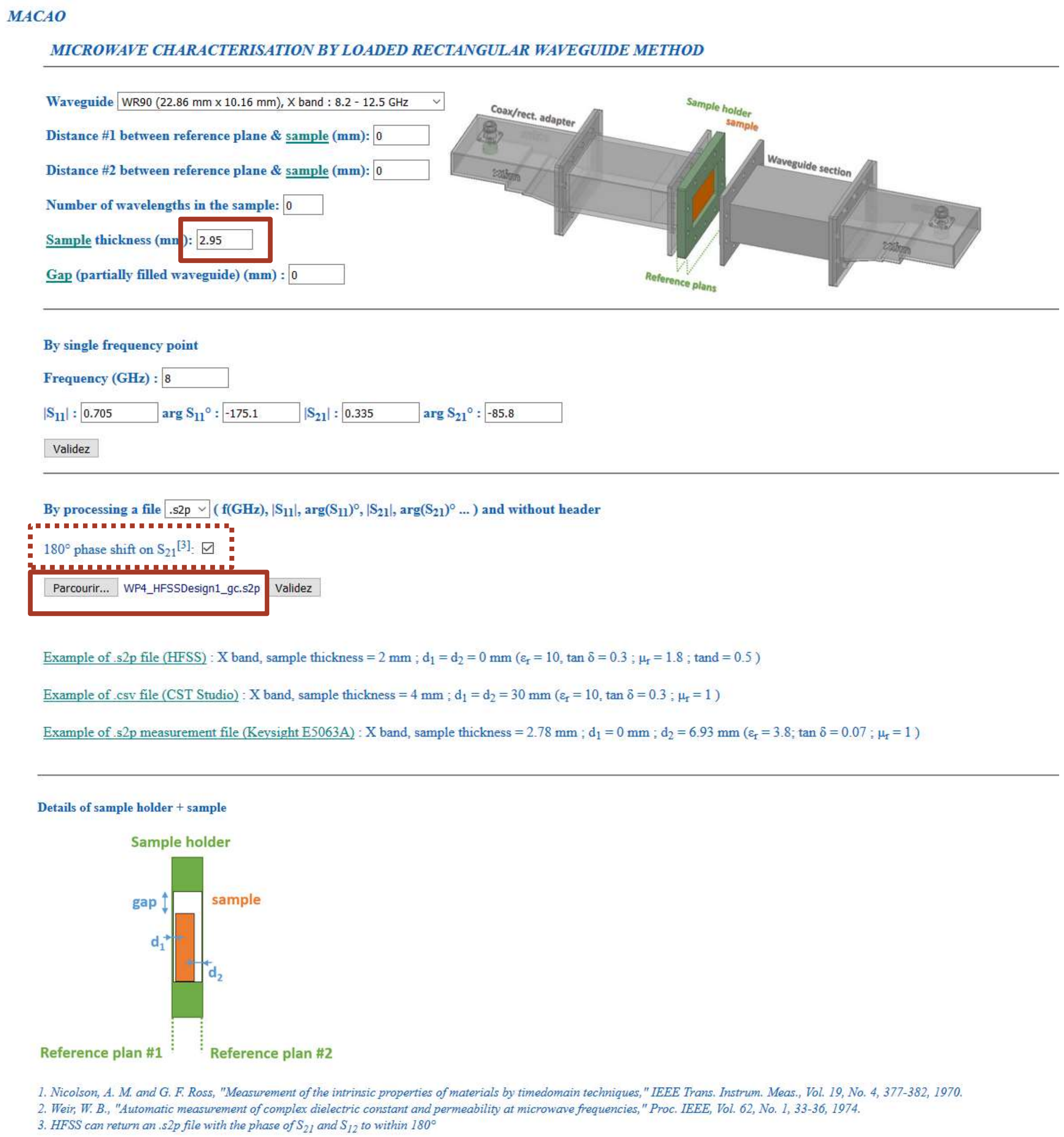
- Go to **Excitations**, **Port1**, **Post Processing** tab
- Tick **Deembed**
- Set **Distance** to 10 mm
- Click on **OK**
- Do the same for **Port2**

Now we have to export the S-paramters, as Touchstones Files (.s*n*p):

- Go to **Solution Data**
- Click on **Export Matrix Data**
- Name the file "PW4" and click **OK**
- Tick **Do Not Override Solution Normalization**
- Un-tick **Include Gamma and Impedance Comments**
- Click on **OK**

- Now, with a web navigator, go to http://achille.xlim.fr/
- Then go in the sub-section Caractérisation de matériaux (= material characterization) and open the page "*Guide rectangulaire pour la caractérisation de matériaux » (=rectangular waveguide for material characterization)*

The following page should open:



– **Figure** 7 –

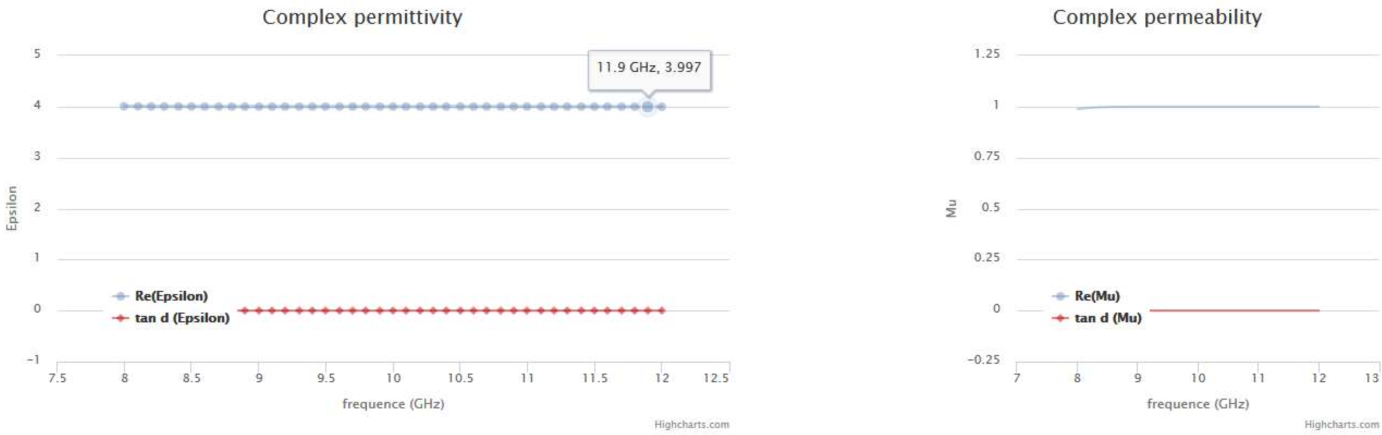
Enter the parameters of the simulation, as shown in the figure above.

To browse to your file, click on **Parcourir**.

There is a known « bug phase » with HFSS, and it is sometimes required to add π to the phases of transmissions. For a lossless structure, if the phases Φ_{ij} are OK they should verify the following relationship (in radians):

$$\Phi_{12} = \frac{\Phi_{11} + \Phi_{22}}{2} + \frac{\pi}{2}$$

- Test this formula for the central frequency in the **Solution data** window of HFSS.
- If the formula is satisfied (or close to be satisfied), do not tick **180° phase shift on S21** on the web page, otherwise tick it.
- Click on **Valider**, you should then obtain the following results where the permittivity is calculated for each given frequency:



– **Figure** 8 –