Semester S1

-

Foundations of electromagnetic wave propagation

Practical Work PW1

-

Electromagnetic analysis of planar microwave circuits: introduction to "Momentum"

I. INTRODUCTION

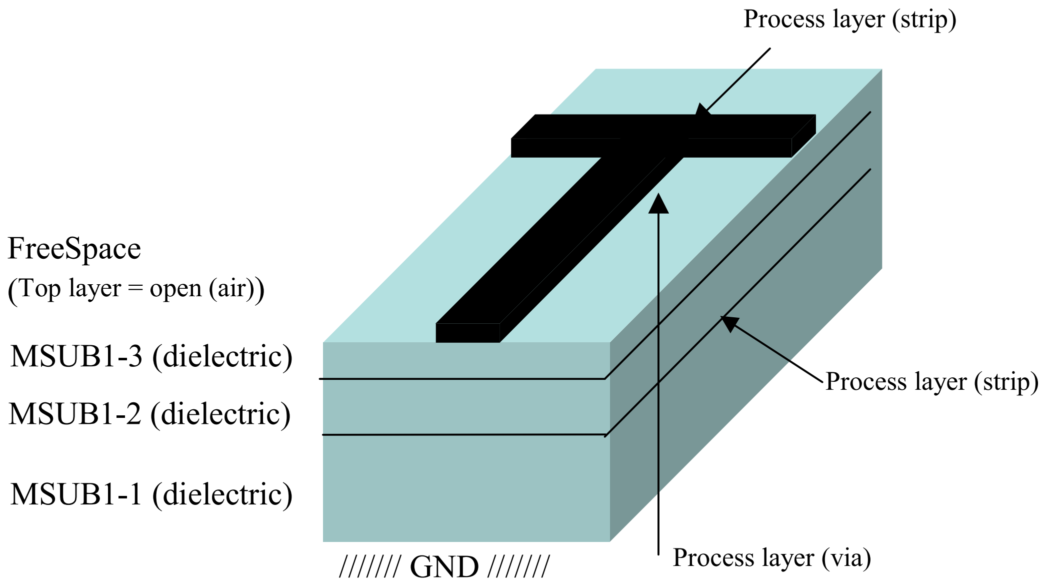
When designing circuits, simulation is an essential step in predicting the electromagnetic and electrical behavior of the devices developed.

The software used can be classified into two families: circuit simulation software and electromagnetic simulation software.

Circuit simulation software such as ADS (HP), SUPERCOMPACT (COMPACT SOFTWARE) based on the nodal method, allow fast simulations of complex circuits, including localized elements (chokes, capacities,...), distributed elements represented by equivalent models (microstrip lines,...) and active elements (transistors,...). However, the models used have limitations (in frequency, geometry, etc.). In addition, due to the application of a segmentation method, these software do not take into account the indirect electromagnetic interactions between all the components.

To overcome these problems, it is then necessary to use electromagnetic simulators such as CST Microwave Studio (CST), HFSS (ANSYS), Momentum (HP), Sonnet (Sonnet), Microwave Explorer (Compact Software), etc… Each of these simulators has advantages and drawbacks depending on the nature of the circuit to be analyzed: some are adapted to simulate multilayer planar and planar circuits (Momentum) while others are more efficient to simulate volume structures (HFSS, CST). In all cases, they allow the circuits to be analyzed rigorously but require a large amount of memory and a significant calculation time. Yet, they are essential for the theoretical analysis of certain kind of devices. Nowadays, the increase of computer's performance and their parallelization make it possible to consider them as a solution.

In this PW, we propose to use Momentum software which applies to the treatment of quasi-planar circuits (method 2D½): the structures are multilayered; the metallization of the different layers can be connected by metallized holes; the devices can be closed in a metal shield or radiate towards infinity (figure 1).



– Figure 1 –

This software is based on a numerical electromagnetic method called the Method of Moments (MoM). By this technique, the conductors are "meshed", divided into simple elements (triangles or rectangles). The surface currents induced on the conductors are decomposed in a function based on each of these elements. The coefficients of this decomposition are therefore the surface currents, the unknown of the problem.

This numerical technique allows one to obtain an integral equation, directly deduced from the Maxwell equations, as a matrix system. The output parameters of the software are:

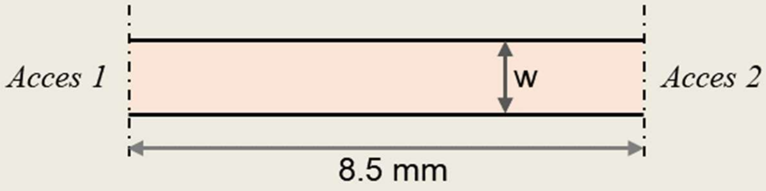
- The [S] parameters concerning the physical access of the devices. They take into account all the interactions that can occur between the metallization layers of the device.
- A set of quantities, directly deduced from the field values on the conductors.

To use the Momentum software, it is necessary to describe all the physical and geometrical characteristics of the parameters of the device under test. Different possibilities are available for the end-user in the HP ADS environment:

- The circuit can be described by the elements contained in the ADS library. Momentum then works directly from the data described in the **Schematic** window.
- The circuit under test cannot be described from the elements of the library. It is then drawn in the **Layout** window. The nature of the test(s) is given in the Momentum window.

II. ANALYSIS OF A MICROSTRIP LINE

The circuit to be tested is described in Figure 2.



Technology: **Microstrip line:** w = 0.5 mm

Substrate: **Permittivity:** $\varepsilon_r = 9.8$; $tg\delta = 10^{-4}$

Substrate height: H = 0.5 mm

Metallization thickness: Epm = 8 µm

Conductivity: $\sigma = 5.7\ 10^7$ S/m

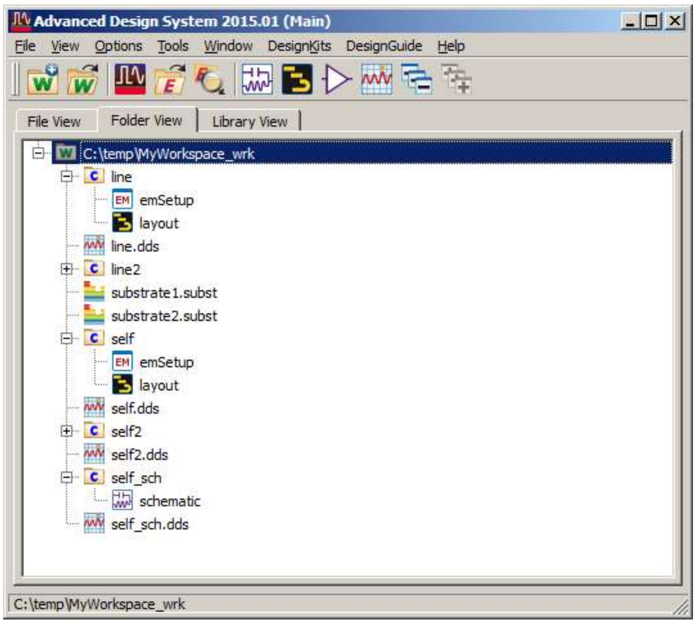
Roughness: RGH = 0 µm

– Figure 2 –

1. OPENING ADS SOFTWARE

Click on the ADS icon on the desktop:

The following window should appear:



– Figure 3 –

Then, depending on the version installed on your computer, two different ways are possible to create a new project.

a. Project creation (V2012 to V2016)

Create a new workspace (by clicking on the button ).

Then follow the different steps (click Next to move from a tab to another) and:

- enter "PW1_wrk" for **workspace name** in **Workspace Name tab**,
- select **ADS Libraries/AnalogRF** and **DSP** in **Add libraries tab**,
- choose **Standard ADS layers, 0.0001 millimeter layout resolution** in **Technology tab**.

Finally, open a new **Momentum** window by clicking on button 

b. Project creation (v2020 and above)

Create a new workspace (by clicking on the button ).

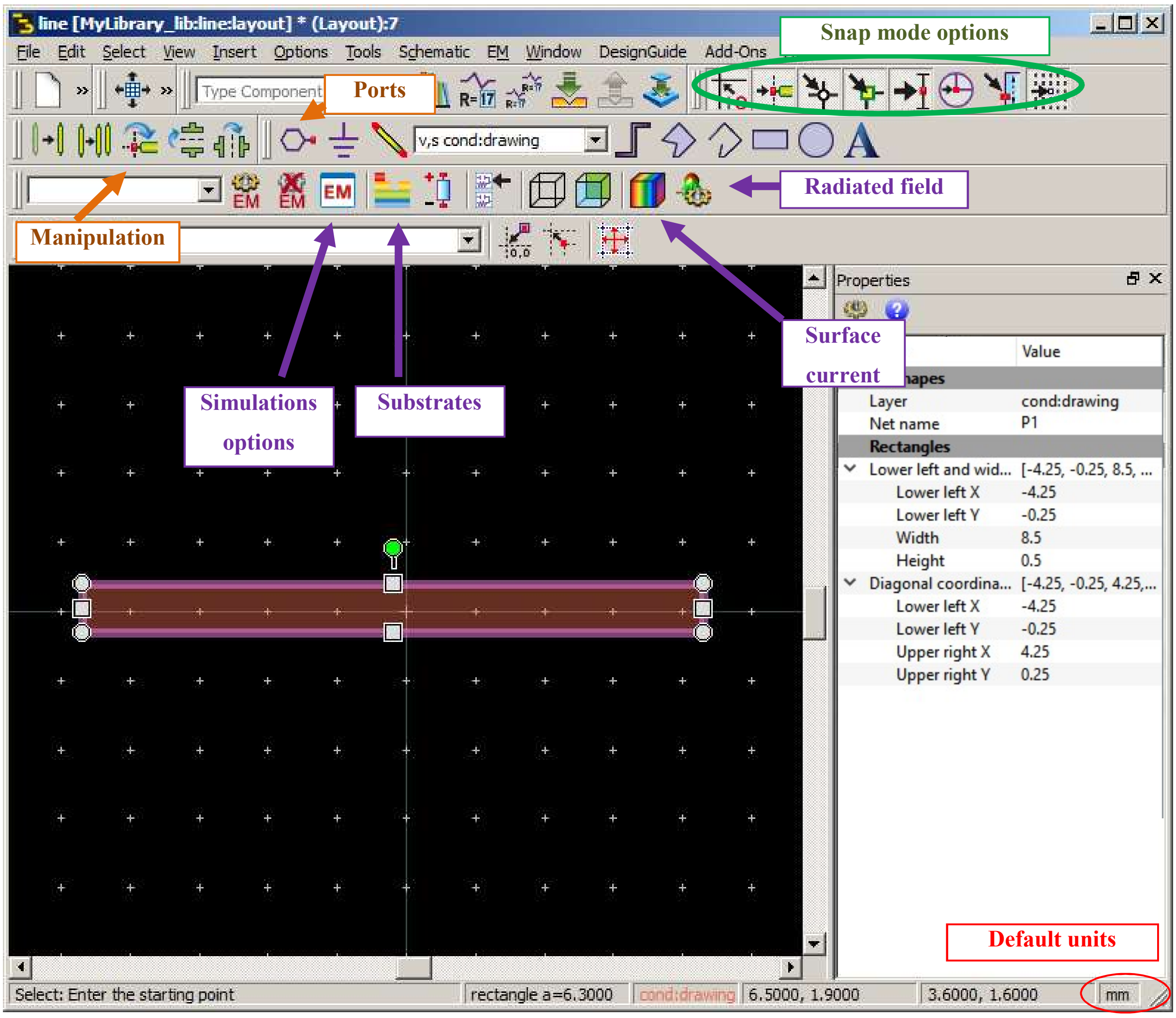
Then follow the different steps:

- enter "PW1_wrk" for **workspace name**,
- click on show advanced,
- verify that you have the libraries **ADS Libraries/AnalogRF** and **DSP,**
- tick **set up layout technology immediately after creating the library**
- choose **Standard ADS layers, 0.0001 millimeter layout resolution** in the other window that appears just after creating the workspace.

Finally, open a new **Momentum** window by clicking on button 

2. Circuit creation

The **Layout** window is shown in figure 4:



– Figure 4 –

At the bottom on the right the units "mm" should appear, they indicate that the default unit is the millimeter.

a. Setting the mouse movement step:

- In the **Layout** window (Momentum):

  LC (**Left Click**) on **Options** > **Preferences**.

  In the tab **Grid / Snap**, and in **Spacing, Grid Snap Distance (in Layout Units)**:

  Under **X** type 0.25

  Under **Y** type 0.25

  LC on **Apply**

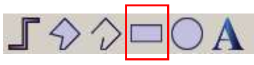
  LC on **OK**

Notice at the bottom of the window the indication of the mouse position, in real and relative values.

b. Drawing

You can draw the line directly with the mouse by creating rectangles and then modify their properties, or directly by entering its coordinate points:

- Open the **Coordinate Entry** window (**LC** on **Insert** then **LC** on **Coordinate Entry**).
- In the **Layout** window:

  LC on the **rectangle** in the toolbar: 
- In the **Coordinate Entry** window:

  Enter the coordinates **x** = 4.25 mm and **y** = -0.25 mm.

  LC on **Apply**

  Then enter the coordinates **x** = -4.25 mm and **y** = 0.25 mm.

  LC on **OK**

Check the dimensions by applying the procedure hereafter.

c. Measuring the length of the line:

- In the **Momentum** window

  RC (Right Click) in the window

  LC on **Measure …**

  LC on the points that delimit the stub to obtain its length

  Check that this dimension is equal to the one indicated in the **Properties** of the rectangle.

d. Creation of the excitation ports (pin) of the structure:

- In the **Momentum** window:

  LC on the icon  (*insert pin*) on the menu bar

  Place a port in the beginning and in the end of the line (access 1 and access 2)

  Use the **Increment Component Orientation By -90** rotation icon if necessary to choose a horizontal orientation of the access.

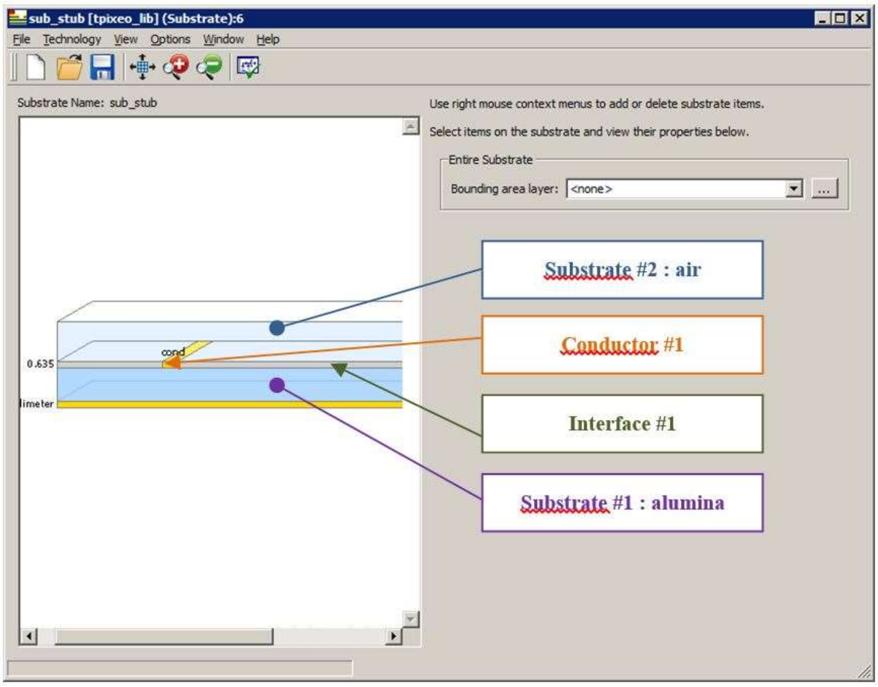
Note: The position of the ports in the edge of the line does not matter but the ports must be positioned in the edges and not inside.

RC on each ports then **LC** on **Properties**…

- Check that the reference impedances of the ports are at 50 Ohms (**properties** area).

e. Creation of the substrate:

- In the **Substrate** window (**LC** on **EM** then **Substrate** OR **CG** on):



– Figure 5 –

LC on **File** then **New**

Name the substrate « sub_line ».

LC on **substrat #1**: the substrate properties should be displayed on the right

- In the window about properties of **substrat #1**:

  Thickness: 0.5 mm

  LC on the button […] to the right of **Material**, the **Material Definitions** window opens.

- In the **Material Definitions** window:

  Permittivity – **Real**: 9.8

  Permittivity – **TanD**: 0.0001

LC on **conductor #1**: the properties of the metallization must be displayed.

- In the window about properties of **conductor #1**:

Layer: **cond**

LC on the button […] to the right of **Material**, the **Material Definitions** window opens.

- In the **Material Definitions** window**:**

  LC on **Add Conductor**

  Conductivity – Real: 5.7e7 Siemens/m

  LC on **OK**

- In the window about properties of **conductor #1**:

  Material: choose the conductor you have just created (normally « Conductor_1 »)

  Operation: Intrude into substrate

  Position: **Above interface**

  Thickness: 8 µm

  LC on **Check** to check for errors.

  LC on **Save**

By default, the properties of **substrat #2** (air) and **interface #1** are correct.

f. Creation of the frequency plan:

LC on **EM** then **Simulation Setup** OR **LC** on EM, the **EM Simulation Setup** window opens:

- In the **EM Simulation Settings** window:

  LC on **Frequency Plan**

  LC on **Add**

  On line #1, **Type**: **Linear**.

  On line #1, **Fstart**: 1 GHz

  On line #1, **Fstop**: 16 GHz

  On line #1, **Step**: 1 GHz

  LC on **Add**

  On line #2, **Type**: **Adaptive**.

  On line #2, **Fstart**: 1 GHz

  On line #2, **Fstop**: 16 GHz

  On line #2, **Npts**: 25

The choice of the type of sweep tells the software how often the calculations will be performed:

Single: indicates a particular frequency point.

Linear: performs a standard scan, with a frequency step. Be careful, EM calculations are long, the step should not be too "fine".

Adaptive: the software itself decides at which frequencies to perform EM calculations.

g. Checking the substrate:

When you create multiple substrates in the same **workspace**, it is necessary to verify that the correct substrate will be used for the calculations.

- In the **EM Simulation Setup** window:
  LC on **Substrate**
- Check that "sub_line" appears in **Substrate** (right), otherwise select it.

h. Creation of the mesh:

- In the **EM Simulation** window:
  LC on **Options**
  LC on **Mesh**
  Mesh Frequency: 16 GHz (= Fstop of the frequency plan)
  Cells / wavelength: 20. **Cells / Wavelength** indicates the number of elements per wavelength at the frequency given in the box **Mesh Frequency**.
  Tick **Edge mesh** (Auto-determine Edge Width)
  Tick **Transmission line mesh**
  Number of cells in width: 3.
  Close the **EM simulation Setup** window and save.
  LC on **Output Plan**
  If **All generated frequencies** is not ticked for **Save currents for**, tick it.

The realization of the mesh of a structure allows to divide the conductors into simple elements (outside the ground plane) in order to discretize the problem to be solved. The finer the mesh size, the more accurate the problem resolution (= EM calculation) will be, but the longer the calculations will be.

3. CALCULATION AND EVALUATING THE RESULTS

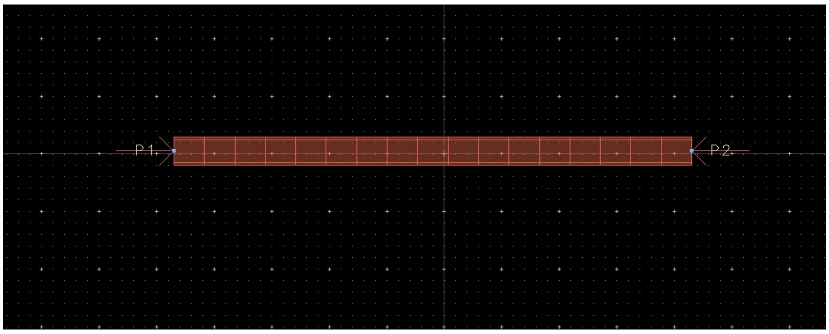
The design is ready to be simulated.

a. Starting the calculation:

- In the **Momentum** window:

  LC on **EM** then **Simulate** (or **LC** on the button EM)

- A **DataDisplay** window appears where you can follow the different calculation steps (Mesh computation Successful, ……, Momentum Analysis is finished).



– Figure 6 –

- At the end of the simulation, a graphical window opens where the responses of the different [S] parameters in module and phase are plotted.

What comments can you do on these results, and especially the matching?

b. Visualization of the surface current:

- In the **Momentum** window, click on **EM** then **Post-Processing** then **Visualization** or on the button .
- An **Agilent Momentum Visualization (stub)** window opens, in which we will be able to visualize the surface currents:

  LC on the tab **Solution Setup**

  Select 8e9

  LC on the tab **Plot Properties**

  LC on the tab **Shaded**

  Tick **log scale**

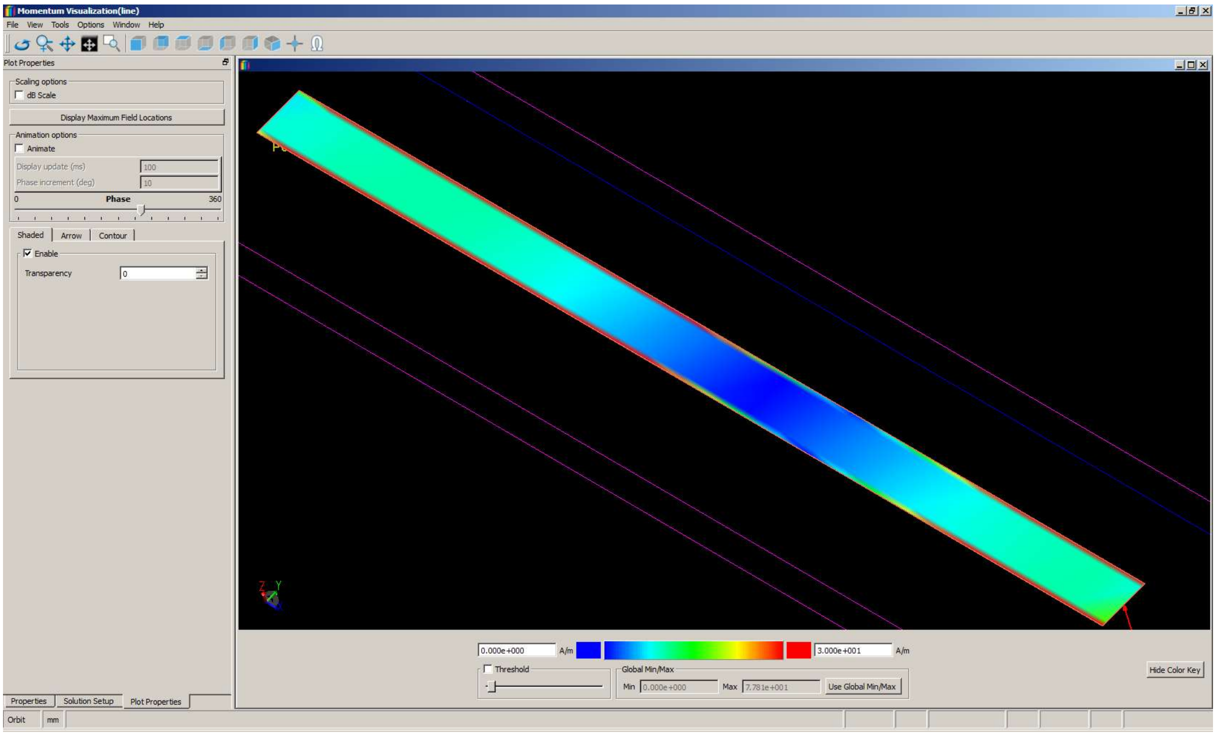
  LC on the tab **Arrow**

  Tick **Enable**

Tick **Animate**

- You then visualize the amplitude of the current at 8 GHz as a function of time.

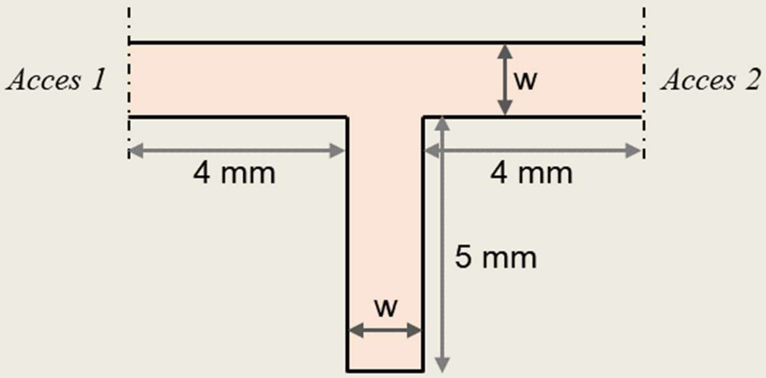
➢ Is what you observe consistent with the [S] parameters obtained?



– Figure 7 –

III. ANALYSIS OF A MICROSTRIP STUB

The circuit to be tested is described in Figure 8.



Technology: **Microstrip line:** w = 0.5 mm

Substrate: **Permittivity:** ε_r = 9.8; tgδ = 10^{-4}
Substrate height: H = 0.5 mm
Metallization thickness: Epm = 8 µm
Conductivity: σ = 5.7 10^7 S/m
Roughness: RGH = 0 µm

– Figure 8 –

1. CIRCUIT CREATION

a. Drawing

Create the structure here above. You can start from the previous design:

LC on **File** then **Save as**, and give it a new name ("stub" for example)

You can now modify the structure.

b. Substrate

Use the same substrate as before.

c. Creation of the frequency plan:

All the other environment parameters are the same as for the line (mesh,…) except the frequency plan.

To update it, **LC** on **EM** then **Simulation Setup** OR **LC** on EM, the **EM Simulation Setup** window opens:

- In the **EM Simulation Setup** window:
  Delete the existing lines (**LC** on **Remove** button)
  LC on **Frequency Plan**
  LC on **Add**

On line #1, **Type**: **Single**.

On line #1, **Fstart**: 6 GHz

LC on **Add**

On line #2, **Type**: **Single**.

On line #2, **Fstart**: 12 GHz

LC on **Add**

On line #3, **Type**: **Adaptive**.

On line #3, **Fstart**: 1 GHz

On line #3, **Fstop**: 16 GHz

On line #3, **Npts**: 25

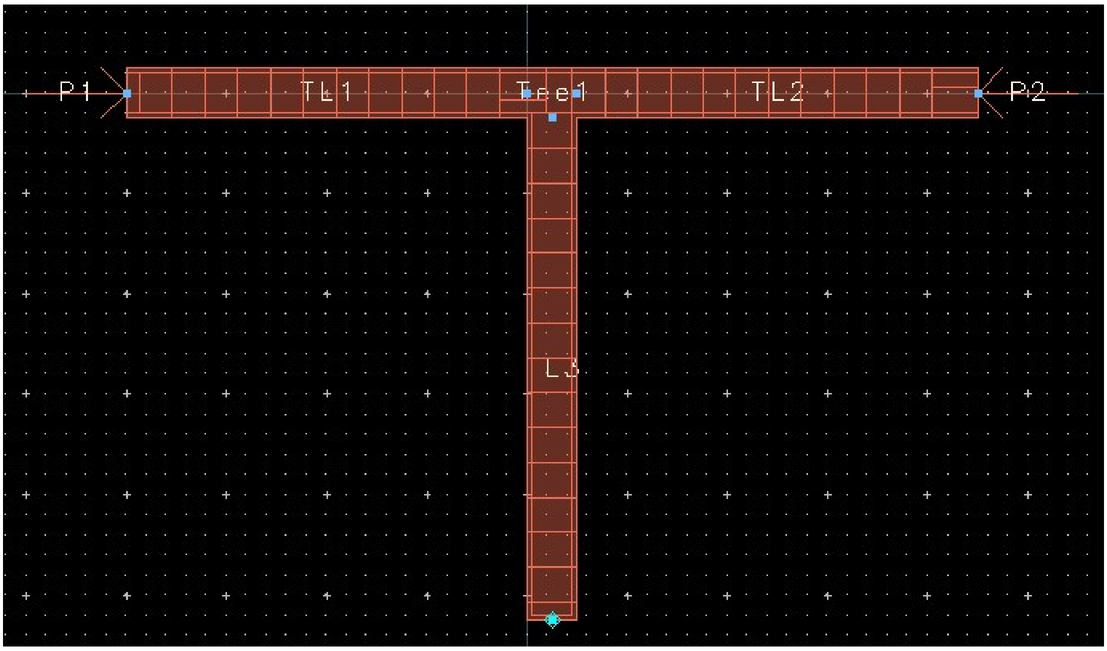
2. Calculation and evaluating the results

a. Visualization of the response:

Start the simulation by clicking on the button 

At the end of the simulation, a window should open.

➢ What comments can you make on the results? Which phenomena is/are the cause of what you observe?



– Figure 9 –

b. Visualization of the surface current:

- Current can be visualized the same way as previously, by clicking on the button.
- An **Agilent Momentum Visualization (stub)** window opens, in which we will be able to visualize the surface currents:

    LC on the tab **Solution Setup**

    Select 6e9

    LC on the tab **Plot Properties**

    LC on the tab **Shaded**

    Tick **log scale**

    LC on the tab **Arrow**

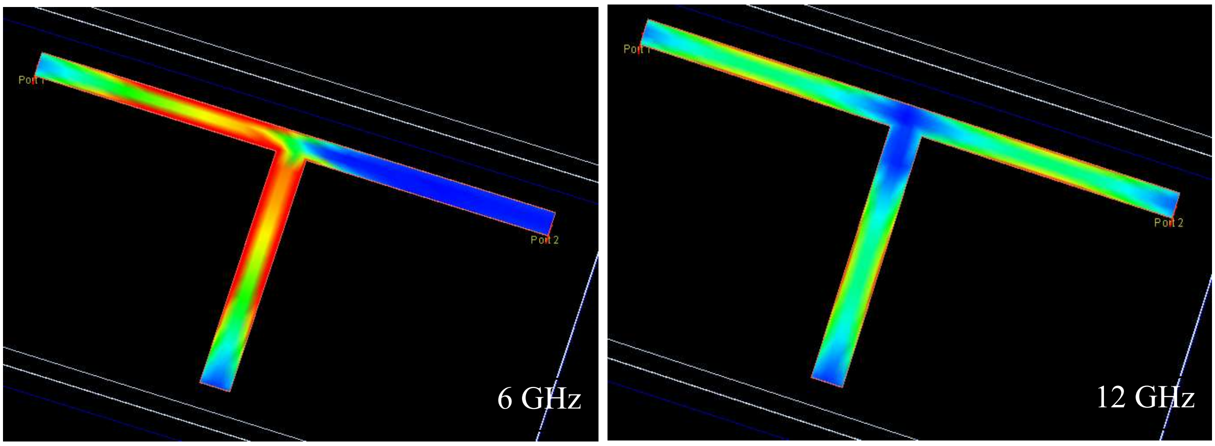
    Tick **Enable**

    Tick **Animate**

- You then visualize the amplitude of the current at 6 GHz as a function of time.

➢ Is what you observe consistent with the [S] parameters obtained?

➢ Visualize the fields at 12 GHz. Same question.

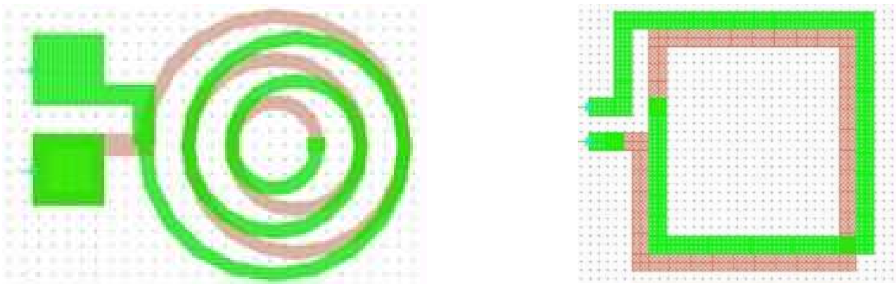


– Figure 10 –

IV. ELECTROMAGNETIC CHARACTERIZATION OF A MULTILAYER CIRCUIT

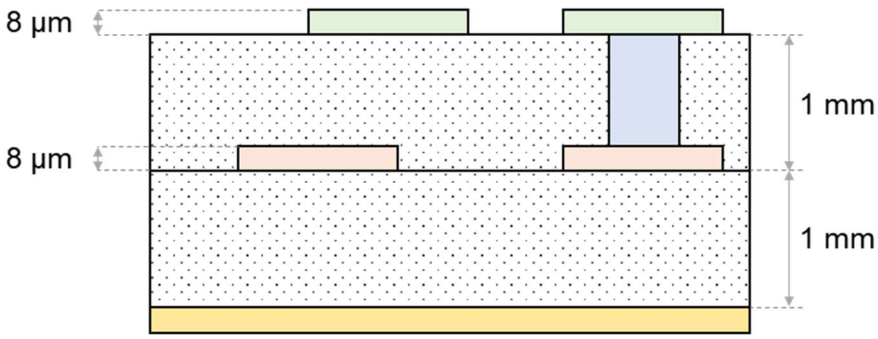
The objective is now to characterize a multilayer circuit: a self-inductance in microstrip technology.

The self-inductances on microstrip technology, figure 11, have low quality factors (50) for values in the range of 1 to 25 nH on substrate with $\varepsilon_r = 4$ (ceramic).



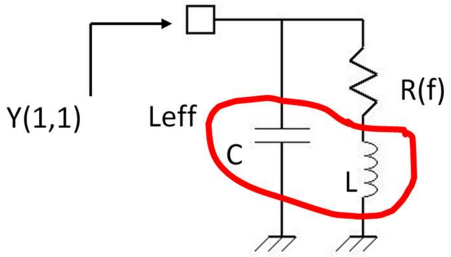
– Figure 11 –

Thanks to judicious topologies, it is possible to improve the value of self-inductance and reduce the value of parasitic capacitance by using several engraved and stacked substrates (Figure 11) and an offset between the patterns of each substrate.



– Figure 12 –

This type of self-inductance can be modelled by the following equivalent scheme:



– Figure 13 –

where L_{eff} is the effective self-inductance

C the parasitic capacitance

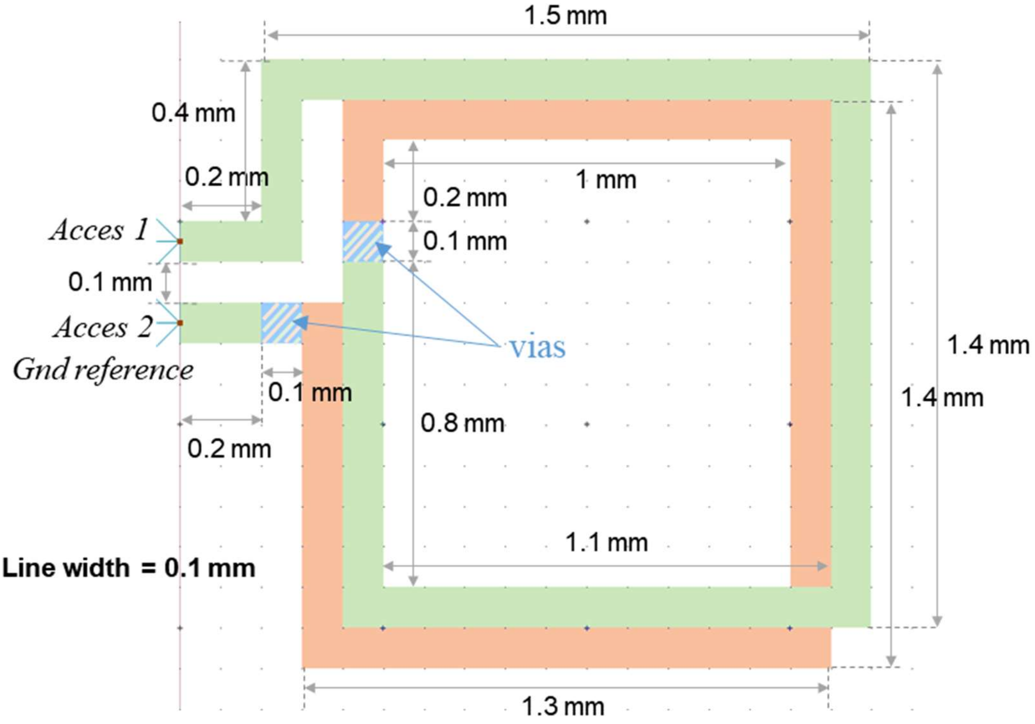
R is the equivalent resistance which depends on the frequency

Notes :

- The parasitic capacitance C is significantly reduced by the shifting of the wound patterns.
- Rectangular patterns are as effective as circular patterns.

1. CREATION OF THE INDUCTANCE

With the Momentum software, perform the layout in Figure 14 and determine the maximum value of the quality factor and self-inductance in the frequency range from 0.5 to 10 GHz (10 maximum frequency points). Also deduct from Y(1,1), the value of the parasitic capacitance.



– **Figure 14** –

Metallic conductors on different levels are described in the same **layout** window.

a. Drawing

- Open a new **layout** window (**LC** on **File** then **LC** on **New** or **LC** on the actual **layout** window button). Name it « inductance ».

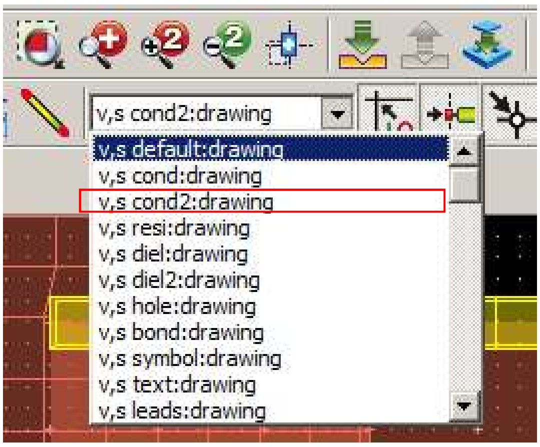
- Set the **Options** > **Préférences** > **Grid/snap**
  Snap Grid Distance to 0.05
  Snap Grid Per Minor Display Grid to **2**
  Minor Grid Per Major Display Grid to **5**

- Draw the structure as above.

The drawing layer **cond** is used to describe top metal layer while **cond2** is used for the inner metal layer. Use the layer **hole** for the via.

To switch from a drawing layer to another:

- In **Layout** window, select **cond2** (instead of cond) in the combo-box as shown below.



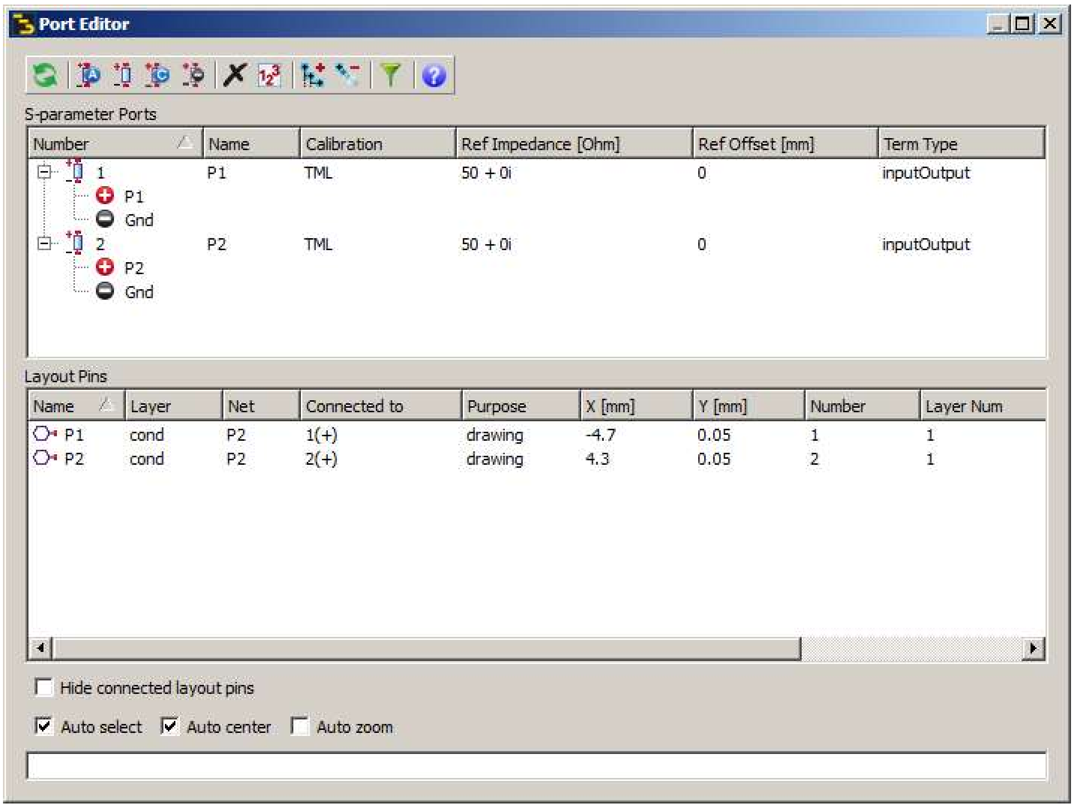
– Figure 15 –

Cond2 should now be indicated in the bottom of the **Layout** window. The new conductors are in yellow on the screen.

b. Creation of the excitation ports (pin) of the structure

- Put the 1st pin at the extremity of the top (on the screen) line.
- Put the 2nd pin on the line just under.
- Then open the port editor by clicking on the button (or go to **EM** > **Port editor**)

The following window should open:



– Figure 16 –

On this window, drag the P2 port and drop it on the GND of port 1. This realizes a pairing between the 2 ports which leads to a better simulation of the self, short-circuited at one extremity. You normally obtain this:

Number	Name	Calibration	Ref Impedance [Ohm]	Ref Offset [mm]	Term Type
1	P1	TML	50 + 0i	0	inputOutput
P1					
P2					

– **Figure 17** –

c. Substrates descriptions

The substrate to define is the one in Figure 11.

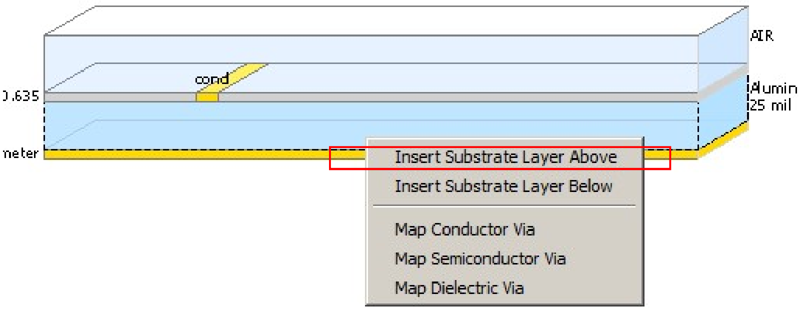
Now, 2 substrate layers, 2 conductor layers and 1 via layer have to be defined.

- In a substrate window:

  LC on **File** then **New**

  Call it “inductance_sub”.

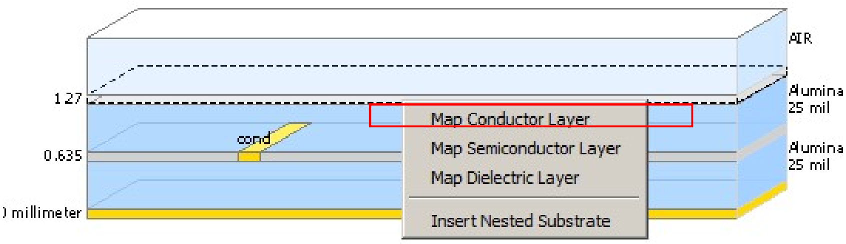
  RC on Alumina, you should obtain the following popup menu:



– **Figure 18** –

  RC on **Insert Substrate Layer Above**. This create a new substrate.

  RC on the interface between this substrate and the substrate AIR:



– **Figure 19** –

LC on **Map Conductor Layer**

- For inferior substrate:
  Material > **Permittivity** – **Real**: 4 (create material if required).

  Material > **Permittivity** – **TanD**: 0.001

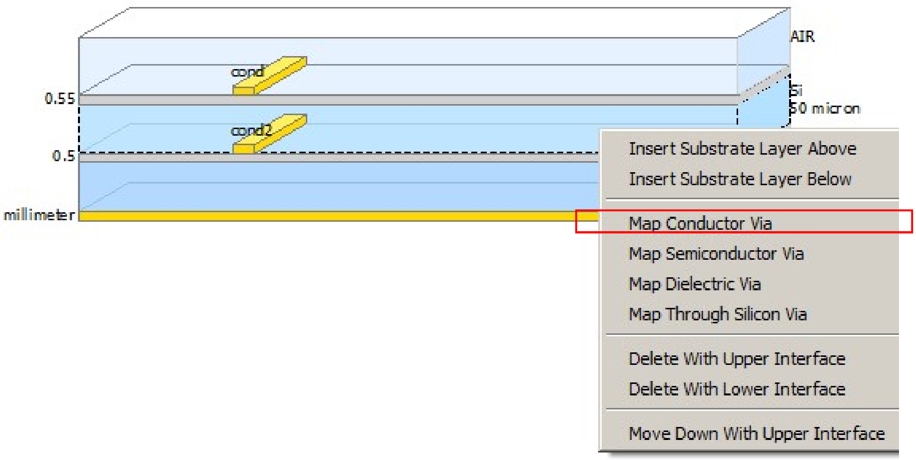
  Thickness: 1 mm
- For intermediary substrate:
  Material > **Permittivity** – **Real**: 4.

  Material > **Permittivity** – **TanD**: 0.001

  Thickness: 1 mm

For the via layer:

LC on the intermediary substrate:



– Figure 20 –

LC on **Map conductor Via.**

In **Conductor Via** area**,** select **hole** as the drawing layer (do not forget to set its conductivity to the same value as for other metals).

For conductors, as previously, define:

- For bottom conductor:
  Layer: cond2

  Material > **Loss Parameters** > **Real**: 4.1e7 S/m (create material if required).

  Tick **Intrude into substrate** and **Above Interface.**

  Thickness: 8 µm

- For superior conductor:
  Layer: cond

  Material > **Loss Parameters** > **Real**: 4.1e7 S/m.

  Tick **Intrude into substrate** and **Above Interface.**

  Thickness: 8 µm

- Check.
- If no errors, save the substrate.

d. Frequency plan creation:

- In the **EM Simulation Setup** window (**LC** on **EM** then **Simulation Setup** or **LC** on 

):
  LC on **Frequency Plan**

  LC on **Add**

  line #1, **Type**: **Adaptive**.

  line #1, **Fstart**: 0.5 GHz

  line #1, **Fstop**: 10 GHz

  line #1, **Npts**: 10

e. Substrate check:

- In the **EM Simulation Setup** window:
  LC on **Substrate**
- Check that « inductance_sub » is selected as the current substrate in the **Substrate** combo-box (on the right).

f. Mesh creation:

- In **EM Simulation** window:
  LC on **Options**

  LC on **Mesh**

  Mesh Frequency: 10 GHz

  Cells/wavelength: 15

  Tick **Edge mesh** (Auto-determine Edge Width)

  Tick **Transmission line mesh**

  Number of cells in width: 6.

Mesh Frequency.

Close the **EM simulation Setup** window and save.

2. CALCULATIONS AND RESULTS

Start the simulations.

At the end of the calculations, a display window should appear.

We want to know the effective inductance and the quality factor of the structure.

- The effective self-inductance can be expressed as:

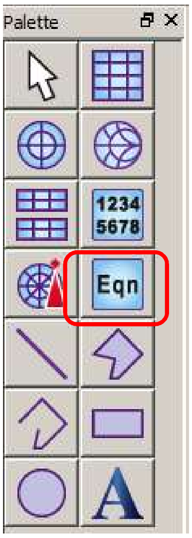
  $L_{eff} = -1 / (\text{imag}(Y(1,1))*\omega)$

  Where Y(1,1) is the admittance of the device and ω the generator pulsation

- The quality factor is given by:

  $Q = -\ \text{imag}(\ Y(1,1)) / (\text{real}(Y(1,1)))$

In the display window, create 2 equation blocks, one for Leff and one for Q, and enter the equations above:



– Figure 21 –

Then display the results on graphs:

– Figure 22 –

- In the window that opens, select **Equations** in the combo-box **Datasets and equations**.
- Double click on Leff.
- Create another graph for Q and do the same thing to plot the values.

➢ Comment the results.

3. Variants

- Repeat the same calculations for the case where conductors make a half turn more and less. Compare the results and conclusion. Trace the evolution of the self-inductance and the quality factor according to the number of revolutions.
- To see the capacitive effect of two superimposed microstrip lines, modify the first layout so that at least two line sections are facing each other. Conclusion.

APPENDIX

Conversions between parameters normalized to Z0 = 1 of a quadrupole*,

$$\text{avec } \Delta^K = K_{11}K_{22} - K_{12}K_{21}$$

	S	Z	Y	H	A
S	$\begin{pmatrix} b_1 \\ b_2 \end{pmatrix} = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix} \begin{pmatrix} a_1 \\ a_2 \end{pmatrix}$	$S_{11} = \frac{(Z_{11}-1)(Z_{22}+1)-Z_{12}Z_{21}}{(Z_{11}+1)(Z_{22}+1)-Z_{12}Z_{21}}$ $S_{12} = \frac{2Z_{12}}{(Z_{11}+1)(Z_{22}+1)-Z_{12}Z_{21}}$ $S_{21} = \frac{2Z_{21}}{(Z_{11}+1)(Z_{22}+1)-Z_{12}Z_{21}}$ $S_{22} = \frac{(Z_{11}+1)(Z_{22}-1)-Z_{12}Z_{21}}{(Z_{11}+1)(Z_{22}+1)-Z_{12}Z_{21}}$	$S_{11} = \frac{(1-Y_{11})(1+Y_{22})+Y_{12}Y_{21}}{(1+Y_{11})(1+Y_{22})-Y_{12}Y_{21}}$ $S_{12} = \frac{-2Y_{12}}{(1+Y_{11})(1+Y_{22})-Y_{12}Y_{21}}$ $S_{21} = \frac{-2Y_{21}}{(1+Y_{11})(1+Y_{22})-Y_{12}Y_{21}}$ $S_{22} = \frac{(1+Y_{11})(1-Y_{22})+Y_{12}Y_{21}}{(1+Y_{11})(1+Y_{22})-Y_{12}Y_{21}}$	$S_{11} = \frac{(h_{11}-1)(h_{22}+1)-h_{12}h_{21}}{(h_{11}+1)(h_{22}+1)-h_{12}h_{21}}$ $S_{12} = \frac{2h_{12}}{(h_{11}+1)(h_{22}+1)-h_{12}h_{21}}$ $S_{21} = \frac{-2h_{21}}{(h_{11}+1)(h_{22}+1)-h_{12}h_{21}}$ $S_{22} = \frac{(1+h_{11})(1-h_{22})+h_{12}h_{21}}{(h_{11}+1)(h_{22}+1)-h_{12}h_{21}}$	$S_{11} = \frac{A+B-C-D}{A+B+C+D}$ $S_{12} = \frac{2(AD-BC)}{A+B+C+D}$ $S_{21} = \frac{2}{A+B+C+D}$ $S_{22} = \frac{-A+B-C+D}{A+B+C+D}$
Z	$Z_{11} = \frac{(1+S_{11})(1-S_{22})+S_{12}S_{21}}{(1-S_{11})(1-S_{22})-S_{12}S_{21}}$ $Z_{12} = \frac{2S_{12}}{(1-S_{11})(1-S_{22})-S_{12}S_{21}}$ $Z_{21} = \frac{2S_{21}}{(1-S_{11})(1-S_{22})-S_{12}S_{21}}$ $Z_{22} = \frac{(1-S_{11})(1+S_{22})+S_{12}S_{21}}{(1-S_{11})(1-S_{22})-S_{12}S_{21}}$	$\begin{pmatrix} V_1 \\ V_2 \end{pmatrix} = \begin{bmatrix} Z_{11} & Z_{12} \\ Z_{21} & Z_{22} \end{bmatrix} \begin{pmatrix} I_1 \\ I_2 \end{pmatrix}$	$Z_{11} = \frac{Y_{22}}{\Delta^Y}$ $Z_{12} = \frac{-Y_{12}}{\Delta^Y}$ $Z_{21} = \frac{-Y_{21}}{\Delta^Y}$ $Z_{22} = \frac{Y_{11}}{\Delta^Y}$	$Z_{11} = \frac{\Delta^h}{h_{22}}$ $Z_{12} = \frac{h_{12}}{h_{22}}$ $Z_{21} = \frac{-h_{12}}{h_{22}}$ $Z_{22} = \frac{1}{h_{22}}$	$Z_{11} = \frac{A}{C}$ $Z_{12} = \frac{\Delta^A}{C}$ $Z_{21} = \frac{1}{C}$ $Z_{22} = \frac{D}{C}$
Y	$Y_{11} = \frac{(1-S_{11})(1+S_{22})+S_{12}S_{21}}{(1+S_{11})(1+S_{22})-S_{12}S_{21}}$ $Y_{12} = \frac{-2S_{12}}{(1+S_{11})(1+S_{22})-S_{12}S_{21}}$ $Y_{21} = \frac{-2S_{21}}{(1+S_{11})(1+S_{22})-S_{12}S_{21}}$ $Y_{22} = \frac{(1+S_{11})(1-S_{22})+S_{12}S_{21}}{(1+S_{11})(1+S_{22})-S_{12}S_{21}}$	$Y_{11} = \frac{Z_{22}}{\Delta^Z}$ $Y_{12} = \frac{-Z_{12}}{\Delta^Z}$ $Y_{21} = \frac{-Z_{21}}{\Delta^Z}$ $Y_{22} = \frac{Z_{11}}{\Delta^Z}$	$\begin{pmatrix} I_1 \\ I_2 \end{pmatrix} = \begin{bmatrix} Y_{11} & Y_{12} \\ Y_{21} & Y_{22} \end{bmatrix} \begin{pmatrix} V_1 \\ V_2 \end{pmatrix}$	$Y_{11} = \frac{1}{h_{11}}$ $Y_{12} = \frac{-h_{12}}{h_{11}}$ $Y_{21} = \frac{h_{21}}{h_{11}}$ $Y_{22} = \frac{\Delta^h}{h_{11}}$	$Y_{11} = \frac{D}{B}$ $Y_{12} = \frac{-\Delta^A}{B}$ $Y_{21} = \frac{-1}{B}$ $Y_{22} = \frac{A}{B}$
H	$h_{11} = \frac{(1+S_{11})(1+S_{22})-S_{12}S_{21}}{(1-S_{11})(1+S_{22})+S_{12}S_{21}}$ $h_{12} = \frac{2S_{12}}{(1-S_{11})(1+S_{22})+S_{12}S_{21}}$ $h_{21} = \frac{-2S_{21}}{(1-S_{11})(1+S_{22})+S_{12}S_{21}}$ $h_{22} = \frac{(1-S_{11})(1+S_{22})-S_{12}S_{21}}{(1-S_{11})(1+S_{22})+S_{12}S_{21}}$	$h_{11} = \frac{\Delta^Z}{Z_{22}}$ $h_{12} = \frac{Z_{12}}{Z_{22}}$ $h_{21} = \frac{-Z_{21}}{Z_{22}}$ $h_{22} = \frac{1}{Z_{22}}$	$h_{11} = \frac{1}{Y_{11}}$ $h_{12} = \frac{-Y_{12}}{Y_{11}}$ $h_{21} = \frac{Y_{21}}{Y_{11}}$ $h_{22} = \frac{\Delta^Y}{Y_{11}}$	$\begin{pmatrix} V_1 \\ I_2 \end{pmatrix} = \begin{bmatrix} h_{11} & h_{12} \\ h_{21} & h_{22} \end{bmatrix} \begin{pmatrix} I_1 \\ V_2 \end{pmatrix}$	$h_{11} = \frac{B}{D}$ $h_{12} = \frac{\Delta^A}{D}$ $h_{21} = \frac{-1}{D}$ $h_{22} = \frac{C}{D}$
A	$A = \frac{(1+S_{11})(1-S_{22})+S_{12}S_{21}}{2S_{21}}$ $B = \frac{(1+S_{11})(1+S_{22})-S_{12}S_{21}}{2S_{21}}$ $C = \frac{(1-S_{11})(1-S_{22})-S_{12}S_{21}}{2S_{21}}$ $D = \frac{(1-S_{11})(1+S_{22})+S_{12}S_{21}}{2S_{21}}$	$A = \frac{Z_{11}}{Z_{21}}$ $B = \frac{\Delta^Z}{Z_{21}}$ $C = \frac{1}{Z_{21}}$ $D = \frac{Z_{22}}{Z_{21}}$	$A = \frac{-Y_{22}}{Y_{21}}$ $B = \frac{-1}{Y_{21}}$ $C = \frac{-\Delta^Y}{Y_{21}}$ $D = \frac{-Y_{11}}{Y_{21}}$	$A = \frac{-\Delta^h}{h_{21}}$ $B = \frac{-h_{11}}{h_{21}}$ $C = \frac{-h_{22}}{h_{21}}$ $D = \frac{-1}{h_{21}}$	$\begin{pmatrix} V_1 \\ I_1 \end{pmatrix} = \begin{bmatrix} A & B \\ C & D \end{bmatrix} \begin{pmatrix} V_2 \\ -I_2 \end{pmatrix}$

*: *G. D. VENDELIN, "Design of amplifiers and oscillators by the S-parameter method", John Wiley*

So, if S21 = S12 = S22 = 0, it comes that Y11 = (1 – S11) / (1 + S11)