

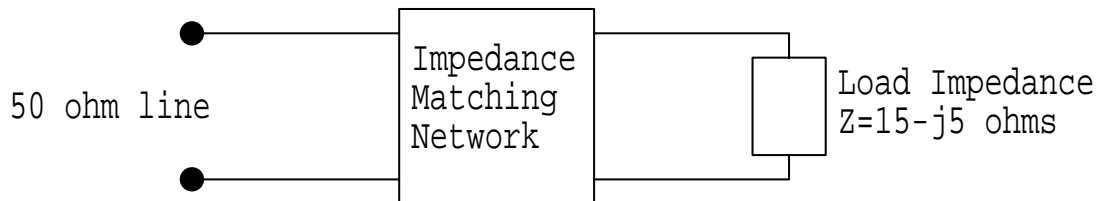
## EE431/531 Microwave Circuit Design I: Lab 2

### 1. Introduction

This lab concentrates on the principles of designing impedance matching networks using Smith charts and CAD tools. You will implement both ell and microstrip circuits to match a given load to a specific line impedance. In addition, you will gain experience with MDS techniques such as subcircuits, parameter sweeps, and optimization.

Although impedance matching networks may seem mundane, their proper design and implementation is critical to the efficient transfer of power along the signal path of an amplifier. If there is an impedance mismatch, electrical signals will undergo reflections at the boundary of the impedance discontinuity, which can cause a significant power loss.

### 2. Design Specifications



**Figure 1:** Block diagram of the matching network

Figure 1 shows the basic layout of the circuit you are to design. There is a load impedance  $Z_L$  that is to be matched to a line with characteristic impedance  $Z_0$ . The large block in the center of the figure will be replaced by your matching network (either ell or microstrip).

The initial values are  $Z_0 = 50 \Omega$ ,  $Z_L = 15 - j5 \Omega$ , and the design frequency is  $f = 1 \text{ GHz}$ .

From the equation  $\Gamma = (z - 1) / (z + 1)$ , where  $z = Z / Z_0$ , the equivalent reflection coefficient of the load impedance is  $\Gamma_L = 0.5423 \angle -167.5^\circ$ .

In the actual circuit, the  $50 \Omega$  line is represented by an S-port with a port impedance of  $R = 50 \text{ OH}$ . The  $15 - j5 \Omega$  load impedance is approximated by using a  $15 \Omega$  resistor and a  $31.83 \text{ pF}$  capacitor (which has a reactance of approximately  $-j5 \Omega$  at  $1 \text{ GHz}$ ).

You may be wondering what happens at frequencies other than the design value of  $1 \text{ GHz}$ . Of course, the capacitor's reactance will change as the frequency changes, so your matching network is unlikely to maintain a perfect impedance match at off-frequencies. However, this is not as unrealistic as you may think, because real transistor amplifiers have S-parameters that can vary widely with frequency, which greatly affects the performance of an attached impedance matching network.

### 3. Subcircuits and Miscellaneous Notes

#### 3.1 Subcircuit procedure

For simple circuits with few components, it is often best to construct the entire schematic on a single circuit page. As the schematic grows larger in size and complexity, it is more efficient to divide the single circuit into a set of smaller, modular subcircuits that are reusable. Each subcircuit can then be tested individually before being assembled into a master circuit.

The process of creating and using subcircuits is quite easy in MDS. A subcircuit is constructed in almost exactly the same manner as a standard circuit, except for a few additional details:

1. Construct the subcircuit on a new circuit page.
2. Identify and mark the input and output terminals of the subcircuit.
3. Create a symbol for the subcircuit. (Every circuit page has an associated symbol page, which defines the on-screen symbol that MDS uses to represent the subcircuit. Although a standard circuit also has a symbol page, the symbol is usually not defined. Subcircuits, however, require the symbol to be defined explicitly.)
4. Repeat steps 1 through 3 for each subcircuit.
5. Construct the master circuit on a new circuit page. Insert the symbol for each subcircuit into its appropriate position within the master circuit.

You will implement your matching networks (both ell and microstrip) as subcircuits. The master circuit consists of S-ports and the load impedance.

Although the circuits for this lab assignment could easily be built on a single circuit page, you are asked to use the subcircuit method for practice.

#### 3.2 Miscellaneous Notes

The notation developed in Lab 1 is used extensively here. Refer to the Lab 1 handout for more information.

## 4. Ell Matching Networks

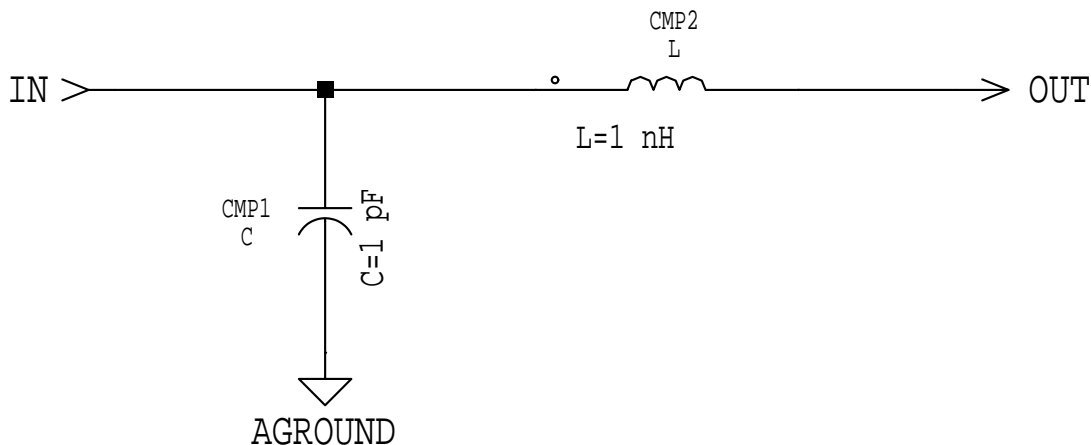
### 4.1 Assignment

You are to design two different ell circuits (i.e., inductors and capacitors in an “ell” configuration) to match the  $15 - j5 \Omega$  load to the  $50 \Omega$  line at an operating frequency of 1 GHz. You have to test both ell matching networks to examine their performance at various frequencies.

You may use any combination of inductors and capacitors that will produce a valid impedance match between the line and the load. The simplest design method is to use ZY-Smith chart paper for your initial computations. Since you will turn in your design charts, carefully annotate them with pertinent information.

### 4.2 Subcircuit construction

- Close all windows that may still be open from your last lab session. Open the MW\_Labs project file (or whatever your project file is called) by double-clicking its icon on the main MDS window.
- Create a new workbench called Lab2 within the MW\_Labs project file.
- Create another new workbench called ELL within the Lab2 workbench.
- Within the ELL workbench, create a new circuit page called MN\_LC1. This circuit is the first of your two ell matching networks and should be implemented as a subcircuit.



**Figure 2:** An ell matching network configured as a subcircuit

- Figure 2 is an example of what the ell subcircuit might look like (use your own configuration and component values). Notice that the input terminal (“IN”) and output terminal (“OUT”) of the subcircuit are clearly identified. The analog ground is a common node for both the input and output ports of the subcircuit.
- Build the core of your matching network using the inductor, capacitor, analog ground, and wire icons in the Components Palette. If the icons are not visible, click [More parts ...] on the Components Palette, scroll through the ‘Parts’ list window, and choose the “Lumped passive” option. If you wish, you can also directly insert these components by using the menu bar’s INSERT menu.

Next, click [More parts ...] and choose the “Connectors” option from the ‘Parts’ list. Select the Components Palette icon that has the floating label *Connector in*. Use the mouse to place the input marker at the input node of the matching network. You may have to rotate the marker to position it properly. In the blue box next to the input marker, type IN to identify the input terminal (you can type any text you want, but descriptive text is usually better).

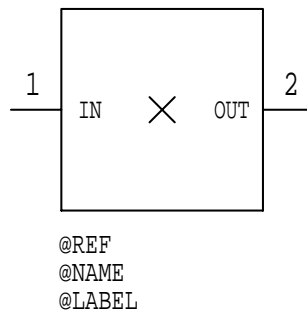
Select the Components Palette icon that has the floating label *Connector out*. Place the output marker at the output node of the matching network. Type OUT in the output marker’s blue box.

- Make sure ‘MN\_LC1: CIRCUIT PAGE 1’ is the active window. Choose [MB:WINDOW/CHANGE PAGE/SYMBOL 2]. The original circuit page is now replaced by its associated symbol page. Don’t worry, your schematic is safe and unaltered. The symbol page is initially blank except for a dot grid and a cross in the center of the page. This means that no symbol has been defined yet.
- Select [MB:PERFORM/CREATE SYMBOL]. This automatically creates a default symbol (a box with numbered and labeled input/output terminals) and places the symbol in the middle of the symbol page.

Select [MB:INSERT/REF-NAME-LABEL]. A block containing three labels tags along with the mouse pointer. Place the block underneath the box symbol and click the mouse button to drop the block onto the symbol page. MDS uses these labels to uniquely identify the subcircuit.

Figure 3 is an example symbol page for the subcircuit.

### Matching Network - LC1



**Figure 3:** The subcircuit’s associated symbol page

- The optional text above the box was put there by choosing [MB:INSERT/TEXT], clicking the mouse at the desired starting position on the symbol page, typing *Matching Network - LC1*, and then pressing the ESC key to finish input. You can change the color of the text by selecting the text with the mouse and then choosing [MB:SET/COLOR SELECTED/???], where ??? is one of the defined MDS colors (such as RED).

When you insert the subcircuit in the master circuit, the subcircuit will look exactly like the symbol defined on the symbol page. The default box is quite dull, so MDS includes a number of drawing tools you can use to create better looking symbols. For the time being, just leave the symbol as it is (unless you are feeling very artistic).

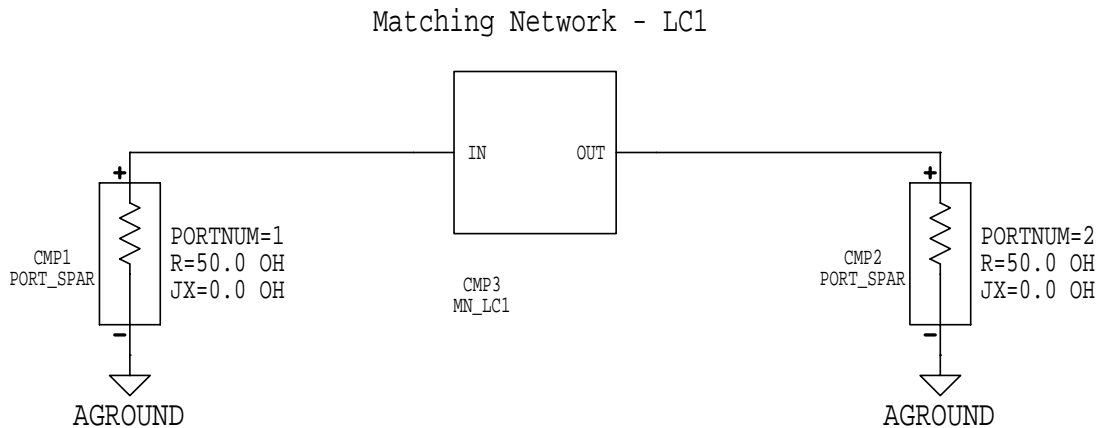
- Change back to the circuit page by choosing [MB:WINDOW/CHANGE PAGE/CIRCUIT 1]. Your subcircuit is now complete! Close the ‘MN\_LC1: CIRCUIT PAGE 1’ window.

### 4.3 “No load” test case

The load impedance is initially omitted and replaced by an S-port for  $S[2,2]$  measurements. You can then observe what the load would “see” as the impedance at the output node of the matching network.

#### 4.3.1 Test circuit construction

- Create a new circuit page called Test\_No\_Load1 within the ELL workbench.



**Figure 4:** Master circuit for testing the ell matching network subcircuit with no load impedance

- The Test\_No\_Load1 circuit you need to build looks like Figure 4.
- Place S-ports ( $R = 50.0 \text{ OH}$ ,  $JX = 0.0 \text{ OH}$ ) on the circuit page using the usual techniques.

Choose [MB:INSERT/COMPONENT/BY LABEL]. A dialog window pops up that asks you for the label of the design icon to be used as a component. What this is really asking for is the name of the subcircuit you want to place on the circuit page. Type MN\_LC1 in the input box and press RETURN (or click [OK]).

An outline of the box symbol for your subcircuit tags along with the mouse pointer. Position the symbol where you want it to go and click the mouse button to drop it onto the circuit page. You might have to rotate the symbol to get the proper orientation. MDS adds the name of your subcircuit to its [MB:INSERT/COMPONENT] menu for quick access.

Insert analog grounds and connect the various components with wire.

#### 4.3.2 Simulation

- Click [CIRCUIT:Simulation] to activate the ‘Simulation Setup’ dialog window.
- Set the simulation conditions to *Type*=“S-parameter”, *Sweep Type*=“Linear”, *Start*=800 MHz, *Stop*=1200 MHz, and *Step-size*=10 MHz.
- Change the default dataset name from DATASET to DS\_Test\_No\_Load1. You can do this by clicking [More] on the ‘Simulation Setup’ dialog window, clicking [Set host/dataset names ...], and entering the new name in the pop-up input box.
- Start the simulation. MDS will compute the S-parameters for your test circuit and ell subcircuit.

### 4.3.3 Output

- Create a new display page called Test\_No\_Load1. In the upper left corner of the display page, change the line that says *Dataset=DATASET* to *Dataset=DS\_Test\_No\_Load1*.
- Click [PRESENTATION:Plot] to activate the ‘Plot’ dialog window. Click [Select Data ...] to open the Dataset Browser. Choose “DS\_Test\_No\_Load1” as the dataset and “S[2,2]” as the output variable. Click [OK] to exit the browser. Set *Plot Format* to “Z-SMITH” in the ‘Plot’ window. Click [OK] to have MDS draw the plot.
- Click [PRESENTATION:Listing] to activate the ‘Listing’ dialog window. Open the Dataset Browser. Choose “DS\_Test\_No\_Load1” for the dataset and “freq (Independent 1)” from the *Variables* list. Click [OK] to exit. Change *Function* to “None” in the ‘Listing’ window. Click [OK] to have MDS output a column of frequencies on the same page as the Z-Smith plot.
- Repeat the listing procedure, but this time choose “S[2,2]” as the output variable. MDS will place the column of S[2,2] values next to the frequency column.
- The default data format for S[2,2] is to show real and imaginary parts, which is not very convenient. Click inside the frame that surrounds the column of S[2,2] values. This selects and highlights the column. Choose [MB:SET/LISTING COLUMN/DATA FORMAT/LIN MAG AND PHASE]. This changes the default data format to magnitude and phase (degrees).

### 4.3.4 Results to turn in

**Note:** Remember, you need to repeat the same construction, simulation, and output procedures for your second ell matching network. It is highly recommended that you give the second matching network its own, separate test circuit. Just follow this procedure:

Close the ‘Test\_No\_Load1: CIRCUIT PAGE 1’ window. In the ELL workbench, select the Test\_No\_Load1 circuit icon. Choose [MB:COPY/AS GROUP]. Click and drag the selected icon to another part of the workbench window to create a copy of the original test circuit. Rename the copied icon to Test\_No\_Load2. You can now open and edit the copied test circuit to use the second matching network. Change the dataset name to DS\_Test\_No\_Load2 before running the simulation. You can also copy the original display page and edit it to use the new dataset name.

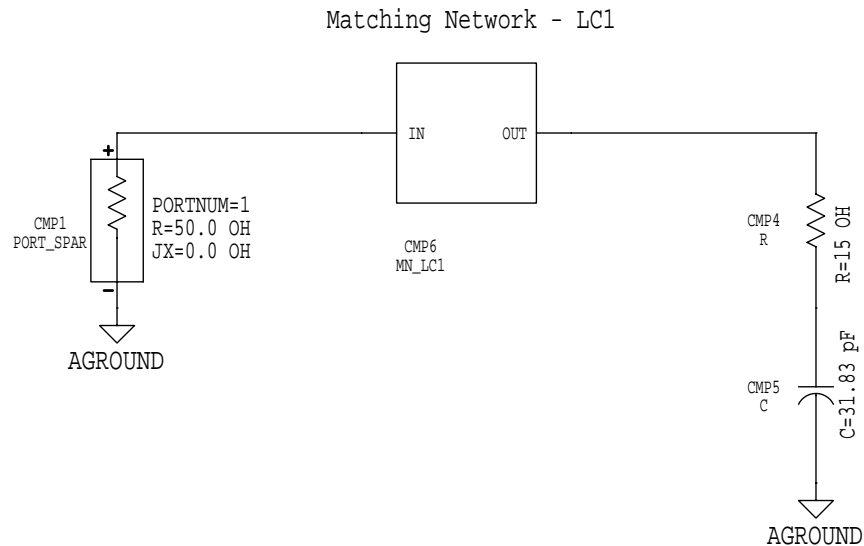
- Include the by-hand Smith charts that you used to design the two ell matching networks. Explain how you computed the required L and C component values.
- Turn in printouts of both ell subcircuits, as well as a printout of the “no load” master test circuit (either ell subcircuit is fine; just choose one).
- Turn in printouts of the combined Z-Smith plot and S[2,2] versus frequency listing for both ell matching networks.
- Briefly answer these questions:
  1. S[2,2] is the reflection coefficient at Port 2 when all other ports are properly terminated. Recall that the load impedance has an equivalent reflection coefficient of  $\Gamma_L = 0.5423 \angle -167.5^\circ$ . How well did your ell networks match the load impedance at the design frequency?
  2. As the frequency varies, how does S[2,2] change? How will this affect the impedance match at frequencies other than the design value?
  3. Do both of your ell networks perform equally as well, or does one do better than the other? Can you think of practical reasons to prefer one ell configuration over another?

#### 4.4 “With load” test case

For this case, the load impedance is placed at the output of the matching network. Only one S-port is used in the simulation. You will track  $S[1,1]$  to verify the effectiveness of your matching networks.

##### 4.4.1 Test circuit construction

- Close all open circuit pages and display pages. Make the ELL workbench the active window.
- Within the ELL workbench, create a new circuit page called Test\_With\_Load1. This will be a different master test circuit that includes the load impedance.



**Figure 5:** Master circuit for testing the ell matching network subcircuit with a load impedance

- The Test\_With\_Load1 circuit you need to build looks like Figure 5.
- Insert an S-port and the necessary load impedance components to build the new test circuit. Use the procedures from the previous section to insert your first ell subcircuit into the master circuit.

##### 4.4.2 Simulation

- Activate the ‘Simulation Setup’ dialog window. The simulation parameters are the same as for the “no load” master test circuit. Change the default dataset name to DS\_Test\_With\_Load1. Start the simulation.

##### 4.4.3 Output

- Create a new display page called Test\_With\_Load1. In the upper left corner of the display page, change the line that says *Dataset=DATASET* to *Dataset=DS\_Test\_With\_Load1*.
- Create a Z-Smith plot of  $S[1,1]$  on the new display page.
- Add two listing columns to the Z-Smith plot. One should be the frequency, while the other should be the  $S[1,1]$  values. If necessary, change the data format of the  $S[1,1]$  column to show linear magnitude and phase.

#### 4.4.4 Results to turn in

**Note:** Remember, you need to repeat the same construction, simulation, and output procedures for your second ell matching network.

- Turn in a printout of the “with load” master test circuit (either ell subcircuit is fine; just choose one).
- Turn in printouts of the combined Z-Smith plot and  $S[1,1]$  versus frequency listing for both ell matching networks.
- Briefly answer these questions:
  1.  $S[1,1]$  is the reflection coefficient at Port 1 when all other ports are properly terminated. For a perfect impedance match, the magnitude of  $S[1,1]$  should be zero. Using this performance measure, how effective are your ell networks in matching the load impedance to the  $50\ \Omega$  line at the design frequency?
  2. As the frequency varies, how does  $S[1,1]$  change?

## 5. Microstrip Matching Networks - Part I

### 5.1 Assignment

You are to design a microstrip circuit to match the  $15 - j5\ \Omega$  load to the  $50\ \Omega$  line at an operating frequency of 1 GHz. For the given load impedance, use balanced open-circuited stubs and a series transmission line.

The substrate is Duroid, with a relative permittivity of  $\epsilon_r = 2.23$  and a height  $h$  of 0.7874 mm. You may assume that the microstrip has zero effective thickness.

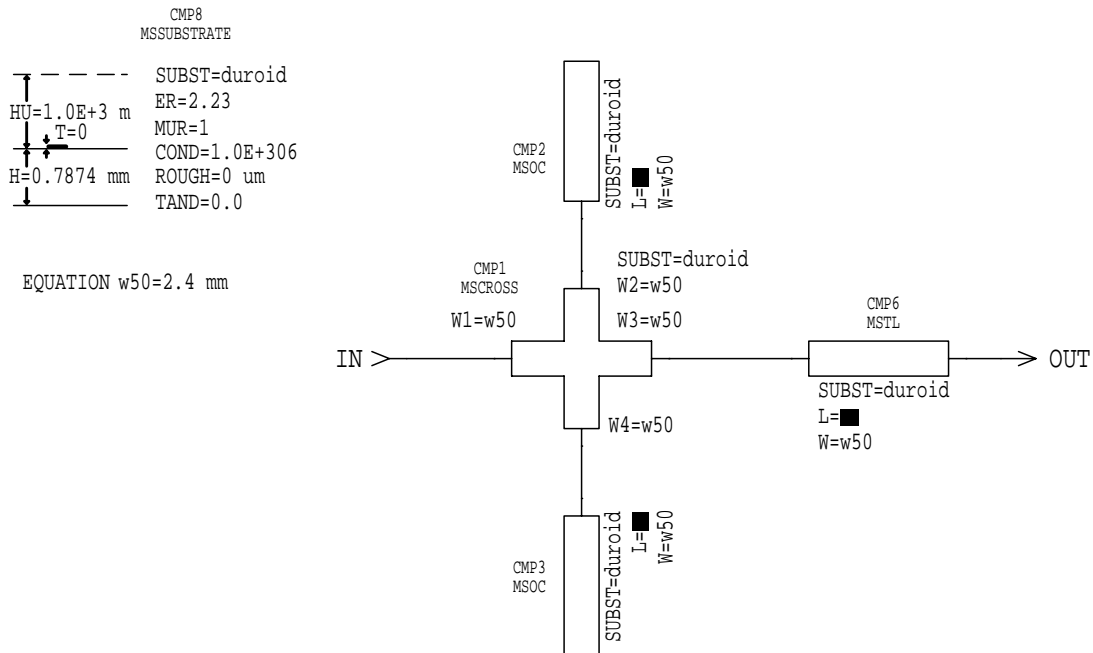
Use standard Smith chart paper for your initial computations. Since you will turn in your design chart, carefully label it to show how you derived your microstrip values. For consistency, you should express your physical line lengths in either millimeters or centimeters.

You need to manually compute the microstrip line width for a  $50\ \Omega$  characteristic impedance with a Duroid substrate. Later on you can use the built-in MDS microstrip calculator to confirm your work.

### 5.2 Subcircuit construction

- Close all open windows except for the Lab2 workbench.
- Create a new workbench called Microstrip within the Lab2 workbench.
- In the Microstrip workbench, create a new circuit page called MN\_Microstrip. This circuit page will contain your microstrip matching network and should be implemented as a subcircuit.





**Figure 6:** A microstrip matching network set up as a subcircuit

- Figure 6 is an example of what your microstrip network might look like (use your own design values). The input terminal (“IN”) and output terminal (“OUT”) of the subcircuit are identified and marked.

Notice how the open-circuited shunt stubs (MSOC - terminal on one end only) and transmission line (MSTL - terminals on both ends) attach to the central cross junction (MSCROSS). There is also a substrate definition block (MSSUBSTRATE) and an EQUATION statement on the circuit page.

- To access the microstrip components, click [More parts ...] on the Components Palette. Scroll down the ‘Parts’ list window and select the “Microstrip lines & bends” option. The Components Palette now has icons for substrate definition, transmission lines, and open-circuited stubs. Place these components on the circuit page.

To access the cross junction component, select “Microstrip misc.” from the ‘Parts’ window. The icon for the cross junction will now be available. You may need to rotate some of the microstrip components to get the proper orientation.

Use wire to connect the various components together. Note that in real life, the stubs and transmission line coming off the cross junction are physically joined to the cross as a single microstrip unit.

Remember to add the *Connector in* and *Connector out* markers to the input and output nodes of the subcircuit.

- Write down the component number (CMP#) of the transmission line (MSTL) component. You will need to refer to this number later in the lab.
- Edit the parameters in the substrate block to have the values show in Figure 6. Add an EQUATION statement to define the constant *w50*. This constant should be set to your computed value for the

width of a  $50\ \Omega$  microstrip line when the substrate is Duroid. For example, if the computed width is 2.4 mm, then use *EQUATION*  $w_{50}=2.4$  mm.

- You can use the built-in MDS microstrip calculator to automatically compute parameters like the physical line width. Once the substrate block has been defined, select any one of the microstrip components on the circuit page by clicking on it (the transmission line component is a good choice).

Choose [MB:TOOLS/LINE CALC]. The first time you call up the microstrip calculator, the ‘LineCalc Setup’ window pops up to allow you to change the model used for the calculations. The ‘MSTL’ model is the default choice, so just click [OK]. After a few seconds, the ‘LineCalc Microstrip Transmission Line’ dialog window pops up.

At the top of the dialog window is a scrollable message area. On the left half of the window are two panel areas. The upper one is for the geometry parameters ( $W$  and  $L$ ), while the lower one lists the current substrate parameters. On the window’s right half is a panel area that lists various electrical parameters of the microstrip line (including the characteristic impedance  $Z$ ).

The default unit for the line width is mils. If you prefer some other unit (like mm), edit the input box next to the  $W$  label. Change the unit from mils to your preference. You can ignore whatever value may be currently in the input box.

Look at the message area. It should be asking you to select a dependent parameter to be calculated. Click the button next to the  $W$  label. The message area should now be asking you to select an independent parameter. Edit the input box next to the  $Z$  label. Change the value there to the characteristic impedance you want (50 OH, for example).

Click [Calculate]. The input box next to the  $W$  label will now contain the computed microstrip line width. The value of  $Z$  will also be recomputed, so do not be surprised if the characteristic impedance value is slightly different from before. Does your manually computed line width agree with the MDS microstrip calculator?

When you are done, close the microstrip calculator window. If MDS asks if you want to update your microstrip component with the newly computed value, answer “No”.

- Each microstrip component has a *SUBST* parameter, which should be set to “duroid” since that is what is defined in the substrate definition block. The  $W$  parameter is the physical (not electrical) width of the microstrip line. Set all the  $W$  parameters (including  $W1$  through  $W4$ ) to “w50”, which you have previously defined in the *EQUATION* statement. The  $L$  parameter is the physical length of the microstrip line. Set the  $L$  parameters to your design values. Notice that the cross junction has no  $L$  parameter.
- Switch to the symbol page that is associated with the circuit page. Using the procedures outlined in the ell section, create a symbol for your microstrip subcircuit (remember to add the REF-NAME-LABEL block underneath the symbol). When you are done, switch back to the circuit page and close the circuit page window.

### 5.3 Test circuit construction

- Create a new circuit page called Test\_With\_Load within the Microstrip workbench.
- Construct a master test circuit that looks like Figure 5, except use your microstrip subcircuit instead of the ell subcircuit. When inserting the subcircuit via the [MB:INSERT/COMPONENT/BY LABEL] command, type MN\_Microstrip when you are asked to enter the label of the design icon to be used as a component.

- Write down the component number (*CMP#*) of the inserted microstrip subcircuit. You will need to refer to this number later in the lab.

#### 5.4 “No optimization” case

You will test the effectiveness of your original microstrip design prior to optimization.

##### 5.4.1 Simulation

- Activate the ‘Simulation Setup’ dialog window.
- Configure a standard S-parameter simulation with a linear frequency sweep from 800 to 1200 MHz in 10 MHz steps. Change the dataset name to DS\_Test\_No\_Opt before starting the simulation.

##### 5.4.2 Output

- Create a new display page called Test\_No\_Opt. In the upper left corner of the display page, change the line that says *Dataset=DATASET* to *Dataset=DS\_Test\_No\_Opt*.
- Create a Z-Smith plot of S[1,1] on the new display page.
- Add two listing columns to the Z-Smith plot. One should be the frequency, while the other should be the S[1,1] values (in linear magnitude and phase format).

##### 5.4.3 Results to turn in

- Include the by-hand Smith chart that you used to design the microstrip matching network. Explain how you computed the required microstrip component values.
- Turn in printouts of the microstrip subcircuit and the “with load” master test circuit.
- Turn in a printout of the combined Z-Smith plot and S[1,1] versus frequency listing for the microstrip subcircuit.
- Briefly answer these questions:
  1. How well did your microstrip matching network perform at the design frequency? How did the microstrip design compare to your ell matching networks?
  2. As the frequency varies, how does S[1,1] change?
  3. If each stub is made longer by an electrical length of  $\lambda/2$ , what would be the new physical length of each stub in your design? How would the longer stubs affect the value of S[1,1] at the design frequency? Compare the S[1,1] - frequency relationship between a circuit with the longer length stubs versus a circuit with the original length stubs.

#### 5.5 “Parameter sweep” case

You will instruct MDS to sweep the length  $L$  of the transmission line microstrip component to see if the impedance match can be improved. The lengths of the open-circuited stubs will be left alone for now.

The subcircuit and master test circuit are unchanged from the “No optimization” case, so you can re-use them. On the other hand, you have to make some minor alterations to the simulation setup for a parameter sweep. If you want to preserve the setup of the original master test circuit, make a copy of the test circuit first. You can then work on the copy without disturbing the original test circuit.

### 5.5.1 Simulation

- Close all open display page windows and make the master test circuit page the active window.
- Activate the ‘Simulation Setup’ dialog window.
- In the ‘Simulation Setup’ window, change the *Sweep Type* to “Single point”. The ‘Simulation Setup’ window should update itself to show a single input box next to the *Frequency* label. Edit the contents of the input box to set the value “1 GHz”. MDS will perform the simulation only at the design frequency.
- Click [Sweeps/control ...]. The ‘Sweeps and Control’ list window pops up. From the list of options shown, choose “Parameter sweep” and click [OK].
- The ‘Simulation Setup’ dialog window changes its appearance again. On the left side of the window is a new panel area labeled *Simulation sequence* that has a list of selectable options. The list’s first option (by default) is “SWEEP:sweep1”, which activates the parameter sweep setup. The other option is “SP:sim1”, which returns you to the standard ‘Simulation Setup’ dialog window.
- On the right side of the dialog window is a panel area for setting the parameter sweep conditions. Remember those component numbers (*CMP#*) you were told to write down? You will need those values now.

For example, if your microstrip subcircuit symbol is *CMP8* and your transmission line component is *CMP6*, then type `cmp8 . cmp6 . L` in the *Parameter name* input box. This tells MDS to 1) access component *CMP8* (the microstrip subcircuit) on the master test circuit page, 2) within the *CMP8* subcircuit, access the component *CMP6* (the transmission line), and 3) select the length *L* of *CMP6* as the parameter to sweep. Use the component numbers from your own circuit designs.

For future reference, if a circuit component has a parameter that can be edited, you can tell MDS to sweep the value of that parameter (as long as it makes sense, like the resistance of a resistor).

- You have to decide on the range of the parameter sweep. The limits and step-size of a parameter sweep are determined solely by the range of values and resolution you want to see.

The base unit of the parameter sweep is derived from the component value’s unit on the circuit page. For example, if the transmission line component has its length specified as  $L=1.0$  cm, then the base unit is meters and not centimeters. Likewise, if you specified the length as  $L=10$  mm instead, the base unit is still meters. (MDS prefers MKS units.)

Suppose  $L_0$  is the original length of the transmission line component. Arbitrarily let  $x$  be 25% smaller than  $L_0$ , and let  $y$  be 25% larger than  $L_0$ . For the example case of  $L=1.0$  cm, this means  $x$  is  $0.75$  cm =  $7.5$  mm =  $0.0075$  m, and  $y$  is  $1.25$  cm =  $12.5$  mm =  $0.0125$  m.

MDS requires that you express the limits and the step-size as numeric values that are modified by a multiplier of the base unit. Some of the pre-defined multipliers are “x1.0”, “milli”, and “micro”, which are used to multiply the numeric value by  $1$ ,  $10^{-3}$ , and  $10^{-6}$ , respectively.

Therefore, you can choose to express  $x$  as either “7.5 milli” or even “0.0075 x01”. Note that there is no built-in MDS multiplier for “centi”, so you cannot express  $x$  as “0.75 centi”.

Now is the time to configure the remaining sweep conditions. Set *Sweep type*=“Linear”. Next to the labels *Start*, *Stop*, and *Step-size* are input boxes for you to enter the required values. Adjacent to each input box is a button for selecting the value multiplier. Just click the button and select the appropriate multiplier option from the pop-up list.

With this information, set  $Start=x$  milli,  $Stop=y$  milli, and  $Step-size=100$  micro. Remember to replace the  $x$  and  $y$  limits with your own numbers (expressed in millimeters).

Under these conditions, MDS will perform a linear sweep of the transmission line's length  $L$ . It will start at  $x$  millimeters and stop at  $y$  millimeters. The step size will be 100 micrometers (0.1 mm).

- Click [More] and change the dataset name to DS\_Test\_Swp. Click [Start] to begin the simulation.

### 5.5.2 Output

- Create a new display page called Test\_Swp. In the upper left corner of the display page, change the line that says  $Dataset=DATASET$  to  $Dataset=DS\_Test\_Swp$ .
- You now need to display the results of your parameter sweep (which is  $S[1,1]$  versus  $L$ ).

Add a listing column of  $S[1,1]$  values (in linear magnitude and phase format) to the new display page.

Add another listing column next to the  $S[1,1]$  column. When in the Dataset Browser, choose "DS\_Test\_Swp" as the dataset and the "cmp8.cmp6.1 (Independent 1)" option as the output variable. Your component numbers may be different. Click [OK] to exit the browser. In the 'Listing' dialog window, set  $Function="None"$ . Click [OK].

Notice that the listing column for the swept  $L$  parameter has units of meters. If there are not enough digits displayed for the precision you want, select the  $L$  column by clicking inside its frame. Then choose [MB:SET/LISTING COLUMN/DIGITS/4] to increase the number of digits to four (you can pick any number of digits you like).

Examine  $S[1,1]$  as a function of  $L$ . Is there a value of  $L$  that makes the magnitude of  $S[1,1]$  closer to zero than your original transmission line length? If necessary, re-run the simulation with a wider sweep range to catch the minimum  $S[1,1]$  value.

### 5.5.3 Results to turn in

- Turn in a printout of the  $S[1,1]$  versus transmission line length  $L$  listing for the microstrip subcircuit.
- Briefly answer these questions:
  1. Were you able to "tune" your microstrip circuit for better impedance matching performance? At the design frequency, how large was the magnitude of  $S[1,1]$  at the optimum length  $L$ ? Compare this to the value you got from the "No optimization" case.
  2. How would you go about optimizing other parts of your microstrip matching network using the parameter sweep method?

## 5.6 "Full optimization" case

As you may have noticed, parameter sweeps are a powerful but tedious method for performing circuit optimizations. In this section of the lab, you will let MDS do almost all the work to fully optimize your microstrip matching network.

You need to make some fairly radical changes to the simulation setup for an optimization run, and you also have to edit the subcircuit. If you want to preserve your original work, make copies of both the subcircuit and master test circuit first.

### 5.6.1 Subcircuit changes

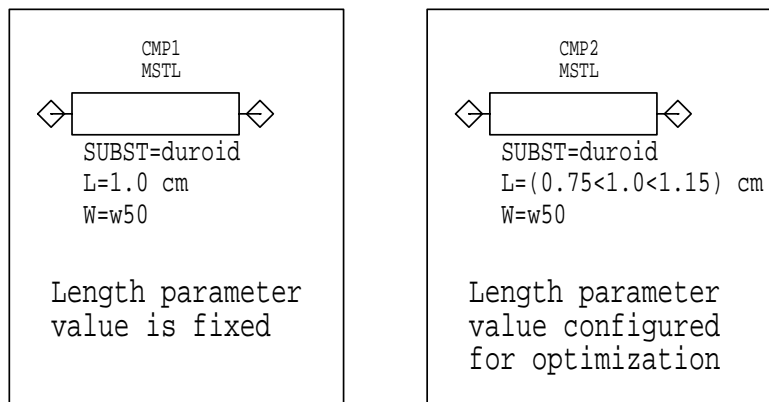
- Before MDS can optimize your circuit, you first have to specify the component parameters in your design that should be optimized. You then edit the parameter values of those specified components to conform to a special optimization format. This format places lower and upper limits on the allowed range of the component's parameter value. For parameter values that you do not want optimized, you can just leave them alone.

After all the component parameters that should be optimized have been properly edited, you next enter the goals (using a special goal editor) that MDS should achieve during the optimization. Once the simulation begins, MDS will try to meet the stated goals by adjusting the specified parameter values within the limits you imposed.

- You can assume that the microstrip line width is a fixed value that depends on the characteristic impedance and the substrate. Hence, MDS only needs to optimize the line length of each microstrip component.
- Suppose one of your microstrip components has a length  $L_o$ . Let  $x$  be 25% smaller than  $L_o$ , and let  $y$  be 15% larger than  $L_o$ . The  $x$  and  $y$  values are the lower and upper limits that constrain MDS during the optimization. Their exact values are arbitrary and should be set to the numbers that are most appropriate for your simulation. The wider the range, the more values MDS can try during the optimization. Be careful in choosing your limits, since they can adversely affect the optimization time and the accuracy of the results. Note that each component parameter may have its own set of limits that differ from another parameter's limits.

Change the  $L=L_o$  parameter of the component to  $L=(x < L_o < y)$ . As a concrete example, if one of your microstrip components has a length parameter  $L=1.0$  cm, change the parameter to  $L=(0.75 < 1.0 < 1.15)$  cm. The initial guess for the optimal solution will be the center value of 1.0 cm. As the simulation progresses, MDS will adjust the initial guess between 0.75 cm and 1.15 cm. Note that the number of digits you use is significant. For instance, a center value of 1.00 cm is a more precise initial guess than 1.0 cm.

Refer to Figure 7 for an example of what the special optimization format looks like for a real component.



**Figure 7:** Editing the parameter value for optimization

- Open the circuit page that contains your microstrip subcircuit. Edit the  $L$  parameters for each of your microstrip components (the two shunt stubs and the transmission line). When you are done adding the limit values, close the subcircuit window.

## 5.6.2 Simulation

- Close any display page window that might still be open and make the master test circuit page the active window.
- Activate the ‘Simulation Setup’ dialog window.
- In the ‘Simulation Setup’ window, set *Sweep type* to “Single point” and *Frequency* to “1 GHz”. Actually, you could request a linear frequency sweep, and MDS would try to optimize your design over the entire frequency range. To keep things simple, just perform the simulation at the design frequency.
- Click [Sweeps/control ...]. The ‘Sweeps and Control’ list window pops up. From the list of options shown, choose “Optimization” and click [OK].
- The ‘Simulation Setup’ dialog window now changes its appearance. On the left side of the window is the *Simulation sequence* list. The list’s first option (by default) is “OPTIM:control1”, which activates the optimization setup. The second option is “GOAL:goal1”, which loads the goal editor. The last option is “SP:sim1”, which returns you to the standard ‘Simulation Setup’ dialog window.
- On the right side of the dialog window is a panel area for setting the optimization conditions. Set *Type* to “Optimization”, *Optimization Algorithm* to “Gradient”, and *Num. of gradient iters.* to “50”.

There are actually three types of algorithms you can choose from. “Gradient” performs a gradient descent search, which is best when your component values are nearly optimal already. It may, however, get stuck in a local minimum and miss better possibilities. The “Random” option uses random search techniques, which is appropriate when your initial values are not very exact. The problem is the random search may leap over a good minimum that the gradient approach would catch. The final optimization algorithm is “Hybrid”, which combines the best attributes of the other two algorithms. Unfortunately, the hybrid approach can be highly time intensive.

- In the *Simulation sequence* list, select the “GOAL:goal1” option to call up the goal editor.
- The ‘Simulation Setup’ dialog window changes its appearance again. The *Simulation sequence* list is still on the left half of the dialog window. On the right half of the window is the goal editor.

In the input box next to the *Expression* label, type `mag(s11)`. This tells MDS to track the magnitude of  $S[1,1]$  as the goal. Next, you need to impose constraints on the goal by defining the minimum (*Min*) and maximum (*Max*) allowed goal values.

Next to the labels *Min* and *Max* are input boxes for you to enter the allowed goal values. Adjacent to each input box is a button for selecting the value multiplier (exactly like a parameter sweep). Set *Min* to “0.0” and *Max* to “0.005”. Use a multiplier of “x1.0”. Finally, set *Weight* to “1.0”.

With the stated goal parameters, MDS will try to optimize the microstrip component lengths so that the magnitude of  $S[1,1]$  is between 0.0 and 0.005. The weight parameter controls relative weighting for a multiple-goal optimization and can be ignored for this single-goal case.

You can change the *Min* and *Max* parameters to any values you want, but be sure what you choose is realizable and reasonable. If the constraints are too tight, the optimization process might fail. If the constraints are too loose, MDS will do a poor optimization job.

For a simple impedance matching network, only one goal is needed. For bigger projects (such as a multi-stage amplifier), you can define multiple, simultaneous optimization goals.

- Click [More] and change the dataset name to DS\_Test\_Opt. Click [Start] to begin the simulation.

- If you want to monitor the progress of the optimization, choose [MB:PERFORM/MNS/SHOW PROGRESS]. If for some reason the optimization process is taking too long, you can abort the simulation with [MB:PERFORM/MNS/ABORT SIMULATION]. You can also use [MB:PERFORM/MNS/SHOW ERRORS] to get details on error messages related to the simulation. These commands are very useful and work with most types of simulations (not just optimization).
- After the simulation is done, the ‘Messages’ window tells you the *Cost factor* of the optimization, which is a measure of the error between the stated goals and the actual results. Lower cost factors are generally better.
- To see the optimal values determined by MDS, choose [MB:PERFORM/MNS/SHOW BEST VALUES]. The ‘Completed optimizations’ window appears, which contains a list of the simulations that have been performed recently. Click the button next to the simulation you want and then click [OK]. The ‘Messages’ window will show the optimized values of your component parameters.
- Try the [MB:PERFORM/MNS/UPDATE DESIGNS] command. The ‘Completed optimizations’ window appears again, so click the simulation you want and then click [OK]. The ‘Messages’ window will inform you that backannotation is completed.

What does this command do for you? Well, MDS takes the optimized values and inserts them into your circuit page as the new center values in the edited component parameters. Why is this useful? If you change the simulation mode back to normal (by clicking [Sweeps/control ...] on the ‘Simulation Setup’ dialog window and choosing the “None” option), the center value of the range is automatically used by MDS as the default component value. You do not have to change the special optimization format back to the original fixed format to perform standard simulations. Note that the number of digits you used for the original center value will determine the number of digits that MDS uses when it updates the center value with the optimized value.

### 5.6.3 Output

- Create a new display page called Test\_Opt. Be sure to set *Dataset=DS\_Test\_Opt*.
- Add an S[1,1] listing column (in linear magnitude and phase format) to the new display page. You should have a single S[1,1] value in the column (for  $f = 1$  GHz).

Add a new listing column to display the optimal length values of your microstrip components. In the Dataset Browser, choose “DS\_Test\_Opt” as the dataset, “Control1.OPT” as the package, and “mn\_microstrip.cmp2.l” as the output variable. Your *CMP#* may be different. Click [OK] to exit the browser. In the ‘Listing’ dialog window, set *Function=“None”*. Click [OK]. MDS now outputs the optimal *L* value for microstrip component *CMP2*.

Add new listing columns for the remaining microstrip components. When you are done, you should have the optimal lengths for both open-circuited shunt stubs and the transmission line.

- If you are not satisfied with the optimization results, change the component parameter limits or adjust the goal constraints and then re-run the simulation.

### 5.6.4 Results to turn in

- Turn in a printout of the S[1,1] value and optimal microstrip component lengths.
- Briefly answer these questions:
  1. How does your optimized S[1,1] value compare to the original unoptimized value?
  2. What is the percentage difference between your original and optimized component lengths?



## 6. Microstrip Matching Networks - Part II

### 6.1 Assignment

Design a new microstrip circuit to match the  $15 - j5 \Omega$  load to the  $50 \Omega$  line at an operating frequency of 1 GHz. This time, use a quarter-wave transformer and either 1) balanced open-circuited shunt stubs of length  $3\lambda/8$ , or 2) balanced short-circuited shunt stubs of length  $\lambda/8$ . Assume the substrate is Duroid, with the same parameter values as in Part I.

Choose one of the stub configurations and compute the physical microstrip line widths and lengths. This would be a good opportunity to use the MDS microstrip calculator to simplify your work. Just be sure that the MDS calculator is using the correct substrate values.

This is a paper design only. You do not have to construct and simulate your design using MDS, although you are certainly invited to do so if you want to experimentally verify your calculations.

### 6.2 Results to turn in

- Turn in all Smith charts (if any) that you use to design the matching network. Explain how you computed the necessary microstrip line widths and lengths.
- Sketch what your completed matching network would look like and label the important parts (quarter-wave transformer, shunt stub, etc.). Indicate the electrical length, physical length, and characteristic impedance of each microstrip component.
- If you decide to perform circuit simulations in MDS, turn in your circuit schematics and plot results.