

# EE431/531 Microwave Circuit Design I: Lab 1

## 1. Introduction

This lab will focus on the basic procedures for constructing and testing microwave circuits via the MDS simulation program. You will learn how to enter a circuit schematic into MDS, how to simulate the circuit at microwave frequencies, and how to extract and present the results from the simulation.

**Note:** Don't be discouraged by the length of this first lab assignment. Most of the text is a detailed list of instructions for accomplishing fundamental tasks. You are not expected to do any original circuit design at this point. Since you will use the same procedures over and over again in future labs, mastering the basic techniques now will make working in MDS much easier for you later.

## 2. An MDS Primer

The HP Microwave Design System (MDS) is a powerful CAD tool for designing and simulating high frequency electronic circuits. MDS often has several ways to perform a given task, and sometimes it can be confusing trying to remember which command to use. Don't worry, though. With experience you will soon learn the best ways to lay out and test circuits in MDS.

**Note:** For the rest of this lab, the text that you should type will be highlighted in the Courier font.

### 2.1 How to run MDS

First log into your user account on one of the EE department's Sun workstations. You can use the class account assigned to you, in which case MDS comes preconfigured. If you would rather use your own EE account, you can manually set up the MDS program by typing `addpkg` from the UNIX command line.

If you intend to run MDS directly on your local workstation, then type `mds &` at the UNIX prompt. It will take several minutes for MDS to load completely.

If you want to log into your local workstation but actually run MDS on a different host computer, then there are some preliminary steps you need to take:

- Identify the display type of the local workstation you are physically logged into. Look on the front of the computer chassis or on the monitor for a label that tells you the computer's display name. For example, if you are sitting at the workstation "jafar", its display name will be `jafar.ee.pdx.edu:0.0`.
- Suppose you want to run MDS on the host computer "flotsam". Perform a remote login to flotsam by typing `rlogin flotsam`.
- Once logged into the host computer, you need to set the `DISPLAY` environment variable to the display name of your local workstation. For example, if your local computer is jafar, you need to type `setenv DISPLAY jafar.ee.pdx.edu:0.0` at the UNIX prompt.
- Now you can run MDS on the host by typing `mds &`.

Why would you want to execute MDS on a different machine anyway? Well, sometimes the computer you are physically logged into may be performing very slowly (perhaps caused by multiple users who are using too many CPU resources), so it is more efficient to try a different workstation. You can type `rup` at the UNIX prompt to check the resource loads for various workstations. High numbers mean that a particular computer is heavily loaded, while low numbers mean less of a load and better performance.

## **2.2 MDS file system**

The basic MDS objects are circuit pages (which hold circuit schematics), dataset pages (which contain simulation data), and display pages (which contain plots and listings). Related pages can be grouped together to form an object called a workbench. For example, each lab assignment for this course could be given its own workbench to keep them separate. You can also create workbenches within workbenches to further organize your work. At the highest level is the project file, which groups related workbenches together into a single binary file that is stored in your UNIX directory.

The hierarchical structure of projects, workbenches, and pages is very similar to a hard drive's directories, subdirectories, and files. You can arrange your lab assignments any way you wish, but logical groupings make them easier to navigate.

Each page, workbench, and project file has an associated icon with a unique name. You open a project file or workbench by double-clicking its icon. This causes a window to pop up on the screen, which may be empty or contain other icons. Opening a circuit icon will pop up a circuit page window for schematic editing. When a display icon is opened, a display page window appears.

**Tip:** You can rename an icon by clicking the icon's name label, deleting the old name (by overtyping what is already there or by using the BACKSPACE key), and typing in the new icon name.

MDS saves a number of extra files in your UNIX directory. The one called "screen.dcs" has information about the arrangement of icons and windows on the main MDS window. The "wastebasket" file contains deleted material from your MDS runs.

**Tip:** If you keep all of your work in a single project file, make a backup copy in case you accidentally lose the original. Use the `cp` command from the UNIX command line to copy the project file.

## **2.3 Mouse and window controls**

Like most CAD programs, MDS makes extensive use of a graphical user interface with mouse control. Much of your experience with other windowing environments can be applied here.

### **2.3.1 Mouse operations**

Click once on an object with the left mouse button to select the object. MDS highlights the selected object by either changing its color or by drawing an outline around it. Click the same object again with the left mouse button to unselect the object.

Rapidly click an object twice in a row with the left mouse button (double-clicking) to open the object. To drag a selected object, press and hold down the left mouse button while moving the mouse pointer to a new location. Release the mouse button to "drop" the object at the new location.

Multiple objects can be selected by left-clicking objects in succession, or by dragging a selection box around the objects to be selected.

Clicking on certain objects with the right mouse button can cause a pop-up menu to appear. This pop-up menu will contain commands that can be applied directly to the object.

MDS has two ways of applying a command to an object. You can either select the object first and then activate the command, or you can choose the command first and then select the object. This is not always consistent within MDS, and sometimes the effect is not what you may have intended.

### 2.3.2 Windows

You can usually move a window around the screen by dragging the window's title bar to a new screen position. Some windows have resize buttons (they look like triangles) in each corner of the window. By dragging the resize buttons, you can make the window as big or as small as you need. If the window is too narrow, the window title might not be visible. Just resize the window to see the whole title.

In the upper right corner of most windows, there are two special buttons. Clicking the "tiny square" button will close the window. Clicking the "large square" button expands the window to full size. Dialog windows might only have the close button. A window could also have a vertical column of buttons on the left-hand border of the window, which allows quick access to commonly used commands.

A vertical scroll bar appears on the right side of the window and a horizontal scroll bar appears on the bottom of the window. There are buttons on the ends of each scroll bar that you can click and drag to control zooming. The zoom buttons are particularly useful if you want to see more of your overall schematic or plot in the window at the expense of some detail (or vice versa).

While any window can show information, only the active window can accept user input. The active window will always have a light blue border around the entire window, while inactive windows have a brownish border. You can make a window active by clicking on the window with the mouse.

## 2.4 Main MDS window

The main MDS window has a blue background with a menu bar running along the top, a palette of command buttons on the left side, and several rows of icons along the bottom.

### 2.4.1 Menus

When you move the mouse pointer over the menu bar area, drop-down menus will automatically appear with various program functions. Some of the commands on the drop-down menu will cascade to submenus with more commands. Menus in MDS are context sensitive, which means that the commands available on a given menu may change depending on the active window.

Many of the menu options are also accessible by other means, such as through pop-up menus and palettes. You don't have to understand what every menu option does right now; just become familiar with the basic commands and where they are located.

### 2.4.2 Palette

Palettes contain command buttons that you can click with the mouse to control various program functions. These palettes "float" over the main MDS window and remain visible unless covered by another window.

The SUI (Simplified User Interface) Palette on the left side of the main MDS window is particularly important. It has buttons for quickly accessing design, plotting, and workbench and file creation functions. The following is a short summary of the most commonly used buttons:

- Design* - Create a new circuit page for editing circuit schematics
- Layout* - Generate a device layout from the circuit schematic
- Presentation* - Produce a new display page for plots and listings from simulation data

- Workbench* - Create a new workbench icon in the active window (if possible)
- File* - Create a new project file icon in the active window (if possible)
- Save* - Save the contents of the active window
- Finish* - Shut down MDS and exit

### 2.4.3 Icons

The labeled icons at the bottom of the main MDS window represent various project files and component library files. You can examine the contents of these files by double-clicking the icons. Your own project files will also show up on the main window as icons.

## 2.5 *Miscellaneous*

- All the work you perform during an MDS session is kept in a temporary file. When you finally exit MDS, a dialog window will appear that asks you to select which objects you want to save. The default is to save everything you did during the session. You can also choose not to save certain objects, so any changes you made to those objects during the session will not be preserved.
- MDS is a complicated piece of software that has its share of bugs, some of which will crash the program and cause you to lose all unsaved work. Be sure to save your work at regular time intervals.
- MDS has a powerful array of tools for designing and simulating circuits. Experiment with the various features of the program. You may find new ways of doing things that can simplify your work or make it more productive and informative.

## 2.6 *Getting help*

- Many of the dialog windows in MDS have a help button that will give you some additional information about the available commands. However, the help message is often terse, so you may want to access the full online documentation via the HELP command on the menu bar.

The menu bar HELP command launches a full featured viewer that accesses the complete MDS documentation files. Unfortunately, it takes a few minutes to load, and the keyword searching feature is somewhat cumbersome.

## 3. A Few Sample Circuits

You will now have the opportunity to construct some basic circuits using MDS. The sample circuits are not very useful, but they do illustrate the general principles of circuit manipulation, simulation, and data extraction. The following notation and procedures will be used for this and future labs:

- Unless otherwise indicated, if you are told to click something, use the left mouse button.
- If you are asked to choose a command from the menu bar, the notation will be:

[MB:Menu/Command]

where “MB” means the menu bar, “Menu” is the name of the particular menu to choose from, and “Command” is the name of the specific command within the menu to activate.

For example, [MB:PERFORM/PLOT] means to use the mouse to select the PLOT command from the PERFORM menu on the menu bar. [MB:PERFORM/LINE UP/ALPHABETIC] means select the ALPHABETIC command in the cascading submenu LINE UP under the PERFORM menu on the menu bar. In the case of [MB:MOVE], just select the MOVE menu, which also happens to be a command by itself.

- If you are asked to click a command button on a palette, the notation is:

[PALNAME:Button]

where “PALNAME” is the palette’s name and “Button” is the name of the button to click. For example, the notation [SUI:File] means to click the File button on the SUI Palette.

- If you are asked to click any other type of button, the notation is:

[Button]

where “Button” is the name of the button to click with the mouse. For instance, if you see [OK], then click the button labeled OK with the mouse.

- Titles of windows are in single quote marks, e.g. ‘MW\_Labs: FILE’
- Names of files, workbenches, pages, icons, and objects are in the Helvetica font, e.g. amp\_L\_chart
- Parameter labels and miscellaneous names are in italics, e.g. *Start*
- Isolated parameter values and miscellaneous values that you can choose or set are in quotation marks, e.g. “1.5 GHz”

If the parameter value is in close proximity to the parameter label, the quotation marks are optional if it improves readability, e.g. set *Start* to “1.5 GHz” -or- set *Start* to 1.5 GHz -or- *Start*=1.5 GHz

- The emphasis methods mentioned above are not necessarily absolute. You may see emphasis marks that look inappropriate. However, you can usually determine the proper meaning from the context.
- If you are asked to type something in an input box, you can finish the input by either pressing the RETURN (or ENTER) key or by clicking the OK button. You can fix mistakes by using the BACKSPACE key. Some terminals have cursor arrow keys to move the text cursor.
- Many items in MDS have parameter values that can be changed. For example, a resistor component has a single parameter (its resistance), while a data plot may have several parameters.

Although the parameters of most circuit components are visible on the screen and can be directly edited with the mouse, some parameter changes require you to access a menu or dialog window.

Parameters have a descriptive label (e.g. *PORTNUM*=) that is immediately followed by the value of the parameter. If the parameter value is empty, you will see a blue box that acts as a placeholder for the value. To change a parameter’s current value, click the value itself (or the blue box if the value is empty), and type in the new value.

You can use the BACKSPACE key to remove the original value or to fix mistakes. If the terminal is properly set up, the cursor arrow keys can move the text cursor forward and backward.

### 3.1 Sample #1: An RLC Filter

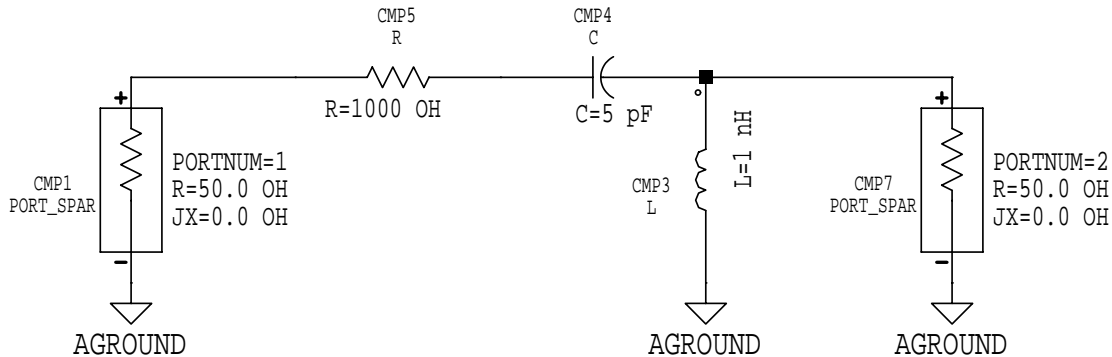


Figure 1: A simple RLC filter

#### 3.1.1 How to construct the circuit

- Click [SUI:File] to create a new project file. Type MW\_Labs in the input box of the 'New File' dialog window. When you are done typing, either press RETURN or else click [OK]. The 'MW\_Labs: FILE' window appears.
- Click [SUI:Workbench] to create a new workbench. Type Lab1 in the input box of the 'New Workbench' dialog window. The 'Lab1: WORKBENCH' window appears.  
**Tip:** You can use [MB:PERFORM/LINE UP/ALPHABETIC] to straighten out the icon display in the active window. You can do this at any time.
- Create another workbench within the 'Lab1: WORKBENCH' window. Type RLC for the name of the new workbench. The 'RLC: WORKBENCH' window appears.
- Click [SUI:Design] to create a new circuit design. The 'New Design' dialog window pops up. Type RLC for the name of the design. The 'RLC: CIRCUIT PAGE 1' window appears.

Notice that a Circuit Palette pops up on the left side of the main MDS window (this happens whenever the circuit page window is active).

The Circuit Palette contains three command buttons:

*Components* - Calls up the Components Palette to select circuit elements

*Simulation* - Calls up the simulation dialog window to adjust and start the simulation

*Presentation* - Calls up the Presentation Palette to generate data plots and listings

- Click [CIRCUIT:Components] to activate the Components Palette. Wait a few seconds for MDS to build the palette. Close the 'Messages' window by clicking the close button in the upper right corner of the window.

Move the mouse pointer over to the icons on the Components Palette. Notice how each icon has a label that "floats" above it to identify the icon's purpose. For example, the S-port component (the rectangle with the resistor symbol inside and polarity markings) has the floating label *S param port impedance*.

Select the S-port component by clicking its icon on the palette. Move the mouse pointer over to the circuit page window. Notice that an outline of the selected component tags along with the pointer. Position the pointer where you want the component to go and click to drop it onto the circuit page. You can then move the mouse pointer to a new location and click again to insert the second S-port.

Next, either choose [MB:SELECT/UNSELECT ALL] or else click [US] on the vertical column of buttons on the left border of the circuit page window. This unselects the S-port icon and turns the mouse pointer back to normal.

The default S-port impedance is 50 ohms ( $R=50.0$  OH and  $JX=0.0$  OH). For the first S-port, change the port number parameter value to  $PORTNUM=1$ . Edit the second S-port to be  $PORTNUM=2$ . Click the analog ground icon on the Components Palette and ground the negative end of both S-ports.

In case you are interested, the S-port is a special signal source that is used to perform S-parameter measurements at high frequencies. In designing microwave amplifiers, knowing how the transistor's S-parameters vary with frequency is critical.

- Click [More parts ...] on the Components Palette. The 'Parts' list window pops up. Scroll down the list and click on the "Lumped passive" option. The Components Palette changes to a new set of icons. Close the 'Parts' list window. Place the necessary resistor, inductor, and capacitor components onto the circuit page.

Notice that each component on the circuit page has a  $CMP\#$  label that uniquely identifies it. Don't worry if the  $CMP$  numbers on your design do not match the ones in Figure 1. The exact numbers depend on the order you placed the component on the page, so it will vary from person to person.

You can change the default component values to the ones shown in Figure 1 now or wait until later.

- You will probably still need to move the components to their final positions. You can select the component first, choose [MB:MOVE], and then click and drag the component to a new position on the circuit page. You can also do [MB:MOVE first], and then click and drag the component.

You can rotate a selected component by using the [MB:PERFORM/ROTATE] command (or you can use the rotation/flip icons at the top of the Components Palette). You can delete a component by using [MB:DELETE]. [MB:UNDO] allows you to undo your last move, copy, or delete.

If you right-click on a component, a pop-up menu may appear with the commands that can be applied to the component. Unfortunately, MDS is a bit mouse sensitive, so it is sometimes difficult to choose a move or delete command from the pop-up menu.

- You can connect the various components together by inserting wire between them. To activate the wiring mode, you can try any one these techniques: 1) Click the wire icon on the Components Palette, or 2) Click [IW] on the circuit page window's vertical column of buttons, or 3) Choose [MB:INSERT/WIRE].

To actually insert the wire, click the starting node, drag the mouse pointer to the ending node, and release the mouse button.

The wire routing feature of MDS can sometimes make really strange wire paths. If this occurs, select the misplaced wire by clicking on it with the mouse and then delete the wire with [MB:DELETE]. You can also activate the DELETE command first and then select the wires to erase. In any case, you can insert new wire after the old wire is gone.

- Close the Components Palette and make sure the circuit page is the active window.

### 3.1.2 How to simulate the circuit

- Click [CIRCUIT:Simulation]. The ‘Simulation Setup’ dialog window pops up. The default simulation *Type* is “S-parameter”, and the default *Sweep Type* is “Linear”.

You need to type in new values for the frequency sweep. Set *Start* to 1.0 MHz, *Stop* to 5.0 GHz, and *Step-size* to 25 MHz. This sweeps the S-port frequency from 1 MHz to 5 GHz in 25 MHz steps.

- Click [Simulation Parameters] on the ‘Simulation Setup’ window. The ‘Simulation Parameters’ window pops up. Select both the “S-parameter” and “Y-parameter” options by clicking on their buttons. This tells MDS to compute S- and y-parameters during the simulation. Click [OK] to exit the ‘Simulation Parameters’ window.
- Click [Start] on the ‘Simulation Setup’ window to begin the actual simulation. A ‘Messages’ window pops up to inform you of the progress of the simulation.

If errors are reported, fix your circuit schematic and restart the simulation. If the run was successful, then close the ‘Messages’ window. MDS creates a new dataset icon called DATASET in the ‘RLC: WORKBENCH’ window. This dataset page contains the output data from the simulation. At this point, you can close the ‘Simulation Setup’ window.

### 3.1.3 How to plot the output data

- Click [CIRCUIT:Presentation]. Type `RLC_Plot` in the input box of the ‘Presentation’ dialog window. The ‘RLC\_Plot: DISPLAY 1’ window appears. Notice that a new Presentation Palette pops up on the left side of the main MDS window.
- Click [PRESENTATION:Plot] to activate the ‘Plot’ dialog window. Click [Select Data ...] to call up the ‘Dataset Browser’ dialog window. Scroll through the *Variables* list and click on the “Y[2,1]” option. The y-axis of the output plot will now be the admittance Y[2,1]. The default x-axis is the S-port frequency. Click [OK] to exit the Dataset Browser.
- In the ‘Plot’ dialog window, check that the *Function* is “dB”. Next to the *Plot Format* label, click the [LIN vs LIN] button to get a pop-up menu of plot types. Choose the “LIN vs LOG” option for a linear y versus logarithmic x plot. Click [OK] to exit the ‘Plot’ window.

MDS now draws the plot on the display page. Examine the resulting plot carefully. Experiment with the display page window’s resize buttons. Try the horizontal and vertical scroll bars. Click and drag the zoom buttons on the ends of each scroll bar to zoom into or out of the display page.

- You can easily generate a printout (laser printer output) of images from your simulation. Choose the window you want to print (either the circuit page or the display page) and make it the active window.

Now use [MB:PERFORM/PLOT] to call up the ‘SELECT A PLOTTER’ dialog window. Click the button for the appropriate printer (lw4 - lw10, whichever one you have a paid account on). Select the scaling and rotation values you want, and then click [Plot] to send the image to the printer queue.

### 3.1.4 Which results you need to turn in

- Make printouts of the circuit schematic (RLC) and the Y[2,1] versus frequency plot (RLC\_Plot).
- Briefly answer these questions:
  1. What does this particular filter do? How does the frequency affect the output from the filter?
  2. How does changing the values of R, L, and C alter the shape or position of the curve?



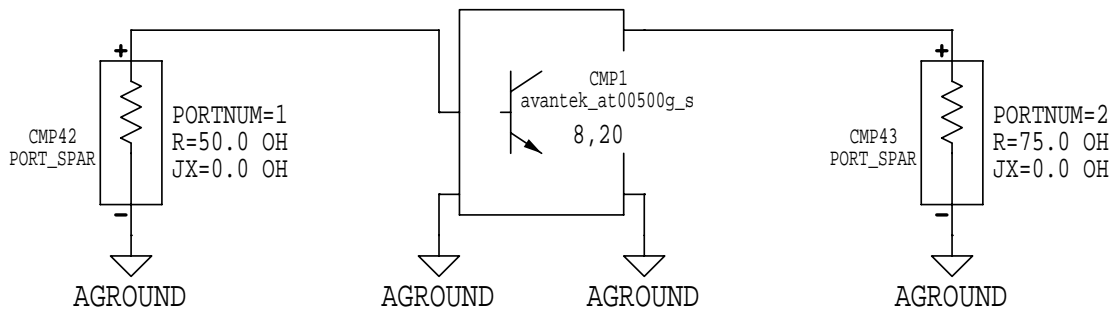
### 3.2 Sample #2: An Amplifier With No Matching Networks

```

EQUATION del=s11*s22-s12*s21
EQUATION Knum=1-mag(s11)^2-mag(s22)^2+mag(del)^2
EQUATION Kden=2*mag(s12*s21)
EQUATION K=Knum/Kden

EQUATION Gmsg=msg(s21/s12)
EQUATION G_Tmax=Gmsg*(K-sqrt(K^2-1))

```



**Figure 2:** A transistor amplifier with no matching networks

For optimum amplifier power transfer, the input and output ports of an amplifier stage should be impedance matched to the source and load impedances. The S-parameters of the stage have to be characterized over the frequency range that the amplifier is supposed to amplify. Once the S-parameters are known, special procedures can be applied to design the matching networks.

As a preliminary step, the amplifier stage is tested by itself with two S-ports. Notice from Figure 2 that the transistor's DC bias network seems to be missing. Well, that particular transistor is a special parameterized component that requires no biasing. This is possible because the device manufacturer has supplied built-in data at a single bias level (e.g.,  $V_{CE} = 8 \text{ V}$  &  $I_C = 20 \text{ mA}$  for the Avantek at00500g). This greatly simplifies S-parameter measurements, as long as the supplied bias is what you want. If you need a different bias level, then you have to simulate an unparameterized version of the same transistor. In that case, you must explicitly design a DC bias network for the transistor.

To simplify matters further, you will simulate the transistor from 1.5 GHz to 2.5 GHz. Over this frequency range the Avantek device is unconditionally stable, which makes designing the matching networks much easier. The center frequency for this sample amplifier circuit is 2 GHz, and the bandwidth is 1 GHz.

At this point, you don't have to understand exactly what S-parameters represent. You can think of them as indicators of an amplifier's performance at microwave frequencies, just like h-parameters do at lower frequencies.

### 3.2.1 How to construct the circuit

- Close all open windows except for 'Lab1: WORKBENCH'. You can leave the 'MW\_Labs : FILE' window open if you wish.
- Create a new workbench called Amp\_Basic. The 'Lab1: WORKBENCH' window should now have two icons, one for the previous RLC workbench and one for the new Amp\_Basic workbench.
- Create a new design called amp\_b within the 'Amp\_Basic: WORKBENCH' window. The 'amp\_b: CIRCUIT PAGE 1' window should appear, and the Circuit Palette should pop up.
- Click [CIRCUIT:Components]. Next click [Library ...] on the Components Palette. The 'Library Browser' dialog window now appears.

In the left-most panel of the window, click [Select by Type] and change it to "Select by Library" instead. Scroll down the list and click on the "avantek" option. In the middle *Sub Group* panel, scroll the list and click on the "BIPOLAR" option. In the right-most panel, scroll down the list and click the "at00500g\_s" option. Click [OK] to finish the transistor selection.

MDS will tell you that the avantek library needs to be loaded. Answer "Yes" when prompted. Now place the transistor block on the circuit page.

- Ground both of the lower terminals on the transistor block. Construct the rest of the circuit as shown in Figure 2. Notice that the S-port on the output of the transistor has an impedance of 75 ohms.

If the S-port icon is no longer on the Components Palette, click [More parts ...] to open the 'Parts' list window. Select the "Connectors" option from the list to get the S-port icon back on the Components Palette. You can also find the S-port on the "MDS dependent sources" or "MDS indep ... sources" options. You could also do [MB:INSERT/MDS SOURCES/S PARAM PORT IMPEDANCE] to insert an S-port on your circuit page.

- At the top of Figure 2 are a set of *EQU* statements. These are user-defined equations that MDS will compute during a simulation. MDS can then be instructed to use the equation values as output variables for data plots and listings.

To put these equations on the circuit page, choose [MB:INSERT/EQUATION], move the mouse pointer to the location you wish to place the equation, and then click to drop the equation onto the page. You need to do this several times to insert all of the equations. Carefully edit each of the equation statements to have the values shown in Figure 2.

For your information, the equation for *Gmsg* defines the absolute maximum stable gain that the amplifier could attain if the S-parameters were "ideal". On the other hand, *G\_Tmax* computes the maximum transducer gain from the actual S-parameter values.

### 3.2.2 How to simulate the circuit

- Click [CIRCUIT:Simulation]. The 'Simulation Setup' dialog window pops up. Set *Start*=1.5 GHz, *Stop*=2.5 GHz, and *Step-size*=25 MHz.
- Click [Simulation Parameters] to pop up the 'Simulation Parameters' dialog window and then click [Output variables ...]. In the *Name* input box of the 'Select Output Variables' dialog window, type *Gmsg* and click [Add]. This adds the equation variable *Gmsg* to the list of output variables. Repeat this process to add *G\_Tmax* to the output variable list. Click [OK] twice to return to the 'Simulation Setup' window. Click [Start] to begin the simulation. After the simulation is over, you can close the 'Simulation Setup' window if you wish.

### 3.2.3 How to plot the output data

- Click [CIRCUIT:Presentation] and create a new display page called amp\_b\_list.
- Choose [MB:INSERT/TEMPLATE/templatelib/S\_Parameters/TABULAR\_listing]. Move the mouse pointer to the display page window and click to drop the template onto the window. This template creates a tabular listing of the S-parameters versus the frequency. Notice that there are four S-parameters ( $S_{11}$ ,  $S_{12}$ ,  $S_{21}$ ,  $S_{22}$ ), and each parameter requires two columns (magnitude and phase).
- Click [PRESENTATION:Listing] to activate the ‘Listing’ dialog window. Click [Select Data ...] to call up the Dataset Browser. Scroll through the *Variables* list and click on the “gmsg” option. Click [OK] twice. MDS will append to the existing S-parameter table a new list column containing Gmsg values. Repeat this process to add the G\_Tmax column.
- Find the column heading that says  $dB(gmsg)$  and change it to “10\*log(gmsg)”. Change the column heading  $dB(g_{tmax})$  to “10\*log(g\_tmax)”. This gives the correct dB gain factor for power.
- You can add your own text comments to either the circuit page or display page by choosing [MB:INSERT/TEXT]. Just move the mouse pointer to where you want to place the text, click, and then start typing your text. Pressing the ESC key will exit the text entry mode.
- Close the current display page window and create a new display page called amp\_b\_chart.
- Click [PRESENTATION:Plot] to activate the ‘Plot’ dialog window. Click [Select Data ...] to activate the Dataset Browser. Scroll the *Variables* list and click on the “S[1,1]” option. Click [OK] to exit. On the ‘Plot’ window, change the *Plot Format* to “Z-SMITH”. Click [OK].

MDS now draws a strange-looking circular plot. This is a Smith chart plot of S[1,1] versus the frequency. You could invoke the same procedure again to make plots of the remaining S-parameters. However, there is a better way to do this, but you first have to erase the S[1,1] plot.

- Notice there is a frame that surrounds the S[1,1] Smith chart plot. Click inside the frame to select the plot and then do [MB:DELETE] to delete it. Although the plot is now gone, there is still a line of text that says  $Trace1=S[1,1]$  remaining on the screen that also needs to be removed. Click and drag a selection box around the unwanted text. The selected text will then be highlighted, at which point you can choose [MB:DELETE] to erase the text.
- Do [MB:INSERT/TEMPLATE/templatelib/S\_Parameters/QUAD\_Smith\_polar]. Move the mouse pointer over to the drawing page. Notice an outline of the template tags along with the pointer. Click the mouse to drop the template onto the drawing page.

MDS now draws four separate circular plots on the drawing page. S[1,1] and S[2,2] are in Smith chart format, while S[1,2] and S[2,1] are in polar format. On the S[2,1] plot, change  $Y-FS=1.0$  to  $Y-FS=5.0$ . This rescales the S[2,1] plot to better fit the data.

### 3.2.4 Which results you need to turn in

- Make printouts of the listing (amp\_b\_list) and the quad Smith/polar plot (amp\_b\_chart).
- Briefly answer these questions:
  1. What do you notice about the S-parameters’ dependence on frequency?
  2. The S-port source and load impedances are 50 ohms and 75 ohms, respectively. If the source were perfectly matched to the transistor input, the magnitude of  $S_{11}$  would be zero. Likewise, if the load were perfectly matched to the transistor output, the magnitude of  $S_{22}$  would also be zero. A bad match would have a magnitude of nearly one.

From your simulation, does the basic transistor by itself show good, average, or bad matching?

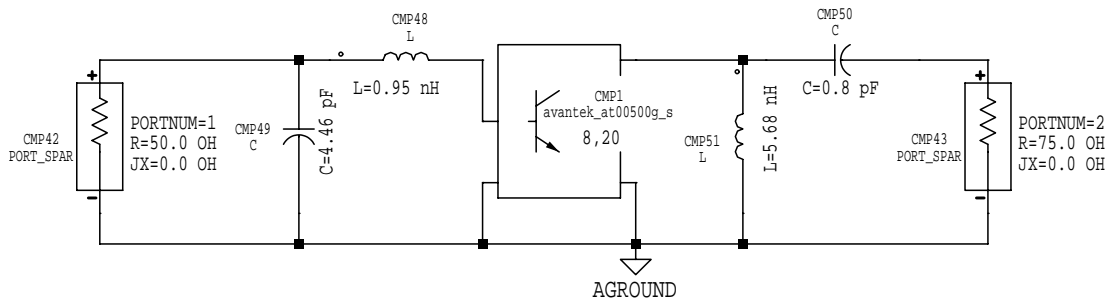
### 3.3 Sample #3: An Amplifier With Lumped Passive Matching Networks

```

EQUATION del=s11*s22-s12*s21
EQUATION Knum=1-mag(s11)^2-mag(s22)^2+mag(del)^2
EQUATION Kden=2*mag(s12*s21)
EQUATION K=Knum/Kden

EQUATION Gmsg=mag(s21/s12)
EQUATION G_Tmax=Gmsg*(K-sqrt(K^2-1))

```



**Figure 3:** A transistor amplifier with lumped passive matching networks

This is basically the same circuit as Figure 2, but now matching networks have been added. The source side Input Matching Network (IMN) and the load side Output Matching Network (OMN) are constructed from idealized lumped passive components.

#### 3.3.1 How to construct the circuit

- Close all open windows except for 'Lab1: WORKBENCH'.
- Make a copy of the Amp\_Basic icon. You can do this in one of two ways.

In the first method, choose [MB:COPY/AS GROUP]. Select the Amp\_Basic icon and drag it to some other part of the 'Lab1: WORKBENCH' window, and then release the mouse button to drop the icon. This creates a copy of the original Amp\_Basic workbench and its contents. You have to perform an unselect operation (via [MB:SELECT/UNSELECT ALL] or clicking [US] on the window's border) to get out of the selection mode.

In the other approach, select the Amp\_Basic icon first, and then choose [MD:COPY/AS GROUP]. Next click the icon again and drag it to some other part of the window. Release the mouse button to drop the icon. You do not have to do an unselect operation.

Notice that the new icon has the name Amp\_Basic~~1 to distinguish it from the original icon. Change the new icon's name to Amp\_Lumped.

- Open the Amp\_Lumped workbench. Notice that the 'Amp\_Lumped: WORKBENCH' window contains the same icons as the original Amp\_Basic workbench.

Rename the amp\_b circuit icon to amp\_L. Change amp\_b\_list to amp\_L\_list. Rename amp\_b\_chart to amp\_L\_chart. Leave DATASET alone.

You don't really have to change the names, but this makes it easier to remember the purpose of the circuits and plots.

- Open the amp\_L circuit icon. Put in the matching networks as shown in Figure 3. Be sure you use the correct component values. Notice that the equations are still defined and do not need to be re-entered (a real time-saver!).

### 3.3.2 How to simulate the circuit

- Since you made a direct copy of the original Amp\_Basic workbench, the simulation parameters are already set up correctly. If you run the simulation, the DATASET object will automatically update itself with the new simulation data.
- Suppose you already had another dataset page called DATASET in the current workbench. Each time you run a simulation, the DATASET object is overwritten with the new data. If you wanted to preserve that original data, what would you do?

Well, you can change the default name that MDS uses for storing data. That way, you can keep different circuit pages in the same workbench, and have each circuit's simulation create its own unique dataset page.

In the 'Simulation Setup' dialog window, click [More]. A list of options will drop down from the button. Click on [Set host/dataset names ...] to pop up the 'Set Host/Dataset Names' dialog window. Change the default dataset name in the input box from "DATASET" to "amp\_L".

It does not matter that the new dataset name is identical to the circuit page name because MDS knows that circuit and dataset pages are different objects. Click [OK] to exit the dialog window.

- Now start the simulation. MDS will create a new dataset icon called amp\_L that holds your simulation data. The information contained in other dataset pages is not disturbed.

### 3.3.3 How to plot the output data

- Because you directly copied the original Amp\_Basic workbench, the listing and Smith chart display pages are already properly formatted, but the information is wrong because you are no longer using the default name DATASET for the dataset page.

In the upper left corner of the listing display page, change the line *Dataset=DATASET* to *Dataset=amp\_L*. This will automatically update the listing to reference the amp\_L dataset page. Make the same change to the *Dataset* line on the Smith chart display page.

- Close the display page windows and create a new display page called amp\_L\_chart2. Manually create a Smith chart for S[1,1] on this display page by clicking [PRESENTATION:Plot] to activate the 'Plot' dialog window. Be sure that the *Plot Format* is "Z-SMITH". In the Dataset Browser, select "amp\_L" as the dataset and pick "S[1,1]" from the *Variables* list. Do not use the template method!

Now suppose you want to see S[2,2] on the same Smith chart plot as S[1,1]. To accomplish this task, click [PRESENTATION:Trace]. In the 'Trace' dialog window, activate the Dataset Browser. Select "amp\_L" as the dataset and choose "S[2,2]" from the *Variables* list. Click [OK] twice to finish.

Instead of creating a separate plot, MDS will draw the S[2,2] data on the same plot as S[1,1]. In general, you can apply this method to put multiple output variables on a single plot for easier comparison.

### 3.3.4 Which results you need to turn in

- Make printouts of the circuit schematic (amp\_L), listing (amp\_L\_list), quad Smith/polar plot (amp\_L\_chart), and the combined Smith chart plot (amp\_L\_chart2).
- Briefly answer these questions:
  1. How does adding the matching networks change the S-parameter values? Do the values of  $G_{msg}$  and  $G_{Tmax}$  change?
  2. How do the quad Smith/polar plots with matching networks compare to the quad Smith/polar plots without matching networks?
  3. The matching networks were designed for a center frequency of 2 GHz. At that frequency, the magnitudes of  $S_{11}$  and  $S_{22}$  should be close to zero (implying a good match). From your simulation data, did this happen? What do you observe at other frequencies?