INTRODUCTION TO MICROWAVE TRANSISTORS
Lab #6

This lab focuses on exploring the basic microwave performance of bipolar junction transistors, as well as a simple approach to the design of a biasing networks for microstrip-based circuits. In this lab session, we'll start with the measurement and assessment of the performance of a microwave bipolar junction transistor (Agilent AT-41485 or Agilent AT-42085). This lab will also serve as a demonstration of measurements using microwave fixtures and network analyzer calibration for connector-less devices such as transistors.

Transistor Characterization

In order to design an amplifier circuit using a transistor, the frequency dependence of the gain as well as the input and output impedances must be determined. For a measurement-based design approach (as an alternative to the use of an equivalent circuit model), the network analyzer and a measurement fixture can be used to make these measurements. The results can then be imported into the CAD software for simulating the performance of circuits using the transistor.

a. Before measuring a device, the network analyzer must be calibrated. Since the transistor does not have standard microwave connectors on its terminals, a test fixture will be used to hold the transistor and bring the test signals in and out of the transistor. In order to calibrate the network analyzer without standard connectors, a set of calibration standards compatible with the test fixture must be used. These calibration standards are electrically similar to those used in previous labs, but they are designed to match the performance of the test fixture as closely as possible. Due to the fragility of these calibration standards, the instructor will perform the calibration; please see the video. Another important point is that since the transistor has significant gain, it is necessary to engage the attenuators that are internal to the network analyzer to avoid overloading the analyzer. The frequency range will be set from 10 MHz to 6 GHz, using 801 data points.

b. As with the previous lab, the IC-CAP software package will be used to control the instruments for measuring the characteristics of the transistor. Unlike the other measurements we have made so far, transistors require both DC and RF signals in order to operate. We will use an Agilent 4155C semiconductor parameter analyzer to supply the DC voltages and currents for the measurements. The outputs (called source-measure units, or SMUs) of the 4155C are connected to the bias connections on the back of the network analyzer; SMU2 is connected to VNA port 1, and SMU3 is connected to VNA port 2. Since the measurement equipment configuration is different than previously (the Agilent 4155C for biasing the transistor has been added and will be operated simultaneously with the network analyzer), go to the hardware window and confirm that the instrument interface is set up (VISA IO libraries are enabled) and click "Rebuild" to update the hardware configuration.

c. An IC-CAP model file to help automate the measurement is available on the course web page (lab6.mdl); download this file and open it in IC-CAP. Before attempting to measure the current-voltage curves for the device, **turn off the**

microwave source in the network analyzer using the front-panel buttons (Press "Local" then "Power" (in the stimulus box, not the power switch…) and then the RF power on/off softkey). These transistors have very high gain, and the DC I-V curves can be corrupted if an RF signal is present during the measurement.

d. Install a transistor in the test fixture, and using the "dc/Ic_vs_Vc_vs_Ib" setup in IC-CAP, obtain a set of current-voltage curves. This setup is configured to act much like a curve tracer; in the default configuration, the collector voltage is swept from 0 to 10 V at several base current steps from 0 to 200 µA. From the data (e.g. from the current-voltage plot), find β (sometimes called h_{FE}, I_c/I_b) for your transistor, and check it against the specification sheet for your transistor. Be sure to note which transistor type (41485 or 42085) you are using.

e. If you have been assigned to build a low-noise amplifier (LNA), the bias point that should be selected is a collector current of approximately 10 mA with a collector-emitter voltage of 8 V. If you have been assigned to build a high-gain amplifier, choose a collector current of 25 mA for the AT-41485, or 35 mA for the AT-42085. You will probably have to interpolate within the measured characteristics to find the exact base current needed to set the bias correctly. If your transistor does not reach the required current, adjust the Ib settings in the setup (often increasing the stop value for Ib and increasing the number of points to keep the step size at 25 µA is a good approach). Be sure to stay below the maximum ratings provided in the data sheets; in particular, $V_{ce} < 12$ V and $I_c < 60$ mA. Save the resulting data and make a note of the filename; we will use this data in the ADS software in a later lab.

f. What are the implications of these choices in terms of maximum collector voltage signal swing? What base current corresponds to this assignment? Be sure to make a note of the full bias details (base current, base voltage, collector current, collector voltage) in your notebook; you will need these to design the bias network.

g. In the "mwv/s_vs_f_at_IbVc" s-parameter measurement setup of the IC-CAP model file, set the bias base current and collector voltage as needed to establish your intended bias point. Be sure to confirm that the instrument options match exactly what was used during the calibration. **In particular, be sure the "Cal/State File Name" field is set correctly (the instructor will provide the name used for the calibration).** Turn the microwave source in the network analyzer on, and have IC-CAP measure the s-parameters of your transistor under bias. The s-parameters should show significant gain in s_{21}, largely capacitive behavior in s_{11}, and a mix of resistive and capacitive impedance for s_{22}.

h. Check to see that the bias is close to your intended point. This is easiest to do by right-clicking on the Ic and Vb "Output" blocks and selecting "View". This brings up a window that shows the DC measured collector current and base voltage, respectively. The "R:measured" column is the measured value; the "I:measured" is the imaginary part of the DC value which is kind of silly. It shows many copies of the same value; they will all be the same (it fills out an array of the same dimension as the frequency sweep). If this is not close to

your intended bias point, make adjustments as needed and re-measure. When
you have everything set correctly, save the model file.

i. It turns out that it is slightly easier to use the data in ADS if you also save just
the RF data in a .set file (rather than selecting the entire model, save only the
RF measurement setup by clicking on the "floppy disk" icon, selecting the .set
option, and making sure that the s_vs_f_at_IbVc option is indicated). Make a
note of your chosen filename in your notebook.

Importing Transistor Data into ADS
To design circuits using the s-parameters that were measured with IC-CAP, it is
necessary to import the data into ADS. We'll do this by setting up a circuit element in
ADS that "contains" the measured s-parameter data from the transistor that we can use in
our design.

a. Create a new schematic design (it is often convenient to work in the same
workspace as the microstrip measurement and optimization from lab 5). Using
the "Data file tool" as before (under the "Tools/Data file utilities" menu),
import the IC-CAP s-parameter .set file that you saved in the previous section,
and enter a unique dataset name. Be sure to note the name and variable name
for the dataset that you choose. When everything is set, click "Read File" to
import the file, and check the status window for error messages.

b. In our actual design, there will be electrical discontinuities at the
board/transistor interface (from the change in geometry), and we need to
account for the layout details (dimensions etc.) of the transistor. To make this
easier, a framework for the transistor is available on the course web site in the
lab 6 section (filename: AT4xx85.7zads). Download this file, and unarchive it
(from the main ADS window, choose "File/Unarchive Project," select the
downloaded file, and press "Next." Set the workspace destination (the same
top-level folder as your other workspace files is often convenient) and press
"Next." When it asks which workspace to open, select "none." We now have
the transistor framework installed, but need to make it available in our
designs. To do this, go to the main ADS window and select "Manage
Libraries." Then choose "Add Library Definition File" and browse to find the
unarchived workspace (the default is AT4xx85_wrk), select the "lib" file
within the workspace, and click "Open." A warning about this library being
in a different workspace from your design may pop up; this is OK. Press
"Close" and the library is now available for use.

Amplifier Design - Bias Networks
The necessary groundwork for designing a microwave frequency amplifier has
been set; the remainder of this lab is a start towards designing an amplifier. The first part
of designing an amplifier is to find a circuit that will properly bias the microwave
transistor at the desired operating point but that will not interfere with the microwave
operation of the circuit.

a. In designing a bias network, it is important to choose a design that will
minimize the "loading" of the rf signal, while still providing the correct DC
bias conditions to the transistor. For this design, it is necessary to supply a
DC base current and collector voltage bias, without loading the microwave
signal present on the base and collector transmission lines. Since we'll be

supplying the bias in shunt (in parallel with the RF signal), this means that the bias must be supplied through a network that presents a low resistance at DC, but a high impedance at the operational frequency of the amplifier. A straightforward way to achieve this is the "bias-tee" circuit below:
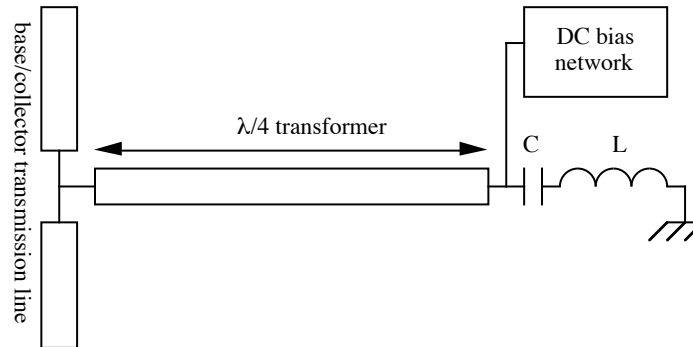

Figure 1. Schematic illustrating bias stub concept.

The capacitor and inductor are realized using a series-resonant chip capacitor whose resonant frequency coincides with the amplifier's design frequency. At series resonance, the impedance of the inductor/capacitor pair is zero. You should verify that this is true for yourself. Thus, at the center frequency of the amplifier, the λ/4 section of transmission line acts as a short-terminated stub. Transforming a short through a quarter-wavelength line results in an infinite input impedance at the center frequency; thus at the design frequency the bias stub will not load the main transmission line. At DC, however, the chip capacitor has infinite impedance, and so the DC bias network circuit is not loaded by the stub or capacitor. Consequently, this is one possible bias network that could be used in an amplifier design.

b. To implement this in ADS, create a new schematic design in the same workspace that will become your amplifier circuit design. Insert an MSUB element with the substrate parameters you found from lab 5, hard-coding any parameters (you will need to copy & paste any values that are set in a VAR element). Insert a microstrip transmission line. Rotate the line so that it is oriented vertically on the schematic; this will become one of the bias stubs for your circuit. Add a series inductor (1.5 nH) and a capacitor to the "top" end of the bias stub. This is the effective inductance of the connection from the top of the board to the bottom. For the series inductance of 1.5 nH, what capacitance is needed for resonance at your assigned design frequency? Set the capacitor to this value.

c. The design of a bias stub consists of determining the appropriate width and length for the microstrip transmission line in order to transform the resonant LC circuit to an open circuit. LineCalc is a utility in ADS that can calculate the performance of microstrip transmission lines that are accurate over a wide range of frequencies and dimensions, and is ideal for this type of calculation. Start LineCalc by going to "Tools/LineCalc/Start LineCalc". Then select the bias stub transmission line (single-click), and choose "Tools/LineCalc/Send Selected Component to LineCalc" from the menu. This will automatically

transfer the substrate parameters and appropriate transmission line model to LineCalc. In the LineCalc window, set the frequency to your assigned design frequency. For the width of the transformer line, use W=50 mil; selection of a narrow width makes for a transmission line with large characteristic impedance, and thus a higher input impedance even if the line is mistuned somewhat. After setting the width, press the "analyze" button. This will calculate the electrical properties (loss, characteristic impedance, electrical length) of the line. The electrical length will probably not be what is desired. Recalling that electrical length = $\beta l = 2\pi l/\lambda$ (in radians), find the desired length (converted to degrees). Set the electrical length, and have LineCalc compute the physical length by pressing the "synthesize" button. LineCalc has now found a $\lambda/4$ transformer at the design frequency with a width of 50 mils.

d. To incorporate this calculation into the schematic page, select "Tools/LineCalc/Update Selected Component" from the menu. The dimensions of the line will be updated to those found in LineCalc.

e. Since two bias stubs are required (one to supply the base current, one to supply the collector voltage), copy this stub (and the inductance and capacitance) to another position in the schematic.

f. Now that the bias stubs have been designed, the circuit to supply the bias current and voltage must be designed. There are many circuits that can be used, ranging from resistive bias networks to current-source biasing schemes and even more complex designs with temperature compensation. Select an approach, and using the DC data acquired earlier in the lab, design a circuit to bias the transistor. A number of references are available to help you in selecting a topology--in particular, see Agilent application note 1293 (on the course web page) for a comparison of some of the choices. Notice that the resonant LC circuit at the end of the bias stubs will act like a short-circuit at the design frequency, and so the details of your circuit selection will not strongly affect the operation of the amplifier at microwave frequencies.

Amplifier Bias Circuit Layout

a. With the bias stubs designed, we're ready to begin laying out the design of the amplifier circuit. Figure 2 below shows a view of the bias portion of the completed amplifier. Use LineCalc to find the dimensions (width only) for the base and collector input lines (the horizontal ones in Fig. 2), assuming a 50 Ω characteristic impedance at the design frequency. At this point the length of these lines is not critical and can be finalized later in the design process.

b. To insert the transistor, select "Insert/Component/Component Library", select "Workspace Libraries" in the pop-up window. Within this library, right-click on the "AT4xx85_transistor" component (right-hand pane of the window), and choose "Place component." Mouse over to the schematic window where you want to insert the transistor, and click where the transistor should go. Close the library window.

c. So far, this transistor framework does not include the measurement data from the transistor. To make this connection, click on the transistor symbol in the schematic, and then select "View/Push Into Hierarchy." This will open a

schematic representation of the electrical behavior of the device. Double-click on the S2P_Spac element, and set the "File" parameter to be the dataset you imported the transistor data into previously (step (a) under Importing Transistor Data into ADS above). Save the schematic file, and return to the main schematic ("View/Pop Out of Hierarchy").

d. Connect the transistor to the input and output lines, and attach the bias stubs you've designed to the line at least 5 mm (250-300 mils) away from the transistor. An example screenshot showing the circuit with bias tees on both base and collector is shown below. Also as shown, you should use the MTEE element to connect the bias stubs to the input and output line for accurate simulation of the right-angle transitions there. Be sure to "match up" the widths in the MTEE elements to the corresponding transmission lines.
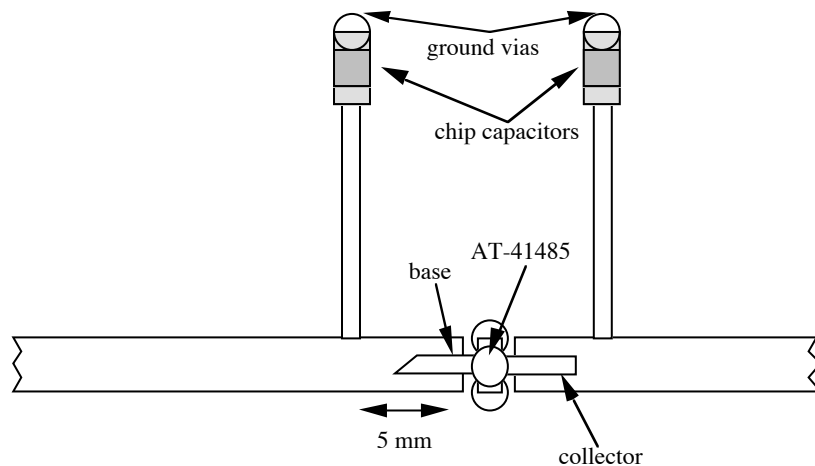

Figure 2. Physical layout of input and output transmission lines, bias stubs, and transistor.

e. Set up an s-parameter simulation (matching the frequency plan used for the measurement of the transistor), and simulate the circuit to make sure everything is working. The s-parameters should show some gain in s_{21} at the design frequency, and deep resonances in s_{21} and s_{12} well away from the design frequency due to the bias stubs. An example is shown below; you can see appreciable gain at the design frequency (2 GHz in this example).

We now have a "core" that we can use to implement an amplifier. It has poor input and output matching, and frequency response is not very good, but we can improve these by designing input and output matching networks.
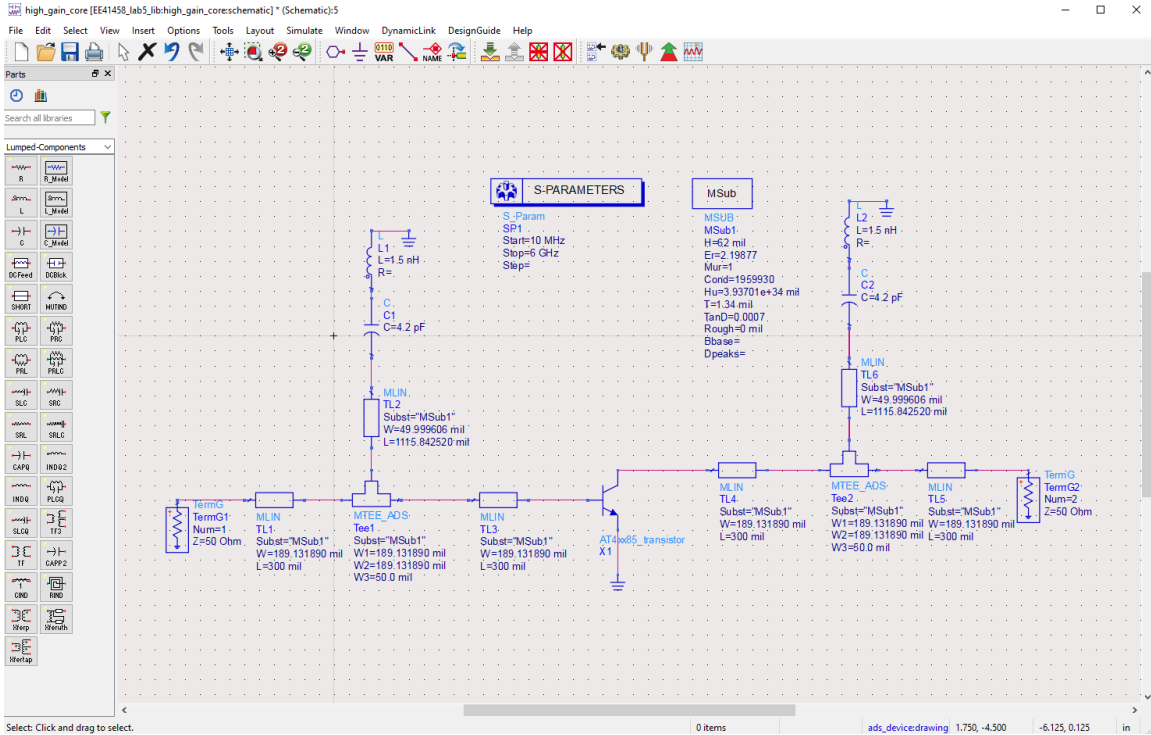
Screenshot: Sample schematic showing attachment of bias stubs.
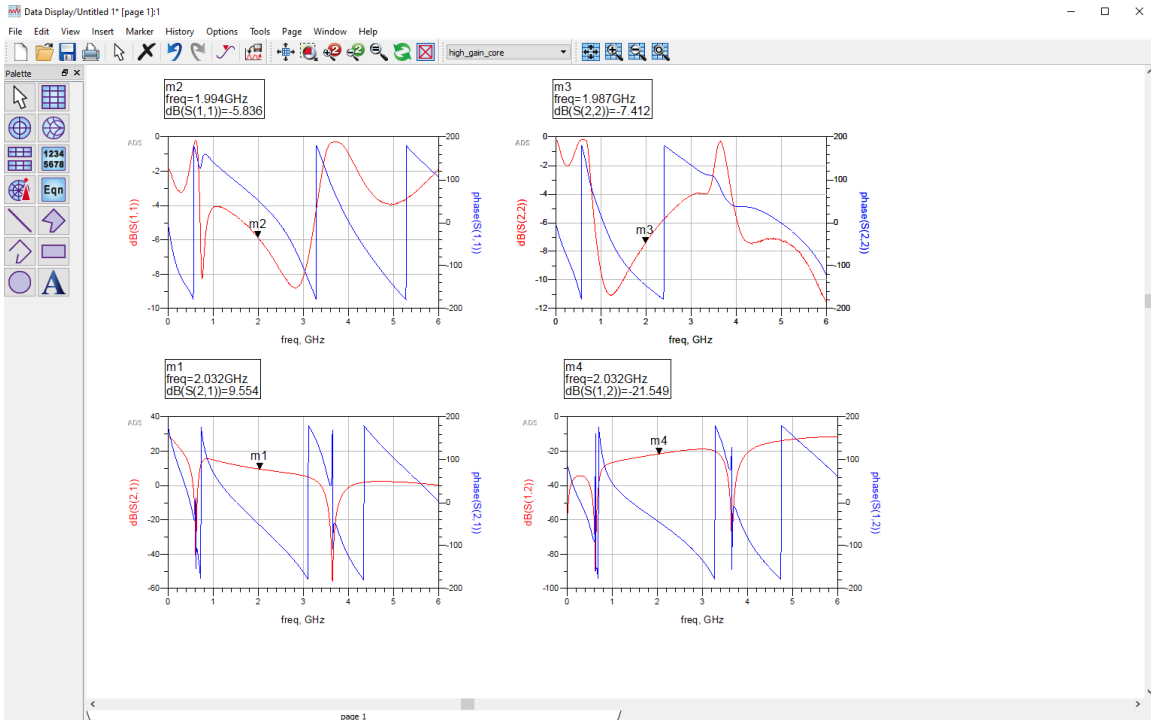
Screenshot: Example simulation result for a 2 GHz design.