

INTRODUCTION TO MICROWAVE TRANSISTORS

Lab #7

In this lab, the basic properties of microwave bipolar junction transistors are explored, and the design of a microstrip-based circuit for amplifying microwave frequency signals is begun. In this lab session, we'll focus on 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 connectorless 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. The network analyzer and a measurement fixture will be used to make these measurements. The results can be imported into the CAD software for simulating the performance of circuits using the transistor.

1.) Before starting the measurement, 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; only the physical configuration is designed to match the performance of the test fixture as closely as possible. Due to the fragility of these calibration standards, the instructor will demonstrate this calibration. 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.

2.) As with the previous lab, the IC-CAP software package will be used to control the instruments for measuring the characteristics of the transistor. The HP 4155C semiconductor parameter analyzer will be used to supply the DC voltages and currents for the measurements. The outputs (SMU's) of the 4155C should be connected to the bias connections on the back of the network analyzer; SMU1 should connect to Port 1, and SMU2 to Port 2. Once these connections have been verified, start IC-CAP (source "ee40458.csh" or "ee40458.sh", then run "iccap"). Since the measurement equipment configuration is different than previously (the HP 4155C for biasing the transistor has been added), go to the hardware window and confirm that the interface "lan[cush116e5810a.ee.nd.edu]:gpib0" is set up and click "Rebuild" to update the hardware configuration. A suitable IC-CAP model file is available on the course web page in the laboratory section; 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 (the source power on/off menu option is under the power menu). These transistors have very high gain, and can damage the network analyzer by supplying too much power to port 2 if the input power is not low enough. Obtain a set of current-voltage curves for the AT-41485 or AT-42085 using the dc measurement setup in IC-CAP. Be sure to stay below the maximum ratings provided in the data sheets; in particular, $V_{ce} < 12 \text{ V}$ and $I_c < 60 \text{ mA}$. Save the resulting data (as a .set file) and make a note of the filename; we will use this data in the ADS software in a later lab. From the data (e.g. from the current-voltage plot), find β (h_{FE}) for your transistor, and check it against the specification sheet for your transistor. Be sure to note which transistor type you are using.

3.) If your group has 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 your group has 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. Check your assigned bias point with your measured current-voltage data. What are the implications of these choices in terms of maximum collector voltage signal swing? What base current corresponds to this assignment?

4.) In the s-parameter measurement setup of the IC-CAP model file, set the bias base current and collector voltage. Turn the microwave source in the network analyzer on, and 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} . Save the data (as a .set file) for use later in ADS.

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 an equivalent 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 (this can be in the microstrip test line workspace if you like or in a separate workspace). Select the "Data file tool" under the "Tools" menu. This will open a new window that allows you to read data files into ADS. Select "Read data file into dataset" and choose ICCAP as the format. Browse for the 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.) Close the data file tool window. In the schematic window, insert an S2P element (located in the "data items" palette; insert an SnP element, and change the "n" to 2 after placing it on the schematic page) and associate it with the dataset that you just imported from the IC-CAP .set file. Be sure to set the file type to "dataset," and set the block name to the measured S parameters (under "view dataset" in the S2P pop-up window). Since in the actual amplifier we will have a discontinuity between the input and output transmission lines and the transistor, add an inductor in series with both ports 1 and 2. Also, the emitter (which will be grounded in the amplifier design) has series inductance associated with it; add a third inductor on the "ref" terminal of the S2P to account for this. In a separate demonstration the determination of appropriate discontinuity inductances will be demonstrated; for now, assume a value of 0.25 nH for all three of the inductors. This makes up the equivalent circuit of the transistor.
- c.) To easily use this circuit as a sub-circuit in designs, a few more steps are needed. Add "ports" to the circuit at the free end of each series inductor. The ports should be numbered 2 for the base (port 1 on the S2P), 1 for the collector (port 2 on the S2P), and 3 for the emitter (ref on the S2P). The "port" icon is the hexagonal symbol next to the ground icon in the toolbar; the defaults in the pop-up box are usually OK. Be sure to save your schematic design.
- d.) Now that the ports are defined, all that is required is to associate this circuit with a schematic symbol to use in subsequent designs and a layout for the actual component. This is defined by making a symbol design associated with your schematic. In the main ADS window (not the schematic window, but the one you used to create a new schematic), click on the "New symbol window" button (looks like an op-amp). This will open a pop-up window; click the "..." button and choose the schematic you saved in step c, and click "Create Symbol." In the "symbol generator" window that opens, select the "Copy/Modify" tab, and

navigate to the Devices-BJT table. Select the “BJTNPN” option, and click “OK.” This will make this subcircuit appear as a standard BJT in the schematic drawings. Save the design. This circuit element is now ready to use in more complicated designs.

Amplifier Design - Bias Networks

The necessary groundwork for designing a microwave frequency amplifier has all been set; the remainder of this lab is a start towards designing an amp. 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.

- 1.) 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 circuit diagrammed below:

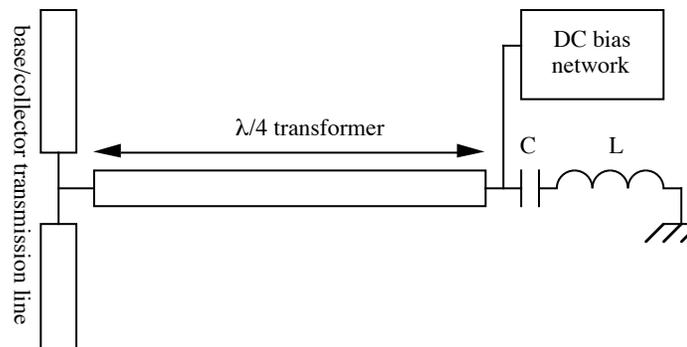


Figure 1. Schematic illustrating bias stub concept.

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 $\lambda/4$ section of transmission line acts as a short-terminated stub. Transforming a short through a quarter-wavelength 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. Thus we have found one possible bias network that is suitable for the amplifier design. The “linecalc” feature of ADS is convenient for performing these types of designs.

- a.) 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 6. 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.

- b.) 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 contains models for microstrip transmission lines that are accurate over a wide range of frequencies and dimensions, and it ideal for this type of calculation. To start linecalc, 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 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 = $\beta l = 2\pi l/\lambda$, 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.
 - c.) To incorporate this calculation into the schematic page, select “Tools/LineCalc/Update Selected Component” from the menu. The dimensions of the line will automatically be updated to those found in linecalc.
 - d.) 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.
- 2.) 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

- 1.) 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, assuming a 50 Ω characteristic impedance at the design frequency. At this point the length of these lines is arbitrary and will be set later in the design process. Insert the transistor equivalent subcircuit, connect it to the input and output lines, and attach the bias stubs you’ve designed to the line at least 5 mm away from the transistor. To insert the transistor, select “Insert/Component/Component Library”, and click on the “Workspace Libraries” in the pop-up window. Within this library, right-click on the component name

you used when you saved the transistor schematic, and choose “Place component.” Mouse over to the schematic window you want to insert the transistor into, and click where the transistor should go. Figure 3 shows an example circuit. As shown in Figure 3, you can use the MTEE element to connect the bias stubs to the input and output line for accurate simulation of the transition there.

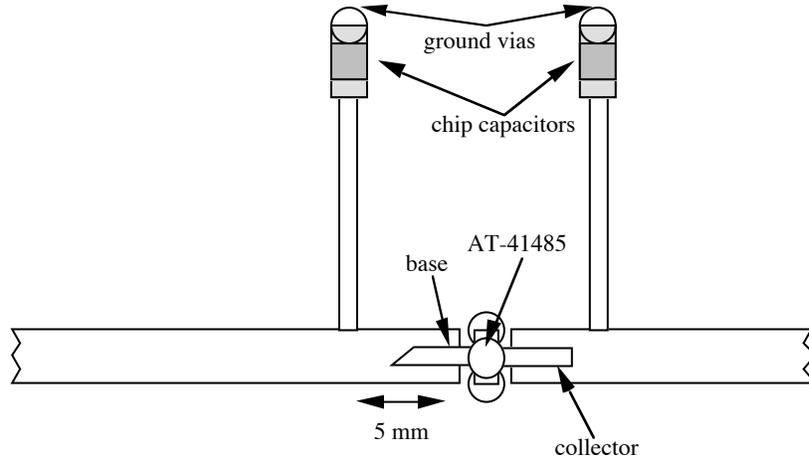


Figure 2. Physical layout of input and output transmission lines, bias stubs, and transistor.

- 2.) Simulate this circuit to make sure everything is working. The s-parameters should show some gain in s_{21} , and resonances in s_{11} and s_{22} well away from the design frequency due to the bias stubs.

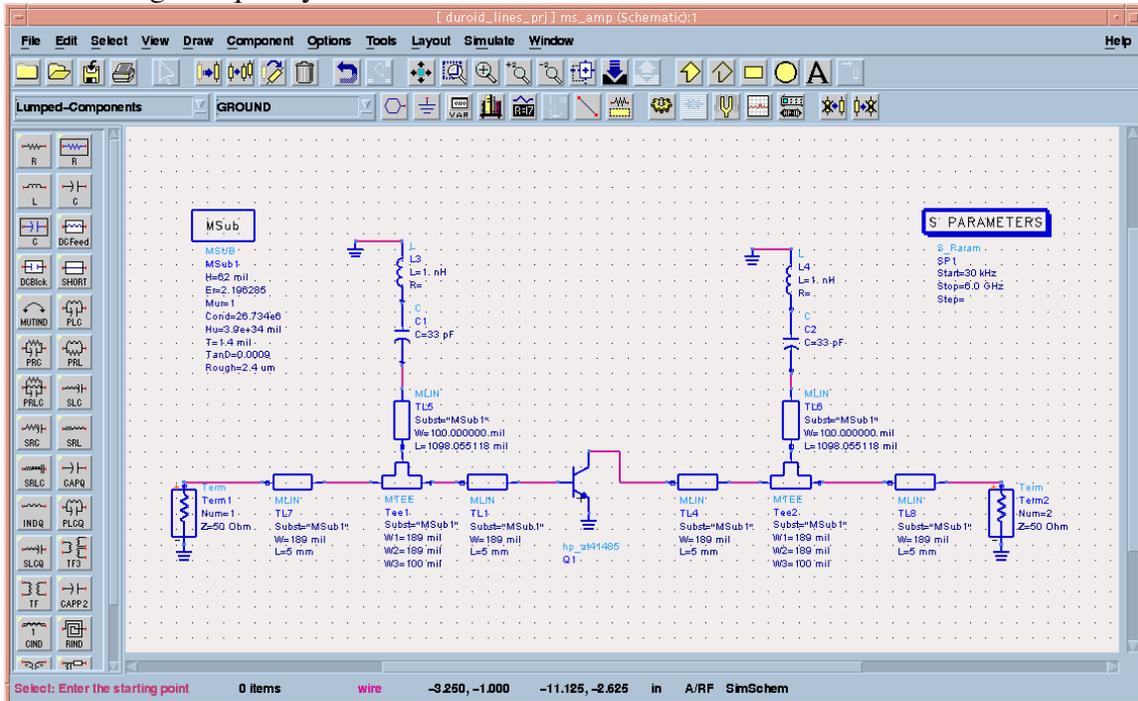


Figure 3. Sample schematic showing attachment of bias stubs.