Microwave Circuit Design: Lab 6

1. Introduction

This lab looks at the design process behind a simple two-port negative-resistance oscillator circuit. Special procedures for testing and simulating oscillator circuits in MDS are also introduced.

2. Design Specifications

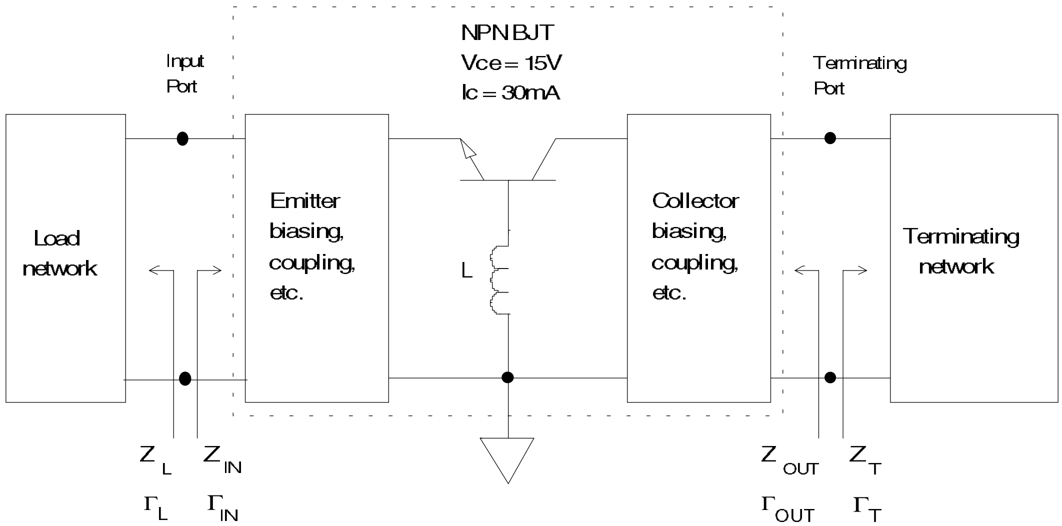

Figure 1: Block diagram of an oscillator using a BJT

You are to construct an oscillator circuit[1] that has a fundamental oscillation frequency of f_{osc} = 3 GHz.

The core of the oscillator is an NPN bipolar junction transistor (BJT) in common-base configuration. In MDS, you can choose transistors that use either linear models or nonlinear models. The linear models are suitable for linear, small-signal applications. However, oscillator circuits assume large-signals and are highly nonlinear by nature. Hence, a nonlinear transistor model is needed in order to accurately simulate the behavior of the oscillator. MDS has a default nonlinear BJT model that you may use for your circuit. The default model will require a few parameter modifications to work properly.

The nonlinear BJT device is not self-biased, so you will need to add DC bias circuits in order to power the oscillator. Fortunately, the bias networks are already provided in the textbook, so you will not have to design them.

In Figure 1, Γ_T and Z_T are the terminating reflection coefficient and terminating impedance, respectively. Also, Γ_L and Z_L are the load reflection coefficient and load impedance, respectively. Γ_{IN} is the reflection coefficient looking into the input port of the BJT circuit (which includes the bias circuits).

[1] The original circuit, along with explanatory notes, can be found on pages 446-448 in the textbook "Microwave Transistor Amplifiers, 2nd Edition".

3. Design Approach

To keep the lab simple, the following procedure will be used:

1. Select a transistor that is potentially unstable at the frequency of oscillation.
2. Choose a Γ_T for the terminating network that will make $|\Gamma_{IN}| > 1$, where $\Gamma_{IN} = \dfrac{S_{11} - \Delta\Gamma_T}{1 - S_{22}\Gamma_T}$.
3. Calculate Γ_L for the load network that will resonate Z_{IN} at the oscillation frequency. If $Z_{IN} = R_{IN} + jX_{IN}$, then $Z_L = R_L + jX_L$ where $R_L = \dfrac{|R_{IN}|}{3}$ and $X_L = -X_{IN}$.

4. BJT Circuit Characteristics

4.1 Assignment

In this section, you will determine the characteristics of the NPN BJT. In order to get valid results, the transistor should be operating at its designed bias point, so the DC bias networks must be in place.

4.2 Circuit construction

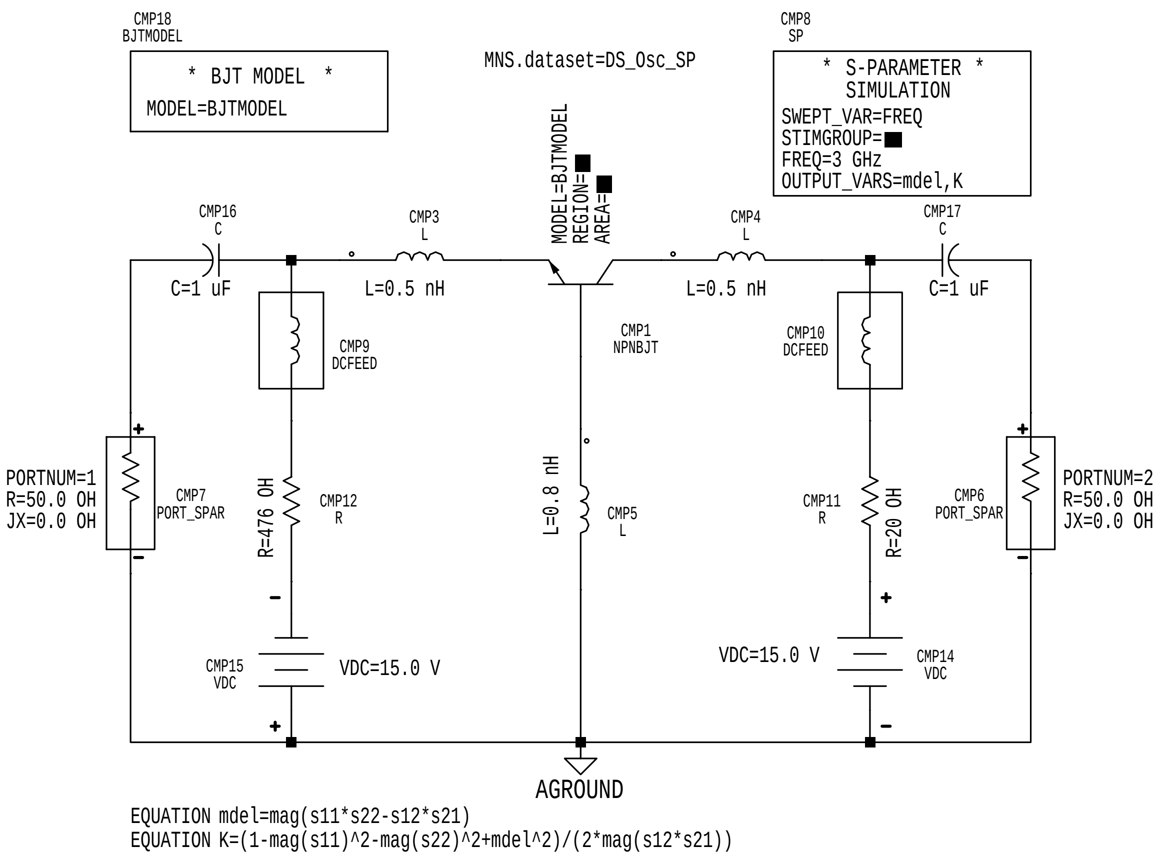

Figure 2: Characterization test circuit

The NPN BJT is configured for common-base operation. The transistor has a *MODEL* parameter that specifies the name of the BJT model to be used in the simulation. The *MODEL* parameter only specifies the model name, but does not define the model parameters.

To actually define the parameters in the nonlinear BJT model, you have to insert a model component onto the circuit page. There are two ways to do this:

1. **BJTMODEL component**
   This component looks like a small box with the label * *BJT MODEL* * and a single parameter called *MODEL*. There are no other visible parameters in the box. The *MODEL* parameter should be set to the same name you specified for the BJT. To edit the model parameters, you need to select the component and then choose [MB:PERFORM/EDIT COMPONENT]. MDS will then open a new dialog window that displays all of the model parameters that you can change.

   The main advantages of the BJTMODEL component are that it hides the details of the model and uses the least amount of space on the circuit page. For all of the schematics shown in this lab, the BJTMODEL approach is used exclusively.

2. **BJTMODELFORM component**
   This component is a big rectangular box with the label * *BJT MODEL* * and the parameter *MODEL* (which should be set to the same name you specified for the BJT). Also within the box is a list of all the model parameters that you can edit.

   The principal advantage of using BJTMODELFORM is that every model parameter is visible at all times, so changing a value here or there is easy. Of course, the major disadvantage is that the form takes up a lot space on the circuit page.

The schematic on page 446 of your textbook is **incorrect** because it has both model components on the same circuit page. You may choose either model component (the effect is the same), but do not use both of them simultaneously (at least not with the same names).

The DCFEEDs, resistors, and DC voltage supplies form the bias networks for the transistor (V_{CE} = 15 V and I_C = 30 mA). The capacitors block DC from reaching the *S*-ports. Attached to the BJT's collector and emitter are 0.5 nH inductors, which model the inductance of the lead wires. The inductor connected to the base of the BJT provides feedback that enhances the instability of the transistor. At the bottom of the figure are equations for computing the stability factor *K* and the magnitude of Δ.

In the upper right corner of the figure is a control box for performing an *S*-parameter simulation. You will use this control instead of accessing the standard 'Simulation Setup' dialog window.

- Construct the circuit shown in Figure 2.
- To access the NPN BJT, click [MB:INSERT/MDS COMPONENTS/NONLINEAR DEVICES/BJT/NPN DEVICE]. Set *MODEL*=BJTMODEL next to the transistor.
- The DCFEED, resistor, capacitor, and inductor components are all located in the same lumped component menu/palette.
- To get the BJTMODEL box, choose [MB:INSERT/MDS COMPONENTS/NONLINEAR DEVICES/BJT/MODEL]. Position the box and click the left mouse button to drop it. Select the BJTMODEL box by clicking on it and use [MB:PERFORM/EDIT COMPONENT] to access the model parameters. These are the values you need to set (see page 446 of the textbook):

  BF=150, *VAF*=75, *TF*=30p, *BR*=1, *TR*=75n, *IS*=2p

CJC=1.5 pF, *VJC*=0.75, *MJC*=0.5, *CJE*=1.5 pF, *VJE*=0.75, *MJE*=0.5

RB=5, *RE*=1, *RC*=3

If you prefer to use the BJTMODELFORM box instead, choose [MB:INSERT/MDS COMPONENTS/NONLINEAR DEVICES/BJT/MODEL FORM] and enter the required parameter values directly into the form.

- To place the *S*-parameter simulation control box on the circuit page, choose [MB:INSERT/MDS CONTROL/ANALYSIS/S PARAMETER/SMALL-SIGNAL]. Erase any value next to the *STIMGROUP* parameter. Set the *FREQ* parameter to 3 GHz. Define the equation variables *K* and *mdel* as output variables by listing them next to the *OUTPUT_VARS* parameter.
- The *MNS.dataset=* line defines the name of the dataset that will hold the results of the simulation. Use [MB:INSERT/MNS/DATASET NAME] to place the line onto the circuit page. You can use whatever name you wish, as long as you remember to use the same dataset name for your presentation page.

4.3 Simulation

- Highlight the control box labeled * *S-PARAMETER SIMULATION* * by clicking it. Start the simulation by clicking the [S] button on the left side of the circuit page window.

4.4 Output

- Use a template to create a tabular listing of the *S*-parameters. Add listing columns for *K* and *mdel*.
- Using the techniques of the previous lab (refer to *Lab 5: Amplifier Design for Noise*), create a blank Smith chart on the same presentation page as the *S*-parameter listings.

  Insert a trace for the load stability circle, which defines the allowable values of Γ_T. Insert another trace for the source stability circle, which defines the permitted values of Γ_L. (It sounds backwards, but that is how Figure 1 defines the oscillator circuit's terminating and load networks.)

  Identify and label the <u>unstable</u> regions of the Smith chart for Γ_T and Γ_L.

4.5 Items to turn in

- Turn in a printout of the characterization circuit.
- Submit a printout of the *S*-parameters and stability circles.

4.6 Questions

1. At a frequency of 3 GHz, what is the stability of the BJT circuit (with its bias networks, etc.)? Justify your conclusion. Pay particular attention to the *S*-parameter, *K*, and $|\Delta|$ values.
2. From the stability circle data, are there any restrictions on the choice of either Γ_T or Γ_L?
3. The textbook suggests using $\Gamma_T = 0.8\angle 70°$ for the terminating network. Does that particular reflection coefficient fall within an unstable region?

5. BJT Circuit Stability vs. Base Inductance

5.1 Assignment

You will examine the influence of the BJT's base inductor on the stability of the BJT circuit.

5.2 Circuit construction

- The stability versus base inductance test circuit is identical to the circuit in Figure 2. You do not have to make any changes.

5.3 Simulation

- Configure MDS for an *S*-parameter simulation using the 'Simulation Setup' dialog window (you won't be using the *S*-parameter control box this time). Set MDS for a single point sweep at 3 GHz. From the dialog window, change the dataset name to prevent overwriting any previously saved data.
- Next, initialize a parameter sweep of the feedback inductor's *L* value. Set the sweep range from 0.0 to 1.0 nH in 0.l nH steps (use the "nano" multiplier). Now run the simulation.

5.4 Output

- Plot both $|S_{11}|$ and $|S_{22}|$ versus *L* on a single graph. On a separate graph, plot *K* and $|\Delta|$ versus *L*.

5.5 Items to turn in

- Turn in the presentation page with the *S*-parameter vs. *L* and stability factors vs. *L* graphs.

5.6 Questions

1. The circuit in the textbook uses *L* = 0.8 nH for the base inductor. From your plots, explain why this inductance value is a reasonable choice.

6. Load Network Calculations

6.1 Assignment

In this section, you will determine Γ_{IN} from Γ_T . Once Γ_{IN} is known, you can then compute the required Γ_L for the load network.

6.2 Calculations

- As mentioned earlier, the textbook suggests using $\Gamma_T = 0.8\angle 70°$ because it is a "convenient" value for the terminating reflection coefficient. From your prior stability circle work, there are clearly many possible choices for Γ_T that may lead to oscillation. How the author settled on that particular value of Γ_T is somewhat unclear, but we will assume that there was a logical reason for doing so. Hence, use the textbook value for Γ_T in your design calculations.

- Once the value of Γ_T has been fixed, Γ_{IN} can be found from the equation $\Gamma_{IN} = \dfrac{S_{11} - \Delta\Gamma_T}{1 - S_{22}\Gamma_T}$. Using the S-parameters computed by MDS for the BJT circuit, calculate Γ_{IN} and express the answer in magnitude and phase format.
- With the results of the previous step, compute Γ_L and write the solution in magnitude and phase format. Also calculate the equivalent unnormalized load impedance Z_L.

6.3 Items to turn in

- Submit your calculations for Γ_{IN} , Γ_L , and Z_L.

6.4 Questions

1. Does the computed Γ_L value fall within an unstable region on the Smith chart?

7. Verifying Γ_{IN} Using MDS

7.1 Assignment

You will use MDS to experimentally find Γ_{IN} and then compare its value to your calculated Γ_{IN}.

7.2 Circuit construction

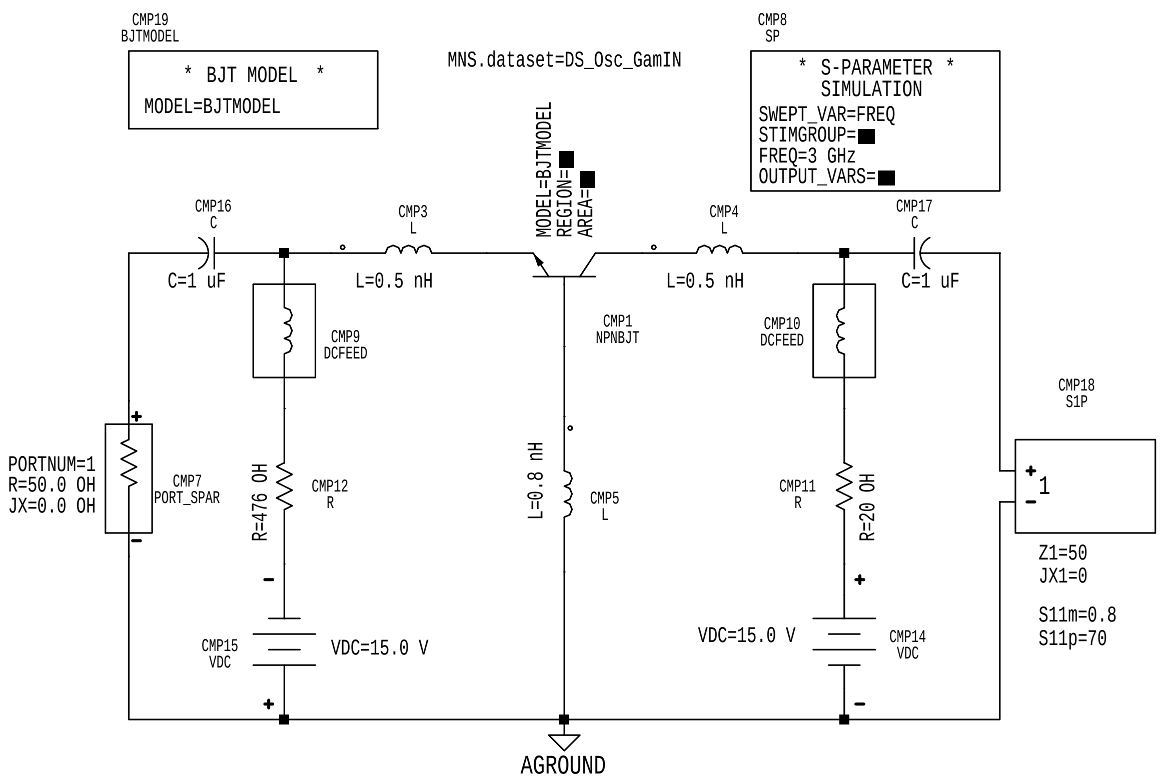

Figure 3: Γ_{IN} test circuit

This is essentially the same circuit as in Figure 2, but the *S*-port on the terminating port has been replaced by a special component labeled *S1P*. S1P is a single port device that presents a fixed reflection coefficient at its input terminals. This allows you to easily insert a terminating network without worrying about the actual implementation details.

- Construct the circuit shown in Figure 3.
- Use [MB:INSERT/MDS COMPONENTS/LINEAR DEVICES/MODEL/ONE PORT/S MAG PHASE] to access the S1P component.

  Set *Z1*=50 and *JX1*=0. This tells MDS that the S1P port impedance is 50 Ω (remember that your *S*-parameter measurements used 50 Ω ports).

  Set *S11m*=0.8 and *S11p*=70. This forces the reflection coefficient presented at the terminals of S1P to be $0.8\angle 70^\circ$. A nice feature of the S1P component is that the reflection coefficient is fixed and won't change even if the frequency shifts (which is a definite problem for networks built from discrete elements or microstrip).

  Don't confuse the *S11m* and *S11p* parameters with the normal *S*-parameters that MDS computes. *S11m* and *S11p* refer <u>only</u> to the S1P component and have nothing to do with *S*-parameters elsewhere in the circuit.

7.3 Simulation

- Highlight the control box labeled * *S-PARAMETER SIMULATION* * by clicking it. Start the simulation by clicking the [S] button on the left side of the circuit page window.

7.4 Output

- Record the magnitude and phase of S_{11} at the input port of the BJT circuit.

7.5 Items to turn in

- There are no formal presentation pages to turn in. However, state your recorded S_{11}.

7.6 Questions

1. For the circuit in Figure 3, $\Gamma_{IN} = S_{11}$. Does the simulation value for Γ_{IN} match your manually calculated value?

8. Basic Oscillator Tests

8.1 Assignment

You will test the oscillator circuit to determine whether or not it has the potential to oscillate.

8.2 Circuit construction

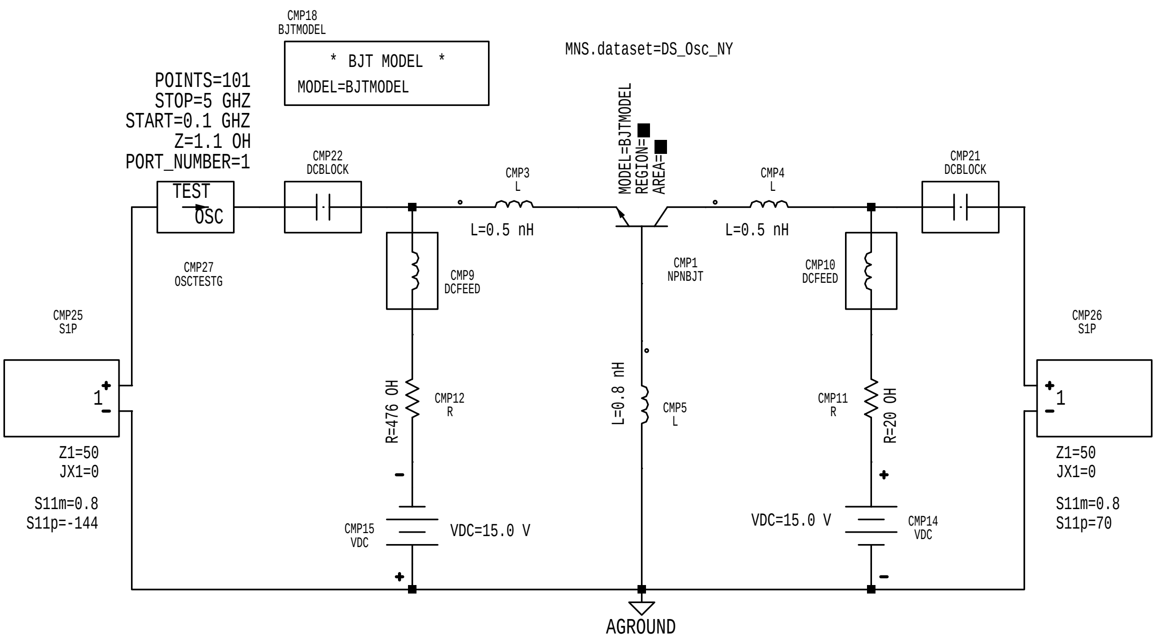

Figure 4: Oscillator test circuit (OSCTESTG - Nyquist diagram)

The circuit in Figure 4 is similar to the one in Figure 3, with four major exceptions:

1. The * *S-PARAMETER SIMULATION* * control box is omitted.
2. The 1 µF blocking capacitors are replaced by DCBLOCK components.
3. The *S*-port that was on the input port of the BJT circuit is replaced by an S1P component (to simulate the load network).
4. A new device labeled *OSCTESTG* is inserted in the circuit. It is a probe that measures the open-loop gain and phase of the closed-loop system. By plotting the results on a polar graph (Nyquist diagram), you can determine if the circuit has the potential to oscillate.

Note that the OSCTESTG probe only checks whether or not the circuit satisfies certain oscillation conditions. It does <u>not</u> actually solve the nonlinear equations necessary to simulate the oscillation behavior of the circuit.

- Construct the circuit shown in Figure 4.
- The load network S1P component is initially set up for $\Gamma_L = 0.8\angle -144°$, which is the non-optimized value mentioned in the textbook. Instead of using that value, edit the *S11m* and *S11p* parameters to use the Γ_L value you computed in the calculations section of the lab.

- For negative resistance oscillators, OSCTESTG is inserted between a negative and positive impedance in the circuit.

  Choose [MB:INSERT/MDS CONTROL/ANALYSIS/S PARAM OSC TEST] to insert the OSCTESTG probe.

  Set *PORT_NUM*=1 to have MDS test S_{11}.

  The *Z* parameter is the initial probe impedance and should be about a factor of five lower than the magnitude of the passive load impedance. You can use the default value of *Z*=1.1 OH.

  Set *START*=0.1 GHz, *STOP*=5 GHz, and *POINTS*=101. This tells MDS the range of frequencies you want tested for possible oscillation.

8.3 Simulation

- Highlight the OSCTESTG probe by clicking it. Start the simulation by clicking the [S] button on the left side of the circuit page window.

8.4 Output

- On a single graph, plot the magnitude and phase of S_{11} versus the frequency. Both axes should be on a linear scale. Adjust the limits of the y-axis to make the horizontal grid line running through the center of the graph represent zero magnitude and zero phase.

  Frequencies at which the phase of S_{11} goes to zero correspond to possible frequencies of oscillation. Depending on the OSCTESTG frequency search range, there may be multiple zero phase crossings. At each zero phase crossing, if $|S_{11}| > 1$ then the circuit has the potential to oscillate at that frequency. If $|S_{11}| < 1$, then the open-loop gain is less than unity and the circuit will not oscillate at that frequency.

  From your graph, locate the frequency at which the phase of S_{11} is zero. Now examine the magnitude of S_{11} at that same frequency. You can get better accuracy by inserting markers on the phase and magnitude traces.

- On the same presentation page, create a Nyquist diagram by making a polar plot of S_{11}.

  If a circuit is capable of oscillating, then the S_{11} curve must make a clockwise track on the polar plot as the search frequency increases, and the curve must encircle the point $1 + j0$ (the unity gain point).

  Insert a marker on the S_{11} curve. Move the marker along the S_{11} trace until you reach the positive real x-axis (i.e., imaginary coordinate is zero). The marker display will tell you the magnitude of S_{11} and the frequency at that point.

8.5 Items to turn in

- Turn in the plot containing the S_{11} vs. frequency (LIN-LIN) graph and the Nyquist diagram.

8.6 Questions

1. From your plot information, what is the potential oscillation frequency?

9. Oscillator Harmonic Balance Test

9.1 Assignment

The OSCTESTG probe of the previous section is useful for performing certain linear tests that examine the potential for oscillation. However, OSCTESTG cannot test the oscillation behavior itself, since that requires a nonlinear simulation. To actually test the oscillation mode of the circuit, you will perform a nonlinear simulation using the technique of harmonic balance.

9.2 Harmonic balance simulation

The nonlinear oscillator simulation tool is the OSCPORTG component, which is inserted into the oscillator circuit to perform a harmonic balance test. In a harmonic balance analysis, a frequency spectrum is built from individual sinusoids, which are then applied simultaneously to the circuit being simulated. The magnitudes and phases of the sinusoids are the Fourier coefficients of the corresponding time-domain waveform. After a harmonic balance analysis is performed, the result is a set of voltage and current spectra for each node and component in the circuit. Because the nonlinear analysis makes fewer assumptions on waveform behavior than does linear analysis, the results are more rigorous. Since the final loop gain of the oscillator depends on the steady state large signal amplitude, harmonic balance provides a more accurate picture of the oscillator's actual performance.

9.3 Circuit construction

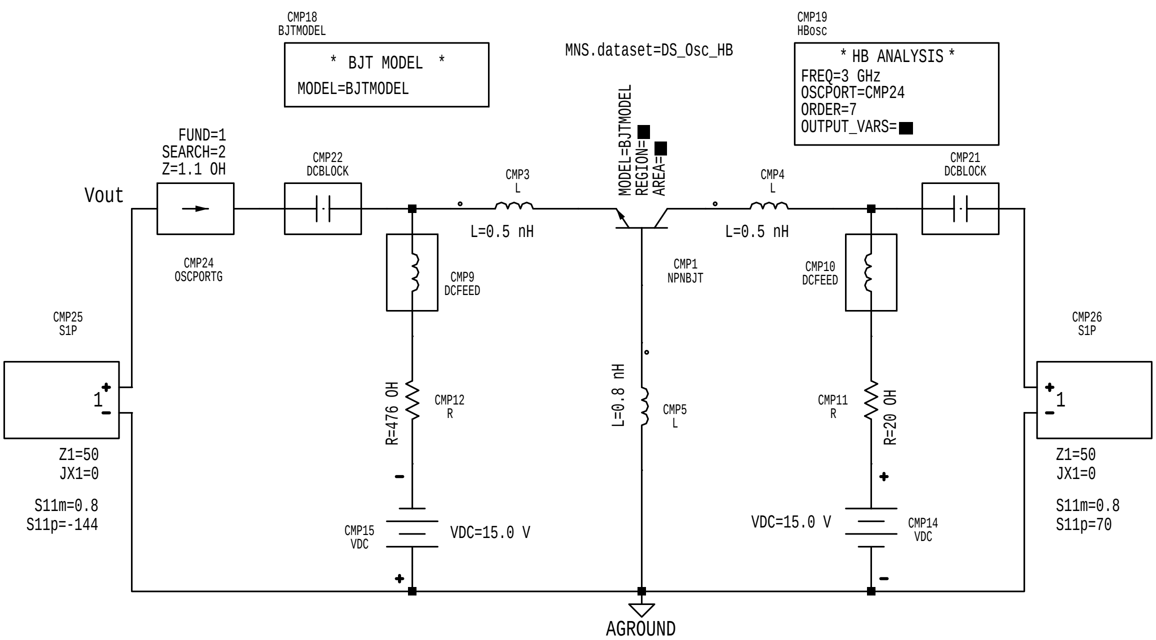

Figure 5: Oscillator test circuit (OSCPORTG - Harmonic balance)

The harmonic balance test circuit in Figure 5 is similar to the Nyquist test circuit of Figure 4, with two exceptions:

1. The * *HB ANALYSIS* * control box is added.
2. The OSCPORTG component replaces the OSCTESTG device.

- Construct the circuit shown in Figure 5.
- For the load network S1P component, edit the *S11m* and *S11p* parameters to use the Γ_L value you computed in the calculations section of the lab.
- For output plotting purposes, you need to identify the voltage output node of the oscillator circuit. The required node is located between the S1P and the OSCPORTG components and is labeled as *Vout* on the schematic.

  To label the node, choose [MB:INSERT/WIRE/LABEL]. An outline of a tiny box connected to crosshairs (that looks like an “X”) tags along with the mouse pointer. Position the crosshairs somewhere on the wire that joins the S1P and OSCPORTG components and click the left mouse button to drop the label. Click inside the tiny blue box to edit it, and type `Vout`.
- Choose [MB:INSERT/MDS_COMPONENTS/PROBES/OSCPORT GROUNDED] to access the OSCPORTG component.

  Z is the initial probe impedance, *SEARCH* is the number of octaves to search, and *FUND* is the fundamental number for the oscillator. Use the default values.
- Choose [MB:INSERT/MDS CONTROL/ANALYSIS/HARMONIC BALANCE/OSCILLATOR] to insert the * *HB ANALYSIS* * control box.

  FREQ is the initial guess for the oscillation frequency. Set *FREQ*=3 GHz.

  The *OSCPORT* parameter identifies which OSCPORTG is to be used for the analysis. Suppose *CMP#* is the component number of OSCPORTG in your own circuit. Set *OSCPORT=CMP#*.

  The *ORDER* parameter sets the upper limit on the number of harmonics computed. Set *ORDER*=7.

9.4 Simulation

- Highlight the * *HB ANALYSIS* * control box by clicking it. Start the simulation by clicking the [S] button on the left side of the circuit page window.
- If your oscillator schematic is correct, then the harmonic balance analysis should be successful. If the simulation fails, the resulting error message will probably be very cryptic and not of much help. If that happens, look over your circuit very carefully to find the problem and then re-run the harmonic balance analysis.

9.5 Output

- On a new presentation page, create these three plots:

  <u>Spectrum vs. Harmonic Index</u>
  Choose [MB:INSERT/TEMPLATE/templatelib/Harmonic_Balance/Spectrum]. Position the outline of the template on the presentation page and click the left mouse button to drop the template. MDS then displays the power spectrum (in dB) at the *Vout* circuit node versus the harmonic index.

  <u>Spectrum vs. Frequency</u>
  Use [MB:INSERT/TEMPLATE/templatelib/Harmonic_Balance/Spectrum] to insert another spectrum plot. Click inside the x-axis label and change it from *harmindex* to *freq*. MDS will redraw the spectrum plot as a function of frequency. If necessary, perform an autoscale on the axes to get a nice looking plot. Add a marker to the spectrum trace. Move the marker to the first spectral line to identify the fundamental (first harmonic) oscillation frequency.

<u>Waveform vs. Time</u>
Choose [MB:INSERT/TEMPLATE/templatelib/Harmonic_Balance/Waveform]. Position the outline of the template on the presentation page and click the left mouse button to drop the template. MDS then displays the voltage waveform (in volts) at the *Vout* circuit node versus time (in seconds).

- On the same presentation page, add a listing column. The column heading should be changed to "freq[2]", which is where the first harmonic frequency is stored.

9.6 Items to turn in

- Submit a printout of the harmonic balance test circuit.
- Turn in a printout of the spectrum/waveform plots and the frequency listing.

9.7 Questions

1. From the spectrum plot data, what is the fundamental oscillation frequency of the circuit? What is the percentage difference between the frequency computed by harmonic balance and the design value of 3 GHz?

   Note: You may have realized that the potential oscillation frequency predicted by OSCTESTG was relatively far away from the actual oscillation frequency. This should reinforce the idea that the linear open-loop gain/phase plots should serve only as a rough guide to the behavior of the oscillator.
2. It is evident that the voltage waveform at the output node is sinusoidal, but there is also a distinct "kink" in the waveform. What do you think is causing this effect? (Hint: Look at the higher order harmonics in the spectrum plot.)

10. Optimization of f_{osc}

10.1 Assignment

The fundamental oscillation frequency of the basic oscillator circuit was close to, but not exactly at the desired frequency of 3 GHz. You will now apply some optimization techniques to get a better fit.

10.2 Harmonic balance optimization

In the textbook, the author simply states that using the value $\Gamma_L = 0.75\angle -140^\circ$ will nudge the oscillation frequency to 3 GHz. Hence, you need to optimize the *S11m* and *S11p* parameters of the load network S1P component.

In the past, you optimized a circuit by using MDS's built-in optimization routine. Unfortunately, the oscillation frequency of an oscillator circuit cannot be optimized using that routine. The only way to get around this glaring omission is to perform parameter sweeps.

Theoretically, you should perform a two-level nested sweep on *S11m* and *S11p*. To save time and effort on your part, you are allowed to perform two individual sweeps. That is, hold *S11m* fixed at 0.75 and sweep *S11p*, and then hold *S11p* fixed at -140 degrees and sweep *S11m*. Since we're already told what the optimal load coefficient should be, this allows you to see the dependence of the oscillation frequency on Γ_L without having to wade through the output from a nested sweep.

10.3 Circuit construction

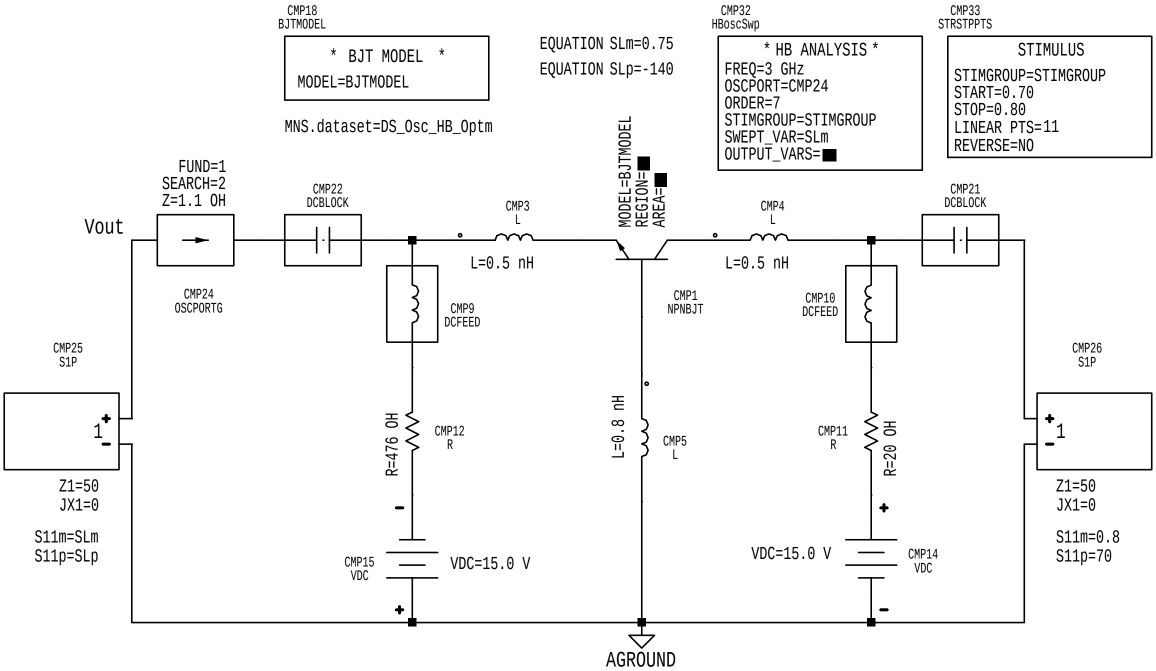

Figure 6: Optimization circuit

Figure 6 is basically the same as Figure 5, with a few critical additions:

1. Equations for *SLm* and *SLp* have been added to define the *S11m* and *S11p* values, respectively. This is necessary because MDS does not allow you to directly access the *S11m* and *S11p* fields of the S1P component for a parameter sweep.
2. The normal * *HB ANALYSIS* * control box is replaced by a special swept version.
3. A *STIMULUS* box is added to define the sweep range.

- Construct the circuit in Figure 6.

- Add the equations for *SLm* and *SLp* to the circuit page. For the load network S1P component, set *S11m=SLm* and *S11p=SLp*.

- Choose [MB:INSERT/MDS CONTROL/ANALYSIS/HARMONIC BALANCE/OSCILLATOR - SWEPT] to insert the swept version of the * *HB ANALYSIS* * control box.

  Use the same values for the *FREQ*, *OSCPORT*, and *ORDER* parameters as in the non-swept harmonic balance simulation. Set *SWEPT_VAR* to *SLm* if you want to sweep the magnitude of Γ_L, or else set *SWEPT_VAR* to *SLp* to sweep the phase of Γ_L.

- Choose [MB:INSERT/MDS CONTROL/STIMULUS/START STOP/LINEAR POINTS] to insert the *STIMULUS* box.

  For a magnitude sweep, set *START*=0.70, *STOP*=0.80, and *LINEAR PTS*=11.
  For a phase sweep, set *START*= -145, *STOP*= -135, and *LINEAR PTS*=11.

10.4 Simulation

- Highlight the * *HB ANALYSIS* * control box by clicking it. Start the simulation by clicking the [S] button on the left side of the circuit page window.

10.5 Output

- After the simulation is done, create a new presentation page. For a magnitude sweep, insert a listing column for *SLm*. For a phase sweep, insert a listing column for *SLp* instead. Add another listing column to show the oscillation frequency at each sweep point (do this by setting the column header to "freq[*,2]").

10.6 Items to turn in

- Turn in printouts of the listings for f_{osc} versus the magnitude and phase of Γ_L.

10.7 Questions

1. From the sweep data, is $\Gamma_L = 0.75\angle -140°$ a good value for the optimal load reflection coefficient?

11. Terminating and Load Networks

11.1 Assignment

In the final section of this lab, you will replace the S1P components (and their fixed reflection coefficients) with normal circuit networks (and their frequency dependent reflection coefficients).

The design requirements are f_{osc} = 3 GHz, $\Gamma_T = 0.8\angle 70°$, and $\Gamma_L = 0.75\angle -140°$. Remember that the original *S*-parameter measurements were based on 50 Ω port impedances. This means that the terminating network must transform Γ_T to 50 Ω, and the load network must transform Γ_L to 50 Ω.

You may build the terminating and load networks using either ell networks or microstrip.

11.2 Circuit construction

- Use the circuit shown in Figure 5 as a starting point.
- Delete the S1P components and replace them with terminating and load networks of your own design. You should tune the reflection coefficients of the terminating and load networks with the built-in MDS optimizer before using them in the oscillator circuit.
- **IMPORTANT**: Delete the DCBLOCK components and replace them with 0.1 μF capacitors.

  For some mysterious reason, the harmonic balance analysis will fail if you replace the S1P components with normal networks and retain the DCBLOCK components.
- Make sure the *Vout* node label is properly located on the schematic.

11.3 Simulation

- Perform a harmonic balance simulation of the completed oscillator circuit. Use the same analysis settings as in Figure 5.

  If the simulation fails (which is possible), then check the design of your terminating and load networks. Play around with some of the OSCPORTG and HB analysis parameters to see if that helps.

11.4 Output

- Make spectrum and waveform plots at the *Vout* node (like the ones you made before).

11.5 Items to turn in

- Turn in a printout of the oscillator circuit with your own terminating and load networks in place.
- Turn in the spectrum and waveform plots.

11.6 Questions

1. What is the oscillation frequency of your circuit? What is the relative error between the circuit's actual oscillation frequency and the design frequency?
2. Do the power spectrum and voltage waveform look any different from the results you got using the S1P components?