EE 514 DESIGN PROJECT – Ku-Band Transmitter  
DUE DATE: DECEMBER 13 BY 5:00PM

This design project is required for students enrolled in EE 514 only. Students enrolled in EE 414 are NOT required to complete this design project.

I. Purpose:
The purpose of this design project is to design a Ku-band microwave transmitter. The transmitter will consist of a differential oscillator, hybrid coupled amplifier, and filters. A block diagram of the system is shown in Fig. 1. The differential oscillator and the hybrid coupled amplifier are assumed to be built as MMIC/VLSI chips. These chips are then integrated into a PCB with microwave integrated circuit (MIC) filters and 3 dB hybrid couplers as shown in Fig. 1.

II. Design Specs:
System Level:

Design Frequency: The frequency of operation is to be 18 GHz plus an offset (in MHz) determined by your name. Your final design frequency will be given by:

\[ f_0 = 18000 + 10k_1 + k_2 \text{ MHz} \]  

where \( k_1 \) and \( k_2 \) are in MHz and are determined using the formula below.

Assign a value of:
- \( k_1 = 500 \) if the first initial of your first name is between A and G inclusive
- \( k_1 = 300 \) if the first initial of your first name is between H and M inclusive
- \( k_1 = 100 \) if the first initial of your first name is between N and Z inclusive

Assign a value of:
- \( k_2 = 750 \) if the first initial of your last name is between A and J inclusive
- \( k_2 = 500 \) if the first initial of your last name is between K and R inclusive
- \( k_2 = 250 \) if the first initial of your last name is between S and Z inclusive

For example, my first name is Nathan therefore \( k_1 = 100 \) and my last name is Neihart so \( k_2 = 500 \). My design frequency would therefore be \( 18000 + 1000 + 500 = 19500 \text{ MHz} \).
Device: The transistor that you will use for this design project is a modified version of the Infineon BFP620 transistor. The model has been modified so as to remove the package parasitics. This model MUST be downloaded from the course web site. Do NOT use the design kit. A datasheet is included in the project directory on Canvas. The DC bias for the part must be set to $V_{CE} = 2.0 \text{ V}$ and $I_C = 30 \text{ mA}$. Note that this bias value is different from what is used in lab.

Gain: The gain of the system will be dominated by that of the amplifier. See the amplifier section in Section III below for more details.

Bandwidth: The bandwidth of the system will be determined by that of the various filters. See the filter sections in Section III below for more details.

Substrate: The substrate material used for this project has a dielectric constant of 13 with a loss tangent of 0.0004. The board thickness is 0.005 inch and the conductor is copper with conductivity of $5.8\times10^7 \text{ S/m}$. The thickness of the copper is $2 \mu m$ (78.74E-6 inch). The parasitic inductance of the ground vias must be calculated for this new substrate. Do not use the values from lab.
III. Design Description:

Device Modeling:

Transistor Model:

The first task that should be performed is to obtain a touchstone file (containing the S-parameters) of the active device over the frequency range of 10 MHz to 100 GHz. Unfortunately, we do not have such a file, so you will need to create one. This can be done by performing an S-parameter simulation over the desired frequency range (e.g., 10 MHz to 100 GHz) using the transient model of the BFP620 downloaded from the course webpage, then exporting the resulting S-parameters into a Touchstone file format. You must remember, however, that the S-parameters are bias dependent, so it is important that you properly bias your transistor before you measure the S-parameters. To set the DC bias conditions, you can use voltage sources in series with resistors or you can use DC blocking capacitors. Usually, it is easier to use the termination resistors in series with a DC source. You should make the DC source voltages (one to set $V_{CE}$ and one to set $I_C$) larger than the desired bias voltage to account for the voltage drop across the series resistors, see Fig. 2.

For layout purposes, you should assume that the part is packaged in a 20 mil X 20 mil package with pads dimensions shown in Fig. 3(a).

Component Model:

Assume that all $R$, $L$, and $C$ components are housed in 0201 (20 mil long by 10 mil wide) packages. You will need to estimate the package parasitics using the supplied “measured” data for various resistors, inductors, and capacitors. The parts were measured without any feed structures (i.e., the data is for the packaged part ONLY). Three different values for each component were measured. The nominal resistor values are: 10 Ω, 50 Ω, and 1 kΩ. The nominal values for the capacitors are: 1 pF, 5 pF, and 10 pF. Finally, the nominal values for the measured inductors are: 500 pH, 1.5 nH, and 5 nH. The value of the part is listed in the header of the appropriate S2P file.

Finally, you will need to create separate models in ADS for the resistor, capacitor, and inductors. See the attached handout describing how to do this. While you must use lumped-
element resistors, it is advisable to use transmission line stubs for inductor and capacitors where possible.

Circuit Design:

Hybrid Coupled Amplifier:

The hybrid coupled amplifier (see “hybrid_amp.pdf” on Canvas) is essentially two amplifiers operating in parallel amplifying signals that are 90° out of phase. Once one amplifier is designed, it can simply be copied to form the complete hybrid coupled amplifier. The amplifier should meet the following specifications:

- Gain: Maximize (Justify your results against maximum stable transducer gain)
- $S_{11} < -15$ dB
- $S_{22} < -15$ dB
- Stable ($\mu > 1$) from 10 MHz to 100 GHz

**Differential Oscillator:**

The oscillator that will be designed in this project will be a fully differential version of the negative resistance oscillator that was designed in the Lab. A simple write-up on the design procedure for these oscillators is provided on Canvas but students are encouraged to find alternative sources as well. The oscillator should oscillate only at your design frequency, $f_0$, and it should be stable (i.e., satisfy Cauchy-Rieman conditions).

The output of the amplifier should not clip due to being overdriven by the oscillator. You should determine the maximum input to the amplifier that will not cause clipping of the output waveform. You should then ensure that the oscillator output does not exceed this value.

**Narrowband End Gap Coupled Filter:**

The first filter is an end gap coupled filter and is used to filter the oscillator output. The filter should be designed to drive a 50 Ω load which will consist of the input impedance of the amplifier. A write-up (by Prof. Robert Weber) on end gap coupled filters is provided on Canvas. The filter is a four-pole Butterworth filter with the following specifications:
- Bandwidth: 2% of your design frequency
- Element values
  - $g_1 = 0.765366865$
  - $g_2 = 1.84775907$
  - $g_3 = 1.84775907$
  - $g_4 = 0.765366865$
  - $g_5 = 1$

**Narrowband Edge Coupled Line Filter:**

The second filter is an edge coupled line filter and is used to filter the harmonics of the amplifier output. This filter will be a three-pole Butterworth filter and have the following characteristics:
- Bandwidth: 5% of your design frequency
- Element values
  - $g_1 = 1$
  - $g_2 = 2$
  - $g_3 = 1$
  - $g_4 = 1$
3 dB Coupler:

The 3 dB couplers can be made from transmission lines or lumped constant equivalents. See Chapter 7 in Pozar’s Microwave Engineering 4th Ed. for a complete discussion on power dividers and directional couplers.

DC Bias Regulator:

The amplifiers and oscillator must be properly biased. You are free to use any DC bias circuit that you wish, however, the bias circuit must be stable to disturbances.

IV. Deliverables

Milestone 1 (Due November 4): Component models and Amplifier Design
Milestone 2 (Due November 25): Differential Oscillator Design and DC Bias Circuit
Final Report (Due December 13): Complete System Performance

Milestone reports are intended to be a progress check. They will be very leniently graded and are to ensure students are working steadily. As long as you include each of the items requested below you will receive full credit. A final design report is also required for this project. Below is a list of items that should be included. Please be advised that all schematics should be of sufficient quality that I can clearly see all component values.

Milestone 1: Component Models and Amplifier Design
- Clearly show the lumped-element models for the resistor, capacitor, and inductor as well as the models’ S-parameters plotted on top of the “measured” S-parameters. Only $S_{11}$ and $S_{21}$ are required.
- Clearly show the schematic of a single amplifier.
- Show plots of $S_{11}$, $S_{21}$, and stability (i.e., $\mu$) of the amplifier.
- Complete derivation of the parasitic inductance of the ground vias, including the diameter of hole that you are using (in mils).
- Describe all design decision as well as any challenges that arose during the design process.

Milestone 2: Differential Oscillator Design and DC Bias Circuit
- Clearly show the schematic of the differential oscillator
- Clearly show the schematic of the DC bias circuit
- Show Harmonic Balance simulations of the small-signal loop gain demonstrating startup.
- Show Harmonic Balance simulations showing precise frequency of oscillation
- Use Harmonic Balance to show the steady-state transient output of the oscillator, clearly showing differential signaling.
- Discuss any design decisions as well as any challenges that arose during the design.
Final Report: Complete System Performance

For the final report, you will be assembling the previously designed components, along with the design of the filters and 3dB coupler into a complete system. You are free to change anything from your previous milestone reports, just make sure that you describe your reasoning. The report must be typed and all citations in IEEE format, including class notes. Below is a list of what is expected to be in your final report.

Final Report Requirements:

1. Component Model – Description along with:
   a. Schematic of the parasitic model for the resistor showing values along with the model S-parameters plotted along with the “measured” S-parameters
   b. Schematic of the parasitic model of the capacitor showing values along with the model S-parameters plotted along with the “measured” S-parameters
   c. Schematic of the parasitic model of the inductor showing values along with the model S-parameters plotted along with the “measured” S-parameters
   d. Transistor characterization, including schematic of how the transistor was biased and the resulting S-parameters.
   e. Complete derivation of the parasitic inductance of the ground vias, including the diameter of hole that you are using (in mils).

2. Hybrid-Coupled Amplifier – Description along with:
   a. Schematic for a single amplifier
   b. Schematic for the DC bias regulator
   c. Plot of $|S_{11}|$ in dB centered at your design frequency with a frequency span of 10%. Place a marker at your design frequency.
   d. Plot of $|S_{22}|$ in dB centered at your design frequency with a frequency span of 10%. Place a marker at your design frequency.
   e. Plot of $|S_{21}|$ in dB centered at your design frequency with a frequency span of 10%. Place a marker at your design frequency.
   f. Plot of Mu from 10 MHz to 100 GHz with a MIN marker placed showing the minimum value of Mu.
   g. Transient plot of the output of the amplifier obtained from Harmonic Balance simulations.
   h. Calculation of the output power from the transient analysis

3. Differential Oscillator – Description along with:
   a. Schematic of the oscillator design
   b. Harmonic Balance simulations showing the small-signal loop gain demonstrating startup
   c. Harmonic Balance simulations showing the precise frequency of oscillation
   d. Use Harmonic Balance to show the steady-state transient output of the oscillator, clearly showing differential output signals
e. If an attenuator is required, provide the schematic of the attenuator.

4. End Gap Coupled Filter—Description along with:
   a. Report the resonator parameters that were required to build the oscillator
   b. Provide a plot of $|S_{21}|$ and $|S_{11}|$ clearly showing the bandwidth and pass-band of the filter

5. Parallel Gap Coupled Filter—Description along with:
   a. Report the resonator parameters that were required to build the oscillator
   b. Provide a plot of $|S_{21}|$ and $|S_{11}|$ clearly showing the bandwidth and pass-band of the filter

6. 3 dB Hybrid Coupler
   a. Report on the chosen design
   b. Use Harmonic Balance simulations to show the transient operation of the coupler
   c. Use Harmonic Balance simulations of the hybrid amplifiers combined with the couplers

7. Complete layout showing all components placed.