



Integrated Circuits and Systems

<http://www.ics.isy.liu.se/en/>

TSEK02 – Radio Electronics

LAB 1: RADIO TRANSCEIVER SIMULATIONS

By Morteza Abbasi, 2013

2014-10 Updated by Duong Quoc Tai (tai@isy.liu.se)

2015-12 Updated by Ted Johansson (ted.johansson@liu.se)

Introduction

In this lab you will simulate a communication link, from the modulator and transmitter to the receiver and demodulator.

You will use Matlab/Simulink in this lab. Log onto your computer. For compatibility with the lab manual and the *.mdl-files, it is important that you load the correct version of the program which is **2013a** or **8.1**.

Check the matlab modules using the command:

```
module avail
```

and scroll the list and check for the -matlab/8.1.

Load the matlab module

```
module add matlab/8.1
```

Start matlab using the command

```
matlab
```

Ask the lab assistant if you encountered any difficulties running the program.

You should make copies of the simulation files to your local computer. The files can be found in
`/site/edu/eks/TSEK02/2016/laboration/`

You can also download them from the link at the course page

```
http://www.isy.liu.se/en/edu/kurs/TSEK02/laboration/
```

Copy them to your own directory. Open them from matlab (double-clicking the local .mdl-files from within matlab is a good method).

During the lab, you are asked to perform several tasks. Make sure you write down your answers and calculation results for future references. In first parts of the lab you are given almost all components and arrangements you need to run the simulation. You are expected to select and make your own arrangements as you move towards the end of the lab.

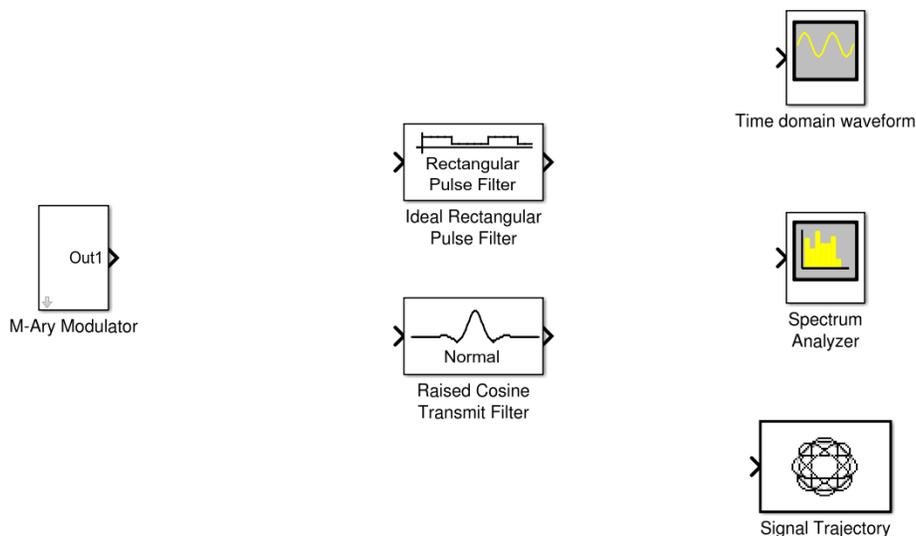
The manual should be self-contained, but ask the lab assistant if you need further clarification or guidance.

If you not have used Simulink before, here are a few tips:

- connect the blocks by using the mouse and draw connect lines from one block output to the next input.
- you should disable the components you do not need in your simulation. You will find this option when you right-click on a component (“comment-out” and “uncomment”).
- you can not connect to outputs together, even if the blocks are disabled. (You will see that directly in the Part 1 with two different filters.
- if you e.g. want to connect two instruments to one output, CTRL-click with the mouse on an existing connection to make another connection. Or draw the line from the instrument to the other connection.
- when you run the simulations, the plots from the “instruments” sometimes appears automatically on the screen or you have to double-click them to show the plots.
- for some elements (e.g. sources, amplifiers, etc.), you can click on the element while the simulation is running and directly change value settings (e.g. gain, IP3, etc.). A nice way to quickly see the effect of a new parameter setting!

Part 1: Pulse Shaping

In the first part of the lab you will simulate the effect of pulse shaping on the time-domain waveform and spectrum of the signal. Open the Simulink model file “Part1_PulseShaping.mdl” (it may take quite some time to open the file in Simulink, so be patient!).



The M-Ary Modulator maps random data sequences onto amplitude and phase of the signal.

You will compare two pulse shaping filters. Rectangular Non-Return-to-Zero (NRZ) and Raised Cosine (RC) pulses. For the raised cosine pulse you can set the value of the roll-off factor α . The roll-off factor determines the bandwidth that the signal occupies.

The measurement instruments are available to view the signal in the time domain (two traces will be shown corresponding to the I and Q signals) as well as in the frequency domain and also in the form of constellation diagram.

- Set the modulation order to 4 (QPSK).
- Set the symbol rate to 1.
- Connect the output of the modulator, to each of the pulse shape filters and then to each of the instruments.
- Run the simulation.

Task 1-1: Explain qualitatively how the bandwidth is affected by the time domain waveform for each pulse shape?

Task 1-2: Change the roll-off factor of the Raised Cosine filter and explain how the bandwidth is affected. Compare the measured bandwidth to the theoretical value.

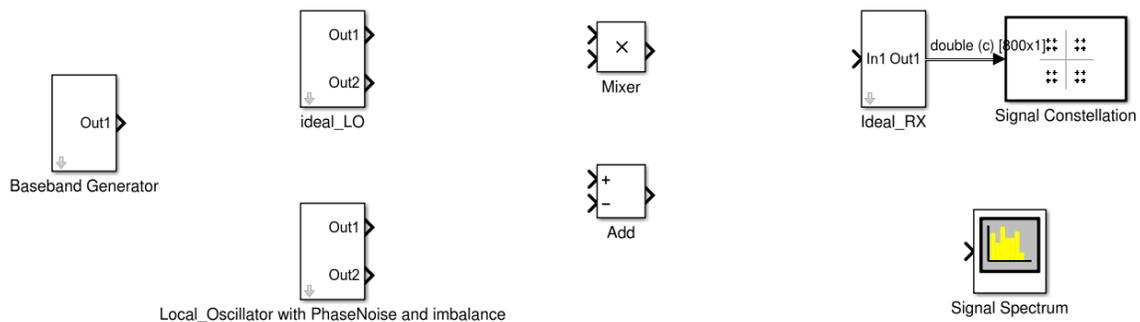
Task 1-3: By referring to the time-domain waveforms, explain the why trajectories of the signals with different pulse shapes look different in the constellation diagram

Task 1-4: Update the modulator to generate BPSK, QPSK, and 16QAM signals and relate the measured time-domain waveforms of these signals to the constellation diagram.

Task 1-5: Check the spectrum analyzer view for each modulation format in Task 1-4 and explain how the bandwidth is affected? How is the level of the signal affected?

Part 2: Quadrature Modulator

In the second part of the lab you will make a quadrature modulator and investigate its performance in presence of phase noise, amplitude/phase imbalance and carrier leakage. Open the Simulink model file “Lab1_Part2_QuadratureModulator.mdl”.



The block called Baseband Generator is used to produce random sequences for the I and Q channels. You will need two of this block in your simulation.

Tips: You can perform copy/paste in Simulink. You can also select and drag a component while holding the right button of the mouse to make a replicate of any component.

- Computers can only generate Pseudo-Random numbers from a seed. Make sure that the seed for the two sequence generators are different (independent streams).
- Set the symbol rate of both generators to 1 Mbps.
- Set the value for the roll-off factor to 0.35.
- Connect the ideal local oscillator block, two mixer elements and one adder to form the quadrature modulator.
- Set the local oscillator frequency to 2 MHz.
- Connect the spectrum analyzer to the output of you modulator.
- Comment-out all unconnected components.
- Run the simulation.

Task 2-1: Explain how you should choose the order of modulation on each of the I and Q paths in order to generate a 16QAM signal

Task 2-2: Describe what you observe on the spectrum analyzer view. What is the bandwidth of this signal and how is that related to the bandwidth of the baseband signal in Part1?

- In addition to the spectrum analyzer, also connect the output of the modulator to the ideal RX to check the constellation diagram of the signal. Set the frequency of the ideal RX to the same carrier frequency (2 MHz).

Task 2-3: Decision regions are hypothetical areas that the detector defines to interpret each symbol. Can you define decision regions for each symbol in the 16QAM constellation?

- Replace the ideal local oscillator with the block called, “local oscillator with phase noise and imbalance”
- Set the phase noise level to -100 dBc (practically no phase noise), k (amplitude imbalance) to 1, and θ (phase imbalance) to 0
- Double click on the LO block while the simulation is running, have both spectrum analyzer and constellation diagram measurement windows open and start to increase the phase noise level.

Task 2-4: How is the spectrum and constellation diagram affected by the phase noise?

Task 2-5: At which phase noise level do you notice that symbols are not anymore bounded to the decision areas you defined in Task 2-3? (less negative number gives higher noise.)

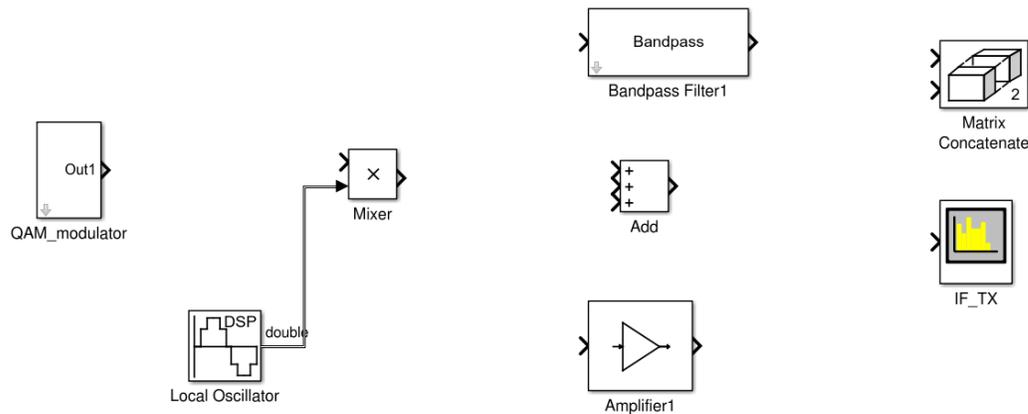
Task 2-6: Reduce the modulation order to QPSK and repeat to examine the effect of phase noise. Which of the two formats are more sensitive to phase noise? Why?

- Reduce the phase noise level back to -100 dBc and start varying the values of amplitude and phase imbalance.

Task 2-7: Which of the two imbalance parameters are more serious in your opinion? Why?

Part 3: Two-Step Up-Conversion Transmitter

In this part you will put together a complete transmitter chain. Open the Simulink model file “Lab1_Part3_TwoStepTX.mdl”.



The block called QAM_modulator is basically the same as what you made in the previous section of the lab. The output is a QAM modulated signal with a given symbol rate, pulse shape and carrier frequency.

- Set the symbol rate to 1 MSym/sec, roll-off to 0.3 and carrier frequency to 2 MHz.
- Use the Local Oscillator block to up-convert this signal to final frequency.
- Connect the spectrum analyzer to the output of the mixer.
- Comment-out other unused components.
- Run the simulation.

Task 3-1: What should be the frequency of the local oscillator if the output desired frequency is 10 MHz?

Task 3-2: Describe the spectrum of the transmitted signal. What are the different signals and how they have ended up at these frequencies?

- Use the bandpass filter to remove everything except the signal at 10 MHz.
- Double click on the filter block and specify the edges of the passband and stopband (keep the input Fs parameter to 40). You can always check the response in the graphical view. Once you are satisfied with the filter, simulate your TX with this filter.

Task 3-3: Can you explain how the filter has affected the output spectrum?

- Your transmitter is supposed to send three channels similar to this one. Place three copies of the QAM modulator and use the three-way add component to add the outputs. Connect the output of the adder to the mixer for up-conversion.

Task 3-4: Given the bandwidth of each channel choose a suitable range of carrier frequencies for the three channels. Once you have made your choice simulate the transmitter and analyze the output spectrum. You may need to iterative a few times to set the carrier frequencies and the filter edge frequencies to obtain an acceptable response.

- Add the amplifier block after the filter, check that the Linear mode is selected.

Task 3-5: How does the spectrum look like before and after the amplifier?

Tips: In order to examine spectrum of two signals on the same spectrum analyzer, use the Matrix Concatenate component.

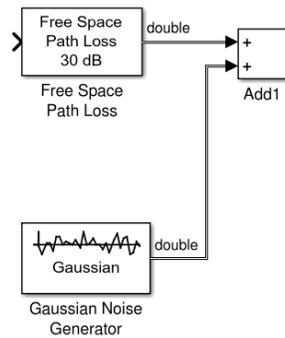
- Double click on the amplifier and select the Cubic Polynomial mode instead of the linear. Set the IIP3 to 10 dBm (OIP3 is 30 dBm with the given gain). Run the simulation again and comment on the output spectrum.

Task 3-6: Can you explain what you see on the spectrum analyzer? What could be the reason for this behavior?

Task 3-7: Copy the same bandpass filter and replace it at the output of the amplifier as well. How can this filter improve the output signal?

Part 4: Heterodyne Receiver

In this part of the lab you will configure the heterodyne receiver. Open the Simulink model called “Lab1_Part4_HeterodyneReceiver” and copy your transmitter components into this model.



- In order make things easier, revert the transmitter amplifier to the linear configuration again.
- Add the channel model with 40 dB attenuation and add the noise with variance of $200E-6$.

Task 4-1: Examine the spectrum of the incoming signal to the receiver with a spectrum analyzer. Can you identify the signals from the noise? What is the signal to noise ratio? Do you think the signals can be recovered?

- The most fundamental tasks of the receiver are filtering and amplification. You need to place a similar filter and amplifier at the input of your receiver (you can copy and paste the same components from the output of the transmitter). Set the amplifier mode to linear and set its noise figure to 4 dB.

Task 4-2: In which order would you place the amplifier and filter combination? Motivate your answer. Look at the spectrum before and after the filter-amplifier combination.

- Copy a mixer and local oscillator component from the transmitter chain and place them properly after the front-end in order to down-convert the signal to an IF.

Task 4-3: The receiver should work with an IF of 3 MHz. How would you choose the local oscillator frequency. Motivate your choice.

- Examine the spectrum of the signal after down-conversion.

Task 4-4: Place a bandpass filter after the mixer. The task of this filter is to select the desired channel. Set the passband and stopband frequencies of this filter (note: do not change the input F_s parameter).

Task 4-5: Plot the spectrum of the signal after the mixer before and after the filter (tips: use the Matrix Concatenate component). Explain what you observe and how the signal is varied after each stage. You may need to correct the design of the filter to get an optimum response.

- Once you are satisfied with your received signal, you may choose to amplify and boost its level. Set the linear gain to 30 dB and noise figure to 10 dB of the receiving amplifier.

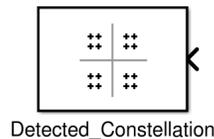
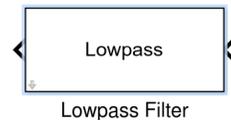
Task 4-6: Plot the spectrum of the received signal. You are obviously receiving only one of the three channels. What would you change in your receiver to receive the other channels? Make this modification and verify that the output spectrum corresponds to another channel.

Task 4-7: Amplifiers are added to compensate for the signal loss during transmission. Increase the channel loss by 10 dB. Which of the two amplifier gains would you prefer to change? Motivate your answer.

Task 4-8: Keep on increasing the channel loss and compensating by amplifier gain. How is the output spectrum affected? Check the SNR of the signal after the channel model. Is the receiver still operating as it should?

Part 5: Demodulation

In this part you will implement the final part of the link which is demodulation of the signal. Open the Simulink model file “Lab1_Part5_demodulation.mdl” and copy your entire TX-Channel-RX into this model.



The IF signal at 3 MHz should be down-converted to zero for detection. We may lose important signal information if we just perform an ordinary down-conversion (phase information will be lost). We should therefore perform a quadrature demodulation to get the in-phase and quadrature components and be able to recover both amplitude and phase of the signal.

Tips: In Simulink you may perform the quadrature demodulation by using a complex signal local oscillator.

Task 5-1: Use a mixer and a local oscillator to demodulate the IF signal output from the receiver. Choose the correct frequency for the local oscillator so that the signal is down-converted to zero. Also make sure to set the output complexity of the signal to “complex” for proper quadrature demodulation. Add a low-pass filter after the demodulator. How should you choose the edge frequency of this filter? (do not change the input F_s parameter)

Task 5-2: Connect the constellation diagram viewer component to output of your demodulator. Describe what you observe.
(This one may be hard to get working, but adjusting filter settings (less sharp) or replacing some of the bandpass filter with a lowpass filter may get it to work.)

Task 5-3: A constant phase error in the reference signal for the demodulator can cause error in detection. Run the simulation again and while it is running double click on the final local oscillator used in the demodulator and adjust the phase offset variable until the received constellation looks like what it should.

Task 5-4: Describe how the constellation diagram of the demodulated signal compares to the transmitted signal you simulated in the first part of the lab? Are received signals still within the decision boundaries you defined?

Task 5-5: Increase the path loss and compensate for it by increasing the amplifier gain of the receive amplifier (similar to what you did in the previous section). How is the constellation diagram affected by this? Can you always compensate the loss by amplification? Explain your observation.

Task 5-6: Revert the channel loss and gain to values where the constellation diagram looks acceptable. Now increase the noise figure of the amplifier and see how the constellation diagram is affected. Describe your conclusion.

Task 5-7: Again set the noise figure to an acceptable value and now set the TX amplifier to nonlinear mode. Describe the effect of varying the IP3 to the detected constellation.

You have successfully passed the lab!

Date:

Lab assistant: