M3 Testing Procedures (2024)

Prof. S. V. Hum

SDR Testing Board

To test your subsystem, you will make use of the ECE295 SDR Test Board (also known as the main board), which is discussed in the ICD. A diagram of the main board appears in Figure 1. This is a large board which has connectors that mate with each of the subsystems. It not only powers and allows each of the subsystems to be connected to each other, but also “breaks out” relevant signals to BNC connectors located on the edge of the board. This allows individual subsystems to be plugged into the board and tested, with the connectors acting as either stimulus inputs to the subsystem, or outputs facilitating easy connection to test equipment.

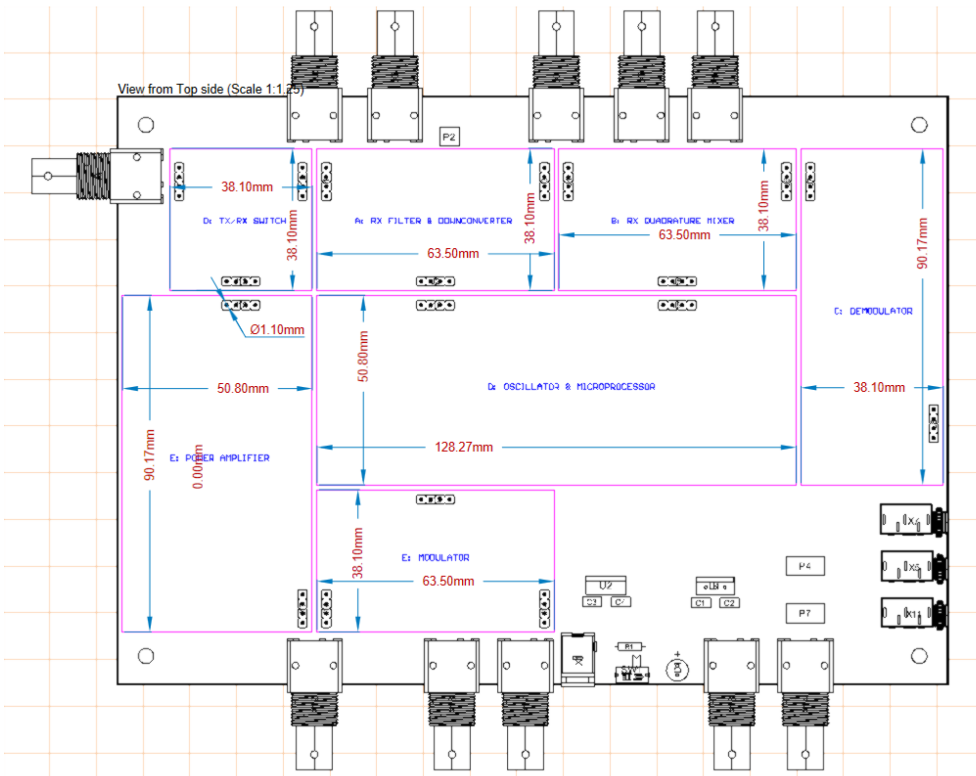

Figure 1: ECE295 SDR Test Board

The main board is available in the MY435 laboratory to be checked out and used. We now outline the important features of this board.

- Power to the board is supplied by a 2.1mm barrel jack at the bottom centre of the board (X1). You must obtain a 2.1mm coaxial cable from the lab staff in order to connect X1 to a power supply. **The polarity of the connection is extremely important**: if you look carefully at the end of the cable with the bare wire ends, you will see that one wire has a white stripe on it, while the other does not. **The wire with the white stripe is connected to the centre pin of the cable,**

which is the positive terminal: connect it to the positive output of Output 2 of the EDU36311A DC power supply. Output 2 should be set for 12V output. **If you connect this cable backwards, the main board will be damaged and rendered unusable**.

- There is a switch SW1 that is used to turn power on and off to the board, along with an LED D1 that indicates if the board is receiving power or not.
- At the bottom right corner of the board are three 1/8" audio jacks.
    - X7 (DEMOD OUT) is the demodulator output, which can be connected to a speaker equipped with a 3.5mm plug (available in MY435) or to the input of a PC sound card, using a suitable 3.5mm patch cable.
    - X5 (IQ OUT) is an additional connector providing the I/Q outputs of Subsystem A, used for systems integration testing after M3. The I/Q outputs are also available from BNC connectors X13 (RX_I) and X14 (RX_Q), so jack X5 should not be needed for M3 testing.
    - X11 (IQ IN) is for the I/Q inputs to Subsystem D. This connector is for connection to a sound card on a PC and is not needed for M3 testing. I/Q signals for Subsystem D can also be applied to the BNC connectors X17 (TX_I) and X18 (TX_Q).
    - Associated with the audio jacks are jumpers P4 and P7, which should be fitted with shorting jumpers if jacks X5 and X11 are to be used. The positions of the jumpers can be used to determine how I and Q are routed to the left and right channels of the audio connector. You should not need to do anything with these jumper blocks for M3.
- The BNC connectors are as follows:
    - X6 (ANT) is usually for connecting to an external antenna. An antenna is not used in M3. For M3, this connector can be used probe the output of the PA in Subsystem E, if the TX/RX switch is in the TX position. Since you may not have access to a TX/RX switch when doing M3 testing, you can simply insert a wire into the headers for the TX/RX switch to connect X6 to the PA output. See the specific test procedures below for details.
    - X2 (RX_SIG) is the received signal coming out of TX/RX switch when the switch is in the RX position. It can be used to stimulate Subsystem A directly without the need for the TX/RX switch in place when using X6 as the input.
    - X3 (LO_F2) is not used.
    - X4 (IF_SIG) is no longer used.
    - X9 (LO_F1_0) and X10 (LO_F1_90) are the 3.3Vpp LO signals.
        - If Subsystem C is installed on the test board, these connectors can be probed to see if the expected signals are present.
        - For other subsystems involving mixers (Subsystems A and D): with Subsystem C absent, you can apply the desired LO signals here using a pair of signal generators to test your subsystems. See the specific testing procedure for details.
    - X13 (RX_I) and X14 (RX_Q) are the I/Q outputs from Subsystem A. They can be used as:
        - Probes to check the correct operation of Subsystem A.
        - Stimulus inputs for Subsystem B if Subsystem A is absent.
    - X17 (TX_I) and X18 (TX_Q) are the I/Q inputs to Subsystem D, which you should not need in M3.
    - X12 (PA_IN) is the input signal to the PA. It can be used for two purposes:

- If Subsystem D is installed, the output of Subsystem D can be probed here.
- If Subsystem E is installed alone, this connector can be used to apply a known stimulus to the PA without the need for Subsystem D.

There is an additional component that is needed for testing Subsystem E: a 50 ohm dummy load. This is available in MY435 and resembles Figure 2.

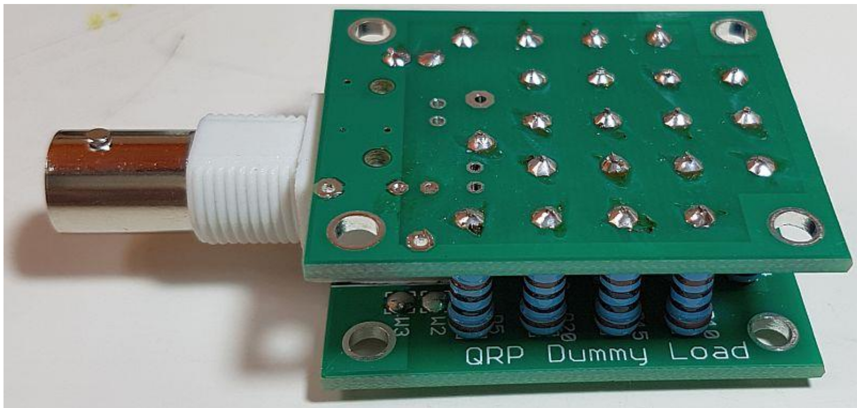

Figure 2: 50 ohm 20W dummy load

Subsystem Testing Procedures

In the subsystem testing procedures outline below, **you should only install one subsystem (PCB) in the testing board at once** (except for Subsystem C, which has two boards). These tests are written assuming all other PCBs are absent.

When doing initial subsystem testing, **you are strongly encouraged to initially set a current limit on the power supply** so that if there is a short circuit or other problem on your board, you may be able to limit the damage to your board. A current limit of 100-200 mA is sufficient for most cases except for Subsystem E, which may require significantly more current, which should be estimated from simulations. The output voltage of the power supply (Channel 2) should be set to 12V.

Detailed wiring diagrams of the tests below are available on Quercus as a separate file.

Subsystem A

Test equipment required

- Oscilloscope
- Function generator
- Power supply and coaxial power cable for test board
- Four BNC cables
- Hookup wire

Procedure

1. The built-in signal generator in the oscilloscope will act as the RF signal source. Set it up as follows. You should set the input signal to be small since you do not want to overload Subsystem B, since it has a very high gain.
    a. Signal type: sinusoid
    b. Signal amplitude: 50 mVpp
    c. Signal frequency: 14 MHz plus an offset equal to the frequency of a sinusoidal test frequency you wish to test. For example, if your message frequency is 1 kHz, the signal frequency should be set to 14.001 MHz.
2. You may first want to probe the output of the signal generator on the oscilloscope prior to connecting it to the testing board, to be sure you are getting the correct signal going into the board.
3. Disable the function generator output and connect the RF signal source to the board.
4. The dual output function generator will be used to generate the LO signals: one output for LO_F1_0 and the other for LO_F1_90. The settings on each channel are as follows:
    a. Output termination: High-Z (**it is very important to set this first, since all the parameters below will not be set properly if you do not**)
    b. Frequency: 14 MHz (or whatever LO frequency you would like to test). You should NOT add the message signal frequency to this frequency value.
    c. Waveform: Sine wave (**even if you want a square wave you <u>must</u> select a sine wave; the function generator cannot generate non-sinusoidal signals with a frequency higher than 10 MHz**).
    d. Amplitude: 3.3Vpp
    e. DC Offset: 1.65V – this will create a 0-3.3V amplitude signal with the above.
    f. Phase: 0 degrees for channel 1, -90 degrees for channel 2.
5. Connect both signal generators to an oscilloscope using identical BNC cables. Phase-synchronize the outputs of the function generator (refer to SL2 on how to use the internal sync mechanism on the function generator) and check on the oscilloscope that they are 90 degrees out of phase with each other.
6. When you are happy with the two signals, disable the function generator outputs and connect them to X9 and X10 on the test board.
7. Connect the outputs of Subsystem A, X13 and X14, to oscilloscope channels 1 and 2.
8. If your board makes use of the /TXEN signal, you will need to set it low manually, since Subsystem C is absent. You can do this by using a piece of hookup wire. Refer to the wiring diagram. Plug in one wire end to the left-most pin of J9; this is the /TXEN signal. Plug the other end into the second pin of

J10 from the left; this is +3.3V, which should set the /TXEN signal high, thereby enabling receive mode.

9. Power up the test board an ensure the current draw seems reasonable. The subsystem should not output any signal with all the function generator outputs disabled.
10. Enable the two function generator outputs responsible for LO signal.
11. Enable the signal generator responsible for the RF signal.
12. Observe the output on the oscilloscope. If everything is working, you should have a signal whose frequency is equal to the offset (message) frequency. If not, it is possible you have the phase relationship between the two LO signals backwards, e.g. one is lagging by 90 degrees when it should be leading.
13. Optional: try testing your board with the /TXEN signal low (transmitter enabled, receiver disabled). You can do this by plugging in the /TXEN wire into the left-most pin of J10, which is +GND.

Questions relating to the specifications of the ICD:

1. What is the total gain from the RF signal to the IF signal?
2. How correct are the outputs?
3. What is the bandwidth of the RX filter?
4. What is the bandwidth of the post-mixer LPF?

List of Measurements to Acquire to Reconcile with Requirements

- ☑ Oscilloscope trace(s) showing the I/Q signals obtained at the output, including measurements of:
  - Amplitude balance between I/Q
  - Phase difference between I/Q
- ☑ Oscilloscope traces or measurements showing the conversion gain of the entire subsystem, input-to-output
- ☑ Plot showing the amplitude response and phase balance of the mixer as a function of RF frequency, which can be acquired using the Python script `sub-a-bpf.py`.
- ☑ Plot showing the magnitude response of the I/Q outputs as the message signal frequency is varied, which can be acquired using the Python script `sub-a-mixer.py`.
- ☑ Optional: operation of the mixer when /TXEN is set to transmit mode

NOTE: It is not possible to test the limiter in subsystem A, since it requires a large signal be injected into Subsystem A, which would overwhelm the receiver amplifier.

Subsystem B

Test equipment required

- Oscilloscope
- Function generator
- One stereo plug to RCA cables
- Three BNC cables
- One BNC-to-RCA adapter
- BNC T-junction
- Speaker (available in MY435)

Procedure

SSB Demodulation Test

Setup note: Channel 1 of the function generator synthesizes the I signal and channel 2 the Q signal. In the wiring diagram, channel 1 is split and fed into channel 1 of the oscilloscope to be used as a triggering source when automated measurements are completed. All measurements are carried out on channel 2 of the oscilloscope, though you can observe the I channel on channel 1 if you wish.

1. If your subsystem supports it, choose USB demodulation mode.
2. The function generator is used to generate the I/Q signals required for the demodulator. One channel is for I and one for is for Q. The settings on each are as follows:
    a. Output termination: High-Z (**it is very important to set this first, since all the parameters below will not be set properly if you do not**)
    b. Frequency: The frequency of the message signal you would like to test. For example, 1 kHz
    c. Waveform: Sine wave
    d. Amplitude: 100 mVpp (adjust as needed)
    e. Phase: 0° for channel 1, ±90° for channel 2 depending on which SSB modulation you want to simulate (-90° for USB, +90° for LSB).
3. First connect both signal generator outputs to an oscilloscope using identical BNC cables. Phase-synchronize the outputs of the function generator (refer to SL2 on how to use the internal sync mechanism on the function generator) and check on the oscilloscope that they are ±90 degrees out of phase with each other. **This is very important; if the outputs are not 90 degrees out of phase, these tests won't work.**
4. Disable the function generator outputs. Connect the Subsystem B to the oscilloscope and function generator using the cables as shown in the wiring diagram.
5. Enable the function generator outputs. You should observe a single-frequency sine wave at the output of Subsystem B. If the amplitude (envelope) of the AC waveform is not constant, you likely are getting both the upper and lower sideband superimposed on top of each other, and you will have to debug the problem.
6. Try changing the frequency of the message signal and see how your subsystem responds. Every time you change the frequency, you will have to phase-synchronize the outputs of the function generator as you did in step 3.

7. If you are satisfied with the signal, let's see how your subsystem behaves with a LSB signal instead of USB signal: either change the mode of your demodulator to LSB (if supported), or if your subsystem only supports USB, reverse the sign of the 90 degree phase shift between the function generator outputs. You should not see any output from your subsystem (or it should be very small). If the behaviour is backwards, you likely have the phase relationship backwards for the input signals (leading versus lagging).
8. Try changing the frequency of the message signal and see how your subsystem responds. Every time you change the frequency, you will have to phase-synchronize the outputs of the function generator as you did in step 3.
9. When you are happy with the behaviour of your subsystem, you can disable the function generators and connect a speaker to jack X7, and configure the function generator to generate a USB signal.
10. Enable the function generators. You should hear a tone from the speaker at the message frequency you chose with the appropriate modulation configured.

Questions to consider relating to the specifications of the ICD:

1. Is the modulator output correct?
2. What is the bandwidth of your demodulator in USB mode?
3. What is the ratio of the desired sideband amplitude to the undesired sideband amplitude (i.e. the *sideband rejection ratio*) of your demodulator in SSB mode?

List of Measurements to Acquire to Reconcile with Requirements

- ☑ Oscilloscope trace(s) showing the demodulated signals obtained at the output for
    - SSB-USB signals
    - SSB-LSB signals
- ☑ Oscilloscope traces and measurements illustrating the SSB sideband rejection ratio for a single message frequency.
- ☑ Plot showing the measured amplitude of the signal as a function of message signal frequency, for all demodulator modes. This can be acquired using the Python script `sub-b.py`
- ☑ Plot showing the sideband rejection ratio, for LSB and USB demodulation. This can be acquired using the Python script `sub-b.py`
- ☑ Be prepared to show the subsystem working with the speaker.

Subsystem C

Test equipment required

- Oscilloscope
- Multimeter
- Power supply and coaxial power cable for test board
- Two BNC cables
- Two BNC-to-alligator cable
- Test leads for the multimeter
- USB cable to connect UART to PC

Procedure

Local Oscillator

1. Connect the oscillator outputs X9 and X10 to the oscilloscope channels 1 and 2 respectively, using identical BNC cables.
2. Connect your Subsystem C board to a PC using the USB-to-serial adapter.
3. Power up the test board an ensure the current draw seems reasonable.
4. Command your subsystem to produce a 14 MHz LO signal using your front-panel interface.
5. Check that you have two signals 90 degrees out of phase on the oscilloscope. You can check the frequency of the signal, as well as the phase difference between Channel 1 and Channel 2 using the Measure key. Do not worry if the amplitude of your clock signals is not exactly 3.3 Vpp.
6. Repeat steps 4-5 using suitable CAT commands issued from the PC.
7. Power down the test board.

TX/RX Switch (DC continuity)

If you used a relay or other electromechanical switch to implement the TX/RX switch, use this test and the first wiring diagram for C.2.

1. Connect one BNC-to-alligator cable to X6.
2. Connect the red alligator clip to the VΩ input on the multimeter using a test probe.
3. Connect one BNC-to-alligator cable to the X2.
4. Connect the red alligator clip to the LO input on the multimeter using a test probe.
5. Put the multimeter in Continuity Test mode (Cont key).
6. Power up the test board.
7. If your subsystem is configured to be in the RX state by default, the multimeter should signal continuity between the probes with an audible beep. If the subsystem is in the TX state by default, there should be no continuity.
8. Command your subsystem to go into TX mode using your front-panel interface and check for correct operation.
9. Repeat using suitable CAT commands issued from the PC.
10. If you wish, you can also check for continuity between the ANT port (X6) and the PA output from Subsystem E via J18. Refer to the ICD for the pinout of this connector.

TX/RX Switch (RF continuity)

If you used a PIN diode or other electronic switch to implement the TX/RX switch, use this test and the second (alt) wiring diagram for C.2.

1. Connect the oscilloscope function generator to X6 using a BNC cable.
2. Connect oscilloscope CH1 to X2 using a BNC cable.
3. Set the function generator to generate a 14 MHz sine wave with 1 Vpp amplitude.
4. Power up the test board.
5. If your subsystem is configured to be in the RX state by default, the oscilloscope should show the sinusoidal signal from the function generator. If the subsystem is in the TX state by default, there should be no signal.
6. Command your subsystem to go into TX mode using your front-panel interface. There should be no signal displayed on the oscilloscope.
7. Repeat using suitable CAT commands issued from the PC.

Questions to consider relating to the specifications of the ICD:

1. Are the LO outputs of the subsystem correct?
2. Can you set the frequency properly using front panel and remote interfaces?
3. Does the TX/RX switch respond correctly to front panel / remote commands?

List of Measurements to Acquire to Reconcile with Requirements

☑ Oscilloscope trace(s) showing the LO signals, including measurements of the following for various frequencies requested from the module. The Python script `sub-c.py` can assist with this when using the computer to control the frequency using CAT.
  - Amplitude balance of the two LO signals
  - Phase difference between two LO signals
  - Frequency of the generated signals

☑ Be prepared to demonstrate:
  - How the LO frequency is changed using your front-panel user interface
  - Correct operation of the TX/RX switch when controlled by either the front-panel interface or the computer interface
  - Correct operation of the “IF” command on the serial terminal

Subsystem E

Test equipment required

- Oscilloscope
- Power supply and coaxial power cable for SDR board
- Two BNC cables
- 50 ohm dummy load
- BNC T-junction
- Male/male BNC adapter
- Hookup wire
- Thermal camera (optional)

Procedure

NOTES BEFORE YOU BEGIN:

- It is very important that a 50 ohm load be present at the output of your PRA during tests, or else your power amplifier may be damaged! This is included in the steps below. Do not run your PA without the load attached.
- The transistors in your output stage may become hot during operation. Ensure you have attached appropriate heat sinks to them and do not touch them.
- The 50 ohm load will become hot when your PA is delivering power to them. It is designed to handle up to 20W of power. Do not touch it.

1. As there is no TX/RX switch installed, you need to hard-wire the ANT connector (X6) to the output of the power amplifier so that you can observe the signal there. You can do this using the female connectors on the test board that normally mate with Subsystem D.2 (TX/RX switch). Use a short piece of solid-core 22 AWG wire to connect the right-most pin of J13 to the bottom pin of J11, as shown in the wiring diagram.
2. Connect a BNC T-junction to X6.
3. Using the male/male BNC adapter, connect one port of the T-junction to the 50 ohm dummy load. **It is very important that this load be present during tests, or else your power amplifier may be damaged!**
4. Configure the function generator on the oscilloscope (WaveGen) as follows:
    a. Output termination: High-Z (**it is very important to set this first, since all the parameters below will not be set properly if you do not**)
    b. Frequency: 14 MHz (or whatever RF frequency you would like to test)
    c. Waveform: Sine wave
    d. Amplitude: 1 Vpp
5. Connect the function generator temporarily to Channel 1 of the oscilloscope using a BNC cable, enable the output, and ensure the waveform is as you expect.
6. Disconnect the BNC cable from Channel 1 and connect it to the PA input X12.
7. Using another BNC cable, connect the other port of the T-junction to Channel 1 of the oscilloscope.

8. If your board makes use of the /TXEN signal, you will need to set manually, since Subsystem C is absent. You can do this by using a piece of hookup wire. Refer to the wiring diagram. Plug in one wire end to the left-most pin of J9; this is the /TXEN signal. Plug the other end into the second pin from the left of J10; this is +3.3, which should set /TXEN high and disable your PA.
9. Power up the test board current draw seems reasonable with your PA disabled. The subsystem should not output any signal with the function generator output disabled.
10. Now set the /TXEN signal low by plugging into the left-most pin of J10 (GND). Determine if the test board current draw seems reasonable with your PA idle (no input signal). The subsystem should not output any signal with the function generator output disabled.
11. Enable the function generator output.
12. Observe the resulting waveform on the oscilloscope. It should be a nice sinusoidal signal at 14 MHz. If not, adjust the amplitude of the input.
13. Calculate the power delivered to the 50 ohm load and determine if this is what you expect. **Caution: the dummy load will get very warm for large output powers from the PA (more than 1 W)**.
14. Using the FFT function of the oscilloscope, measure the amplitudes of the various harmonics (at least to the $n = 5$ harmonic) for various output powers. See the Appendix for more information on using the FFT function.
15. Optional: check out the temperatures on your board using the thermal camera.

Questions to consider relating to the specifications of the ICD:

1. What is the power output of the PA?
2. Does it respond correctly to the input signal?
3. What is the efficiency of the PA?
4. What is the THD of the PA?

List of Measurements to Acquire to Reconcile with Requirements

- ☑ Oscilloscope trace(s) showing the input and output signals
- ☑ Current and corresponding power consumption calculations of your board when amplifying. If you make use of the /TXEN signal, take note of the power consumption when this signal is high (PA disabled) and low (PA enabled).
- ☑ The calculated efficiency of your amplifier.
- ☑ FFT measurements including harmonic amplitudes up to $n = 5$
- ☑ Total harmonic distortion (THD) measurements carried out using measurements using the FFT. This can be acquired using the Python script `sub-e.py`
- ☑ Your assessment of the maximum power output of the PA given requirements
- ☑ Optional: thermal image of your PCB

Appendix A: FFT Measurements

The Fast Fourier Transform (FFT) is a powerful function on the oscilloscope that enables you to visualize the frequency spectrum of your input signal. While turning on and off the FFT function is easy (just use the FFT key on the oscilloscope), setting up the timebase and FFT settings for a meaningful measurement is another matter. This appendix discusses how to properly set these.

When you enable an FFT measurement, you must first select which Source you want to take the FFT of (Channel 1 or 2). Be sure to set this appropriately using the menu keys.

The frequency range of the FFT is determined by setting the *center* frequency and frequency *span* of the FFT. These are also set in the FFT menu using the menu keys and the knob. The FFT will determine the spectrum of the input signal between frequencies f_1 and f_2. The center frequency and span are related to f_1 and f_2 through:

$$\text{center frequency} = \frac{f_1 + f_2}{2}$$

$$\text{span} = f_2 - f_1$$

Inverting this relationship,

$$f_1 = \text{center frequency} - \frac{\text{span}}{2}$$

$$f_2 = \text{center frequency} + \frac{\text{span}}{2}$$

Clearly, the center frequency and span are set very differently, depending on what you are measuring. If you are looking at a modulated RF signal (e.g. output of Subsystem E), you only care about the carrier frequency and a relatively narrow band (span) around that frequency, perhaps only a tens of kHz wide, depending on the bandwidth of the message signal. On the other hand, if you are looking at the harmonics produced by mixer or power amplifier, then you are looking for frequencies at integer multiples of the carrier frequency, resulting in a very wide frequency range spanning many tens of MHz.

This is where the timebase (sec/div) setting of the oscilloscope comes in. Since the oscilloscope samples signals at a fixed rate (2 GSa/s), the timebase must be set so that the scope acquires a sufficiently long measurement (in seconds) to capture enough variation of the signal to provide usable FFT information over the frequency span you have requested. If the timebase is not set correctly, you will not have enough resolution in the FFT to see individual frequency components. The FFT may even not show any useful information at all, if the timebase is not set correctly.

Example 1: FFT of Modulated RF Signals

Consider an RF carrier signal at 14 MHz, AM modulated with a message signal at a frequency of 5 kHz. From our understanding of AM, we only expect frequency components at 13.995 MHz, 14 MHz, and 14.005 MHz at the output of an AM modulator. If the modulation depth is 100%, the 14 MHz carrier is suppressed, and only two frequency components will remain.

To see several periods of the AM signal, we actually need to set the timebase according to the period of the message signal, **not** the period of the carrier signal. The message signal has a period of $T = \frac{1}{5000} = 200\ \mu s$. Usually, we want to display several periods of a periodic signal on the oscilloscope screen. Say we want to show 5 periods. This would require us to acquire $5 \times 200 = 1000\ \mu s$ or 1 ms of samples. Since there are 10 time divisions on the oscilloscope screen, this requires a timebase setting of $\frac{1000\ \mu s}{10} = 100\ \mu s$/div. We can then set the FFT to compute the spectrum centered at 14 MHz and with an arbitrary span of 50 kHz, which should be more than enough to see our two frequency components at 13.995 MHz and 14.005 MHz. The result is shown in Figure 3.

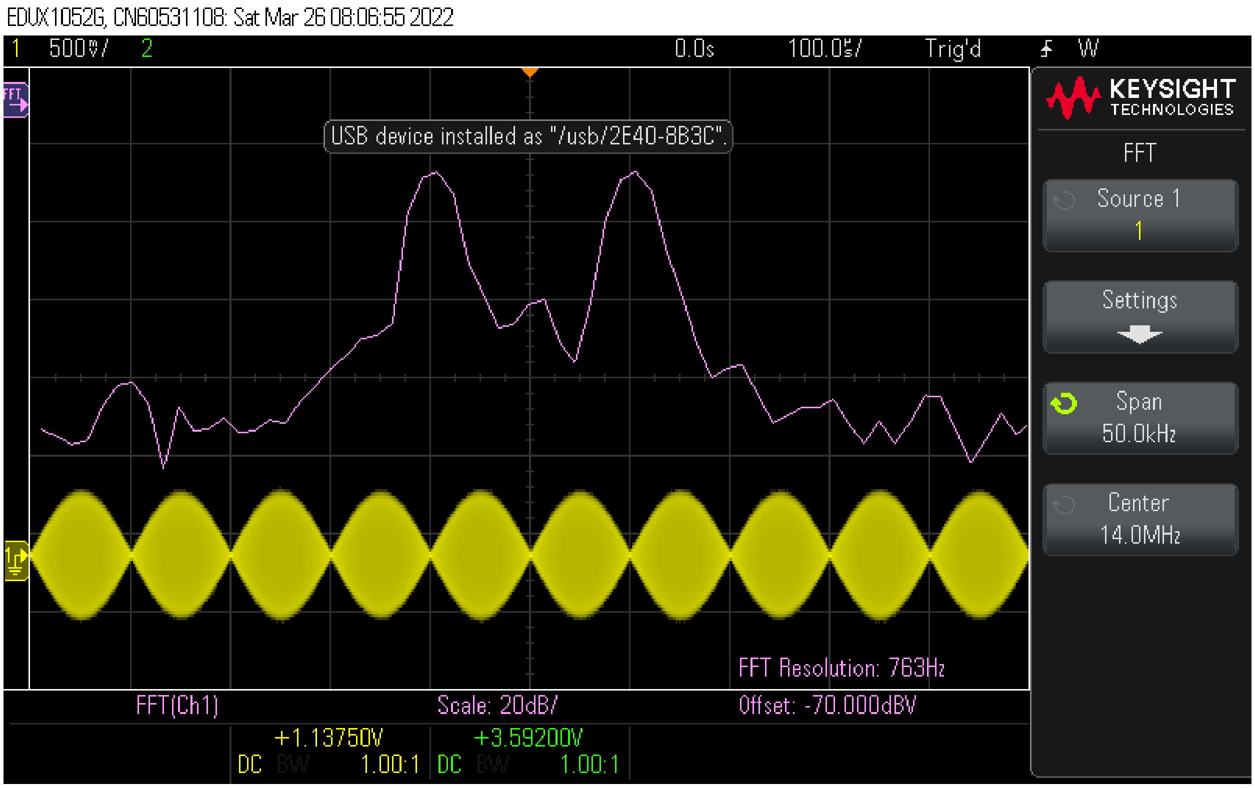

Figure 3: FFT of an AM modulated signal

As expected, we see two spikes in the frequency spectrum. If we use the cursors, we can see that they peak at the expected frequencies. One is the lower sideband and the other is the upper sideband. You may wonder why they are not very “spikey” or narrow like the delta functions we expect. The reason is that the FFT does not have infinite frequency resolution, because it depends on how many time samples were acquired to compute the FFT, which we have limited control over. Here the FFT resolution is reported to be 763 Hz; that means that we can only resolve frequency components more than 763 Hz apart from each other. Given that the “distance” between 13.995 MHz and 14.005 MHz is only 10 kHz, that’s only 13 frequency points separating the upper sideband and lower sideband in the plot; and remember the plot is on a dB scale (with a whopping 20 dB per division), so all things considered, the two frequency spikes are fairly close to delta functions given the limited resolution of our measurement.

Example 2: Harmonic Spectra of an CW Signal

In the second example, we assume that a single-frequency signal (also known as a continuous wave or CW signal) has gone through a nonlinear device like a mixer or power amplifier, resulting in the production of harmonics. If the fundamental frequency is 14 MHz, we expect harmonics at 28 MHz, 42 MHz, 56 MHz, etc. The frequency span we want to measure over is quite wide. The considerations for the timebase are totally different when the expected frequency span is very wide.

In this case, the timebase setting plays a strong role in setting the frequency resolution of the FFT (the spacing between the frequency points), because the timebase directly determines how many data samples are acquired by the scope for the FFT. The scope samples at a fixed rate of 2 GSa/s, so changing the timebase from 10 µs/div to 100 µs/div for example will result in 10 times more data samples being acquired. The number of data samples determines the frequency resolution: the larger the number of data samples (i.e. the higher the timebase setting in s/div), the greater the frequency resolution. If there is insufficient frequency resolution, it can be difficult to pick out various spectral components in the FFT.

Figure 4 shows one example. This is a distorted 14 MHz signal acquired with a timebase setting of 2 µs/div. We can see the 14 MHz fundamental on the left, a weak second harmonic signal, stronger third harmonic, and then a weaker 4th harmonic. The harmonics are hard to see, but they are resolvable. This is because the FFT resolution is only 30.5 kHz – that is, there is 30.5 kHz between individualk frequency points in the plot.

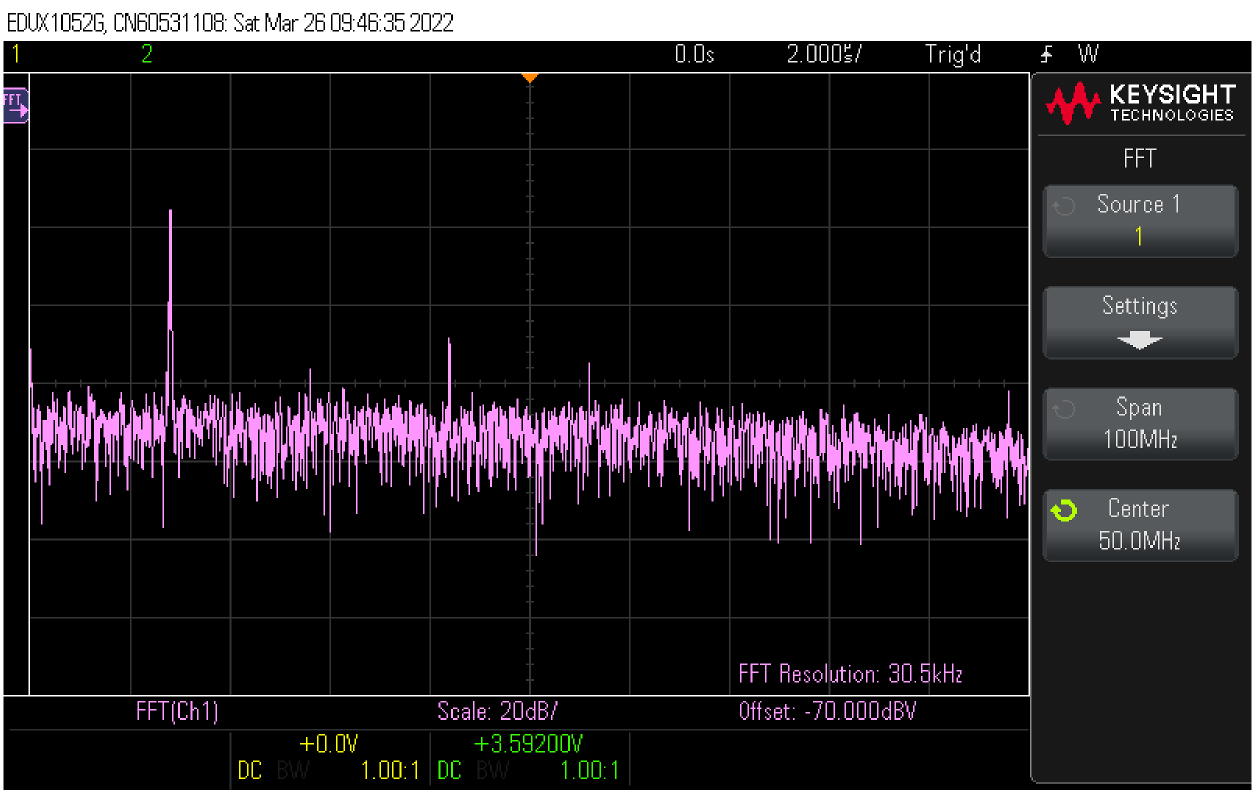

Figure 4: FFT of a distorted RF signal, 2 µs/div

When we increase the timebase to 5 µs/div, the FFT in Figure 5 results. Notice how it is easier to resolve the spectral components in the signal. The frequency resolution is also twice what is was previously:

now there is only 15.3 kHz between frequency points (the plot comprises many more points now, resulting in a more “dense” plot).

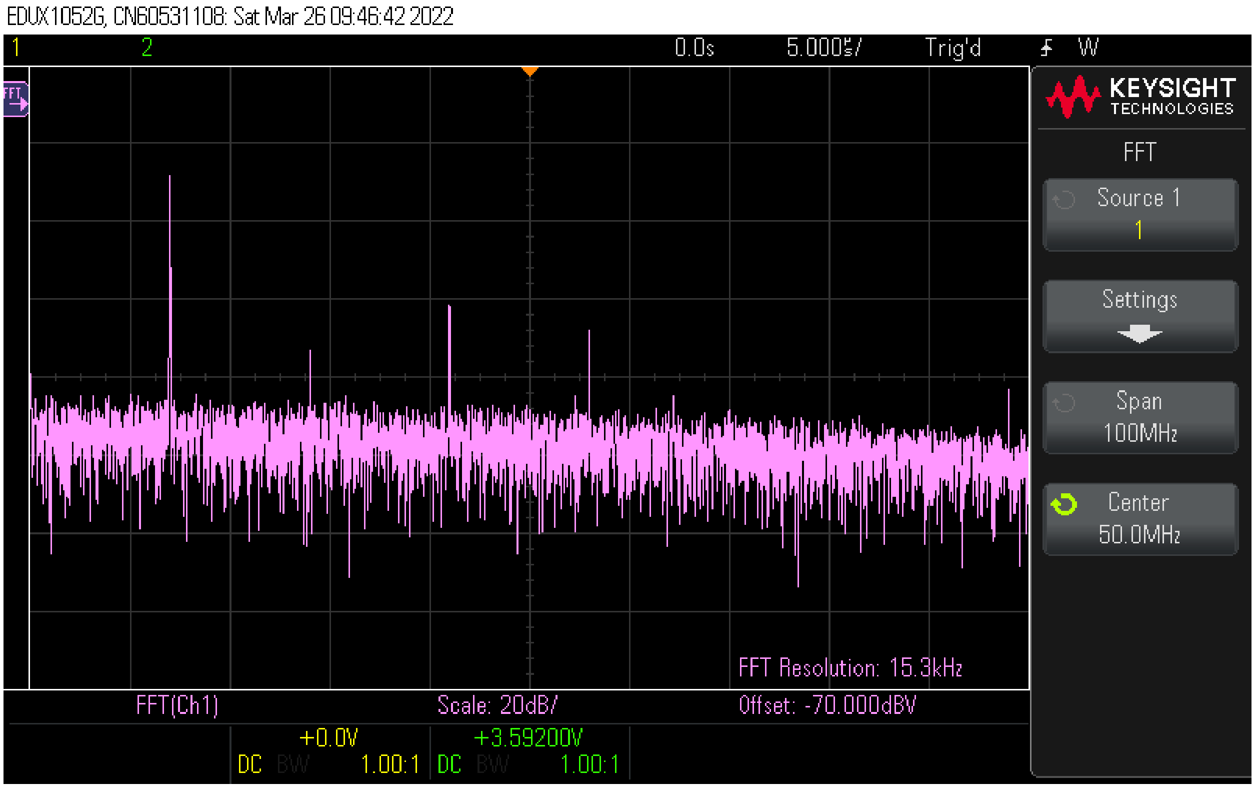

Figure 5: FFT of a distorted RF signal, 5 µs/div

Practically, both plots are usable; for example, they both show the same relative ratios of amplitudes between the fundamental signal and its harmonics. The second requires a longer acquisition time (and more memory), and so ultimately your needs will be determined by the exact requirements of your measurement.