Milestone 0: Introduction to CAD Tools, Soldering, Test and Measurement

1 Introduction

This first milestone in ECE295 is designed to familiarize you with the various computer-aided design (CAD) tools and laboratory procedures that will be crucial for you to carry out your designs in the course. In particular, this milestone has the following objectives:

1. To present a basic electronic circuit design and its theory of operation;
2. To familiarize you with schematic capture in *Multisim*, a commercial circuit simulator, for representing that circuit;
3. To teach you how to analyze the circuit using Multisim;
4. To illustrate how this circuit is implemented in a printed circuit board (PCB) in *Altium Designer*, a software package for designing PCBs;
5. To teach you how to assemble and solder a PCB;
6. To introduce measurement procedures in the lab using the circuit as an example; and
7. To teach you a little bit about voltage regulators.

The activities to complete this milestone will take place through various structured activities: two structured labs (SL1 and SL2) and two structured tutorials (ST1 and ST2). To complete the milestone, you will need to:

1. Submit the results from your circuit analysis in Multisim, documented using a worksheet (TODO); and
2. Demonstrate your assembled regulator and present measurements of the circuit interactively with your TA.

2 A Voltage Regulator Circuit

This milestone will centre around the analysis of a simple *voltage regulator* circuit. The function of a voltage regulator is to take DC voltage at its input and produce a *regulated* voltage output of a fixed value, e.g. 5 V. Ideally, this voltage output should stay constant regardless of changes in the input voltage, or changes in the load (current draw). Voltage regulators form the basis of modern power supplies, since most circuits and systems need specific voltages to function.

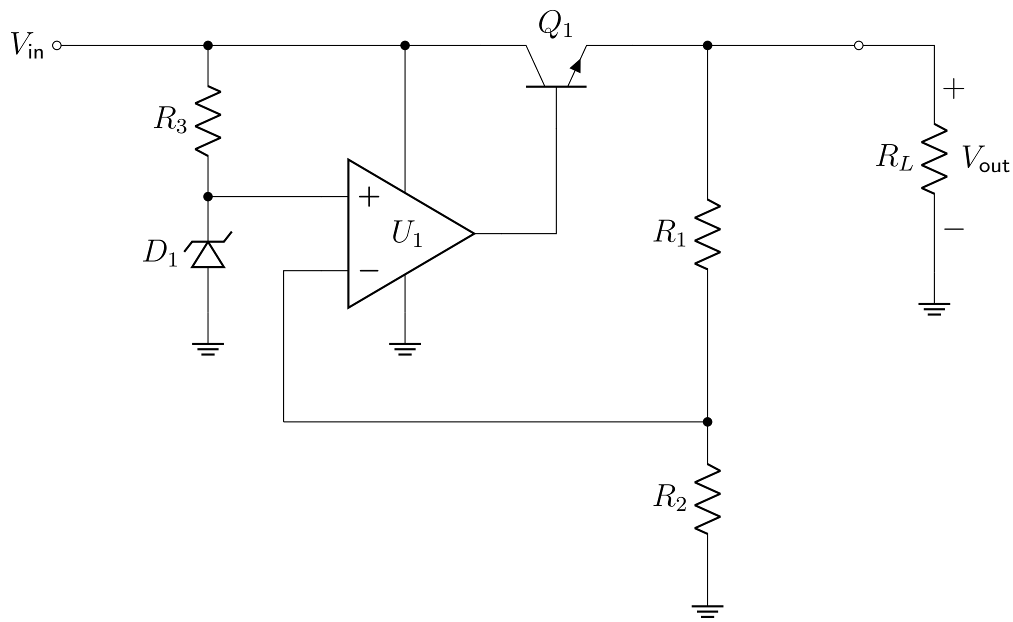

Figure 1: Transistor-based voltage regulator circuit

Figure 1 shows a schematic of a basic transistor-based voltage regulator that you will analyze in this milestone. Using your knowledge from ECE212 and ECE231, it is possible to describe the basic principles of operation of this circuit prior to undertaking a detailed analysis.

The function of the circuit is to take a variable DC input voltage V_{in} and produce a regulated output voltage V_{out} delivered to a load R_L, which itself can be variable.

Let's focus on the opamp U_1 first, since you have studied opamps extensively in ECE110 and ECE212. Here, the opamp functions as an *error amplifier*, taking the difference between the signal at the non-inverting terminal and the inverting terminal, and amplifying it to produce a signal at the output of the opamp. Ideally, the opamp has zero potential between its two inputs.

Let's designate the voltage at the non-inverting input of the opamp as V^+. The non-inverting input is connected to a *reference voltage* that is derived from R_3 and D_1. D_1 is a special kind of diode known as a *Zener diode*, which achieves reverse breakdown at a very specific voltage. In this case, let's choose a specific part for the diode, the 1N5222 Zener diode. Referring to the datasheets provided with this module, we can see that the reverse breakdown voltage of this particular diode is 2.5 V. From Figure 1, we see the diode is wired so that it is reversed biased. R_3 serves to limit the current passing through the diode, since when the diode is in breakdown, it will draw very large amounts of current which are not necessary in order to produce the reference voltage. Hence, $V^+ = 2.5$ V.

The voltage at the inverting terminal of the opamp, which we designate here as V^-, is actually a scaled version of V_{out}, where the scaling factor is determined by the voltage-divider network comprising R_1 and R_2,

$$V^- = \frac{R_2}{R_1 + R_2} V_{\text{out}}. \tag{1}$$

The resistors R_1 and R_2 are chosen to be high so that very little current flows into them compared to the current flowing into the load R_L. In this way, the voltage divider *samples* the output voltage. The opamp takes the difference between the 2.5 V reference voltage and this sample voltage to determine the output signal. We will call this voltage V_b, since it drives the base of the transistor such that

$$V_b = A_v(V^+ - V^-) \tag{2}$$

where A_v is the open-loop gain of the opamp (assumed to be very large). Let's focus next on what happens with the transistor Q_1.

You are just starting to learn about transistors in ECE231. The transistor shown here is a *bipolar junction transistor*, or BJT for short. It is a NPN type, referring to the fact that it is realized from a p-type semiconductor (for the base) sandwiched by two n-type semiconductors (making up the collector and emitter). A transistor can be thought of as a kind of valve, controlling the current flowing from the collector to the emitter. This current can be controlled by a very small current that flows into the base of the transistor. In this way, a small signal (the base current) can control a very large current (the collector current). While the base current adds to the collector current to give the emitter current (because of Kirchoff's current law), the base current is often so small compared to the collector current that we say the collector current is approximately equal to the emitter current. This is because of the current gain (h_{FE}, also symbolized as β) of the transistor, is usually high. The base current is controlled by the output of the opamp: the larger the voltage V_b produced at the output of the opamp, the larger the voltage applied to the base of the BJT, and the more current that is allowed to flow from the collector to the emitter. Placing the BJT in this configuration makes it a *series pass configuration*, since the BJT appears in series between the input and the output of the circuit.

Looking carefully at the circuit, we see that the opamp and transistor are set up in a *feedback configuration*. Let's think about what happens. For simplicity, let's assume that $R_1 = R_2$, meaning that V^- is half of V_{out} according to equation (1). The reference voltage is 2.5 V, so if the V^- is larger than this value (meaning $V_{\text{out}} > 5$ V), then the output of the opamp is driven more negative according to equation (2). The voltage at the base of Q_1 is reduced, which lowers the base current flowing into Q_1 and hence lowers the current passing from the collector to the emitter. Less current flows into the load, so V_{out} drops (because of Ohm's Law). In fact, the voltage will keep dropping until we reach equilibrium such that the sample voltage V^- is equal to the 2.5 V reference voltage ($V_{\text{out}} = 5$ V).

Now let's think about the opposite situation. If $V_{\text{out}} < 5$ V, then the output of the opamp is driven more positive. A larger base current flows into Q_1, which in turn increases the current flowing from the collector to the emitter and subsequently to R_L. The voltage across R_L rises, and continues to rise until $V_{\text{out}} = 5$ V.

This entire configuration is known as a *negative feedback configuration*. You have seen how negative feedback can be used to realize various opamp circuits, such as inverting amplifiers, non-inverting amplifiers, etc. This is a very similar case. By using negative feedback, we make use of the opamp's high open-loop gain A_v to force the condition that $V^- = V^+$. The action of the opamp is to control the load current via Q_1 so that this condition is achieved. Hence, no matter what we do to the input voltage V_{in}, the output will want to be 5 V if everything is working. Voltage regulation has been achieved.

In practice, this is a very basic voltage regulator, and lacking features of more modern implementations of voltage regulator circuits. For example, the breakdown voltage of D_1 is temperature-dependent (which you can see in the datasheet), so the output voltage of this regulator is thus dependent on the circuit's temperature as well. This is not desirable; we want 5 V out from this circuit regardless of the ambient temperature. The regulator also lacks any sort of protection, such as over-current protection (OCP). What happens if we short out R_L? The opamp will create a very high base voltage, which will then cause Q_1 to pull as much current as possible from the input to the circuit. With no load to dissipate the current, all the power is dissipated in the ohmic losses of Q_1, causing it to heat up and eventually thermally fail (self-destruct). This is not only bad for the voltage regulator circuit, but it can cause an unregulated voltage to be delivered to the load circuit which may not have been designed to accept more than 5 V. Thus both the regulator circuit and circuit powered by the regulator may be destroyed.

Nevertheless, this circuit is very simple and practical for building up our skills in ECE295.

3 Schematic Capture and Analysis in Multsim

To analyze this circuit more rigorously under various conditions, we will use a circuit simulator known as Multisim. The first step in the process is to capture the schematic of this circuit in Multsim. To do that, we will need some specific values for the components shown in Figure 1. Let's make selections as follows.

- The sum of R_1 and R_2 must be large so that the majority of the load current flows into R_L. Let's set $R_1 + R_2 = 10$ kΩ, and we can begin by setting the configuration such that $R_1 = R_2$ as in our analysis in Section 2.
- R_3 exists to limit the current that flows through D_1; without it, once the input voltage exceeds the reverse breakdown voltage of the diode, it would pull a large amount of current from the supply and self-destruct. A value of $R_3 = 330\ \Omega$ is sufficient to both limit the reverse current through the diode yet achieve a strong enough reverse breakdown of the diode to make the desired reference voltage appear across its terminals.
- D_1 is the 1N5222 part discussed earlier.
- U_1 can be a general purpose opamp. We will use the LM741 opamp; refer to the datasheet for more information.
- Q_1 is our series pass transistor. Because it must handle potentially large collector currents, we choose a *power transistor* to realize Q_1. Power transistors have lower h_{FE} values but support larger collector currents. We don't need a great deal of gain here because the error amplifier already provides a lot for our closed-loop feedback configuration. We choose the TIP29C power NPN transistor for Q_1. Have a look at its datasheet.
- Finally, there is the load resistance R_L. We will vary this in our experiments, but set it to $300\ \Omega$ for now.

Proceed to enter the schematic in Multsium using the values above. Refer to your tutorial materials from ST1 to do this. Don't worry about the input voltage V_{in} yet; we will place different sources here to simulate different conditions next.

3.1 DC Analysis

1. Begin by using a DC power source set to 12 V for V_{in}. Place a voltage probe across R_L and at other relevant points in the circuit that you can use to verify the operation of the circuit. Run a DC analysis simulation and verify the output voltage across the load is somewhat close to what you want. Verify that internal voltages in the circuit, namely at the opamp input terminals, are what you expect them to be.

2. Due to non-idealities in the circuit, you may have noticed the output voltage is not exactly 5 V. Make some changes to R_1 and R_2 to adjust the output so that the voltage is exactly 5 V. Ensure that the sum of R_1 and R_2 remains fixed at 10 kΩ. You will see why later. Record your values for R_1 and R_2.

3. Now, conduct a sweep of the input voltage V_{in} from $0 - 30$ V.

   (a) There is a minimum voltage difference between the collector and emitter of Q_1 for the circuit to funcion as intended. This is called the *dropout voltage*. Determine the dropout voltage of your circuit.

   (b) Graph the output voltage as a function of the input voltage to verify voltage regulation over the range of voltages exceeding the dropout voltage.

   (c) Fix the input voltage to 12 V, and sweep the load resistance from $5\ \Omega$ to $1000\ \Omega$. Plot the output voltage as a function of load resistance.

   (d) Calculate the load regulation using

   $$\text{regulation} = \frac{V_{\text{oc}} - V_{\text{fl}}}{V_{\text{fl}}} \tag{3}$$

   where V_{oc} is the open-circuit voltage of the regulator, and V_{fl} is the *full load* voltage of the regulator. This is the voltage output produced by the regulator when the load pulls the maximum current that the regulator is designed to handle. Here, we will vary the load resistance as in step 3c to determine what the voltage regulation is as a function of the load resistance. We essentially treat the load resistance at each step as "full load" and see what regulation results. The maximum collector current the TIP29C can handle is 1 A (see the datasheet), so the theoretical full-load voltage should be measured when the the load resistance is $5\ \Omega$. Determine the load regulation at this point, and also plot the load regulation as a function of load current.

3.2 Transient Analysis

Voltage regulators are often used to regulate the output of circuits that transform alternating current into unregulated DC. An example is a circuit shown in Figure 2, which uses a transformer

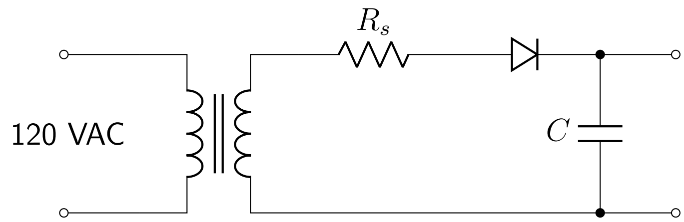

Figure 2: Half-wave rectifier circuit

to step down household AC (120 VRMS @ 60 Hz) to a lower voltage and then a half-wave rectifier to turn it into unregulated DC.

In this circuit, the transformer steps the voltage down by a factor of N, where N is the ratio of the number of turns on the primary to the number on the secondary. Let's assume $N = 13$ so that approximately 9 VRMS appears at the secondary. R_s models the resistance of the windings, which we can set to $10\ \Omega$ here. We employ a rectifier diode, such as the 1N4002, for D_1 (see available datasheet). Finally and most importantly, a filter capacitor $C = 470\ \mu$F is used to smooth the half-wave rectified voltage appearing at the cathode of D_1, so that we have a steady, but unregulated, DC voltage available at the output terminals. These values roughly correspond to a DC rectifier circuit you will use later in the lab.

1. Capture the schematic of the power supply shown in Figure 2 in Multisim.
2. Place a $300\ \Omega$ load resistance at the output and plot the load voltage as a function of time for 20 periods of the input. Observe how the voltage waveform is not steady.
3. Connect the power supply to your voltage regulator, and observe how the output voltage varies with the input. Plot the output voltage for 20 periods of the input.
4. Is there an improvement in the voltage output? What is the peak-to-peak variation of the voltage without the regulator to that with the regulator, as a percentage of the DC output voltage in each case, to quantify this.

4 PCB Design and Assembly

To this point we have engaged in the first steps of design: capturing a circuit using schematic capture, and analysis by hand or using a circuit simulator. To translate this into a physical design, we need to lay out the design on a printed circuit board onto which we can mount the various components in the design. For this milestone, you will not do this, but you will soon learn how to do this in a software package called Altium Designer.

The first step in PCB design is to capture the schematic of the circuit into the CAD package. Unfortunately, this is a bit of a repeat of what we did in Multisim, which we now need to do in Altium. This is done in a `SchDoc` file in Altium. The schematic contains the components and netlists that are used to provide an electrical description of the circuit in a PCB layout editor, which creates a `PcbDoc` file in Altium. In the layout editor, a designer places the components in the design position, defines the size of the board, and lays out printed circuit board traces (conductors) to interconnect the components the same way they appear connected in the schematic.

These steps have been already completed for the purposes of this milestone. A completed Altium design has been created. To get some practice using git, our version control tool, we will use git to download a copy of the appropriate Altium files.

1. Create a new directory to hold the Altium Designer project.
2. Open a git command window and navigate to the directory you just created.
3. Type:

```
git clone git@github.com:svhum/ece295-m0.git
```

A copy of all the files now appears in the directory. Open Altium Designer and the M0.PcbPrj file, and from there have a look at the corresponding schematic file and PCB file. Notice a few useful features in the design:

- To supply a DC voltage to the circuit, a 2.1mm coaxial power jack (J1) has been used, which is commonly used in consumer electronic equipment.
- A light-emitting diode (LED) has been added. It is a useful design practice to use indicators so that the user knows if a part is on or not.
- Instead of a pair of fixed resistors for R_1 and R_2, a *trimmer potentiometer* has been used. Potentiometers can be thought of as a fixed resistance (in this case, 10 kΩ appearing across two terminals, to which a *wiper* terminal has been added that "slides" along the resistor material so that a third connection midway between the endpoints of the resistor is possible. This allows the user to use a small screwdriver to *trim* the output voltage until the desired value is achieved. Remember how we had to tweak R_1 and R_2 in simulations to get the desired output voltage? Without the trimmer, one would have to de-solder and re-solder resistors until the desired voltage was where we wanted it.
- Some filter capacitors, C_1 and C_2 have been added at the input and output to filter out high frequency noise and signals that may reside in the input, improving the transient response of the circuit.
- A diode D_2 has been added in case the voltage supply is applied backwards to the regulator circuit, which would damage the circuit. It won't conduct current into the circuit if that happens.
- A terminal block is used at the output, so that wires can be conveniently connected to the PCB using screw terminals.

- The regulated voltage also is supplied to a pair of headers P_1 and P_2 that can be soldered onto the board so that it can plug into the supply rails on a standard prototyping breadboard.

The PCB design you see in Altium has been manufactured and supplied to you in the labs, along with the components needed to populate the board. As part of this milestone, you will solder and assemble this PCB so that you can later take measurements.

Procedure:

1. It is best to start soldering with U_1. Insert U_1 into the board, being sure to align the notch in the chip package with the notch shown on the silscreen of the board (so that pin 1 of U_1 is on the upper left). It is very important that U_1 is oriented correctly, otherwise the circuit will not work. You may need to bend the leads of the chip slightly so that the chip can be inserted into the pins. Once U_1 is in place, tape it down to the board, invert the board, and solder the 8 pins.

2. Solder capacitors C_1 and C_2. The leads do not need to be pre-formed prior to inserting these components, but it's helpful to bend the leads once a capacitor has been inserted so that it does not fall out.

3. For diodes D_1 and D_2, bend both leads 90° at appropriate positions and insert it into the board. Again, bending the leads will help the components to stay in place during soldering (tape also helps). Pay close attention to the orientation of the diode! The cathode (marked with a band on the component) must match the silkscreen on the board.

4. Next, solder the resistors R_1 and R_3 in the circuit. R_3 is soldered in a similar way to D_1. R_1 is handled differently. Bend one lead on the resistor so that it is parallel to the other lead (Figure 3). This creates a "hairpin" connection where the small U-shaped half-loop you've formed acts as a test point where for example an oscilloscope probe can easily be attached. Insert the resistor in the board so that the body of the resistor is above the lower hole and the bent lead you just made passes through the top hole. This way, voltage measurements at the hairpin will correspond to that at the cathode of D_1, which is useful for testing the circuit later. Solder the resistors in place on the solder side of the board.

5. Solder the LED in place using a simlar procedure as the capacitors. LEDs are polarized. You must install the LED so that the cathode side of the LED is on the right. The cathode of the LED is on the same side of the LED where there is a flat indentation in the body of the LED. Also, the anode lead of the LED is the longer of the two, which should be inserted in the left hole in the LED footprint on the PCB, which is marked with a + sign on the board's silkscreen.

6. Next, solder in J_1 and J_2 using a similar procedure, using tape to secure the components in place prior to soldering.

7. Solder the trimmer potentiometer R_2 in place. It only fits one way, and should be taped in place (or leads bent) prior to soldering.

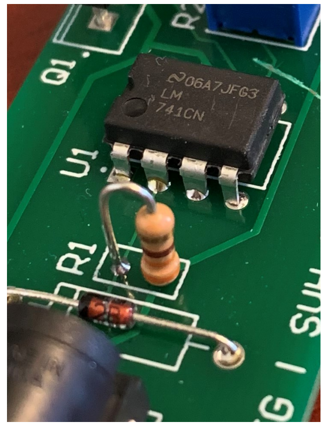

Figure 3: "Hairpin" resistor installation

8. Finally, solder in Q_1. Pay attention to the way you mount this component. The base should be on the left; the emitter on the right. Refer to the datasheet. This will require the transistor to be mounted so that the heatsink faces *away* from the board. Once inserted, you can also bend the leads of the transistor so that it does not fall out when you invert the board to solder it. It is best to solder one lead first, and adjust the position of the component by resoldering, prior to soldering the remaining pins.

Your completed board should resemble Figure 4.

5 Test and Measurement

You now have a completed voltage regulator board that is ready to test. We will begin with a DC test, much as we did in Section 3. A wire has been supplied with your kit that has a 2.1mm coaxial end that plugs into J_1. The other end terminates in bare wire leads that you can connect to the DC power supply in the lab. For the load, resistor leads can be bent so that they can be inserted into J_2 and secured in place using the screw terminals.

5.1 DC Testing

1. Connect OUTPUT 2 of the DC power supply to the voltage regulator and set the voltage to 12 V. Make sure you connect the wire such that the centre conductor of the coaxial plug is positive (+12V) and the outer conductor is negative. Double check this with a multimeter before plugging into your board.

2. Enable current limiting on the power supply (in case you have assembled your board wrong) by setting the current limit to 200 mA.

3. Before attaching a load, power on the circuit. The LED should illuminate, and the current draw should be very low (a few mA). If the LED does not light, you may have installed it backwards.

4. Using the digital multimeter, measure the voltage across the output terminals. You should

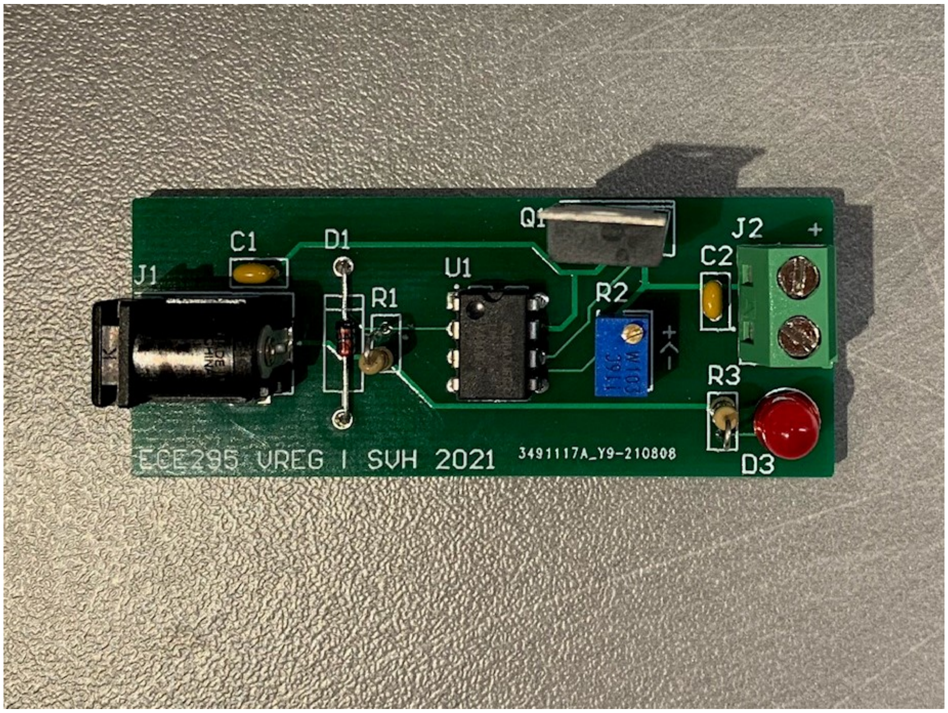

Figure 4: Photograph of completed M0 board

see a reasonable voltage. Adjust the trimmer R_2 until you achieve 5 V at the output.

5. Check the voltages at the inverting and non-inverting terminals of the opamp. What are the voltages here? How do they compare to your expectations? Try to explain what has happened in your notebook.

6. Connect a $330\ \Omega$, 0.25 W resistor to the output. Verify that the output is still 5 V.

7. Sweep the input voltage to the circuit from $0 - 30$ V in half-volt steps on the power supply, and plot the output voltage as function of the input voltage. Is it tedious doing this by hand? We will learn how to automate this process soon, in future structured labs on test and measurement.

5.2 AC Testing

1. There are various "wall-wart" style adapters that plug into the wall that have the DC plug on the end. These are actually unregulated transformer-based power supplies similar to the one you analyzed in Section 3.2, with the transformer, diode, and filter capacitor all bundled together in a single package. A datasheet for these adapters is available that shows a schematic for the rectifier and the value used for the filter capacitor. One such adapter is included in your kit.

2. To make for a more challenging load, obtain a $33\ \Omega$, 1 W power resistor and connect it to the output of your regulator. Plug the power adapter into your module.

3. On an oscilloscope, plot the input voltage by setting the timebase on the oscilloscope such that about 10 periods are displayed.
   (a) With DC coupling, there is not much to see aside from a straight line, since we are so zoomed out that we cannot see the fine variations in the input voltage. In order to zoom in, we need to change the voltage scale on the oscilloscope, while simultaneously offsetting the input, so that the trace remains visible. This is kind of a pain, and makes triggering off the waveform quite difficult.
   (b) Instead, use AC coupling on the scope to remove the DC from the signal. Now we can zoom in quite easily to see the rectified waveform emerging from the adaptor.
   (c) Using the second channel of the scope (and also AC coupling), plot the output voltage using the same vertical scale. Has the voltage regulator improved things? Save plots of the input and output voltages.
4. Take measurements of the peak-to-peak and RMS values of both the input and output voltages using the oscilloscope.
5. Use the digital multimeter to measure both the DC input and output voltages, as well as the AC RMS input and output voltages. How do they compare to the oscilloscope measurements?

The linear voltage regulators discussed in this document are very useful for powering circuits. The main drawback of linear voltage regulators is that the voltage drop between the input and output voltages must be dissipated by the regulator. For example, if we are dropping 12 V down to 5 V, and the load consumes 100 mA of current, then the regulator must dissipate approximately $(12 - 5)(0.1) = 700$ mW of power as heat. You may have noticed your series pass transistor / integrated voltage regulators get quite warm, especially when there are low load resistances powered by the circuit.

You can accurately measure the temperature of the voltage regulator using the U5855A TrueIR infrared thermal imaging camera. Ask a TA to help you with this.

6 Closing Remarks

The voltage regulator you have just built is quite capable and very useful for experiments. For powering the modules on the radio, we will use integrated circuit solutions for voltage regulators, since they incorporate features such as temperature compensation, over-current protection, thermal protection, and other features that your regulator lacks. You can find a schematic of the linear voltage regulator circuits in the SDR mainboard Altium files.

The main disadvantage of linear regulators is that they are not very efficient (think of the heat generation discussed above). A much more efficient solution, which is more common nowadays, is to use *switching* or *switchmode* regulators, which use transistors in a switching mode to achieve the voltage transformations. These are called *DC-to-DC converters*, and comprise a more advanced topic studied in courses such as ECE533 and ECE514.

Nevertheless, linear regulators do not generate high frequency noise like switchmode regulators do. Their "clean" outputs and simplicity make them popular regulators in many applications.