ECE383: Microcomputers – Lab 3
Breadboarding and validation of PIC setup

Goals: The goals of this lab are to introduce students to the initial startup schematic of the PIC24 / dsPIC33, which will be covered more in depth in later weeks in the course, and to get the prototype breadboard prepared for future lab assignments.

1. Introduction

This prototyping walkthrough is to help you prepare the hardware setup that you will need for Lab 6. This hardware is needed so you can execute and debug code you write in MPLAB on the actual microcontroller (dsPIC33) hardware.

This will require you to assemble the circuitry to power the dsPIC33 and the circuitry for USB communication between a computer and the dsPIC33. This will be done in multiple short steps that are outined in this document. Through this process you will need to capture photos of your setup and measurements with the equipment you used in Lab 2 to confirm that your circuit has been build correctly.

The microcontroller we will be using through these labs is the dsPIC33. While it has a slightly different name than the PIC24 we reference throughout the course, it is functionally equipvalent (it has some additional hardware and features beyond the PIC24, but we are not using them in this course).

Figure 1 shows the pinout for the dsPIC33 microcontroller. This will help you understand the functionality of the pins we are using during the breadboarding. Refer back to it when you need to find out the pin number of the dsPIC33 that you need for wiring referenced components to.

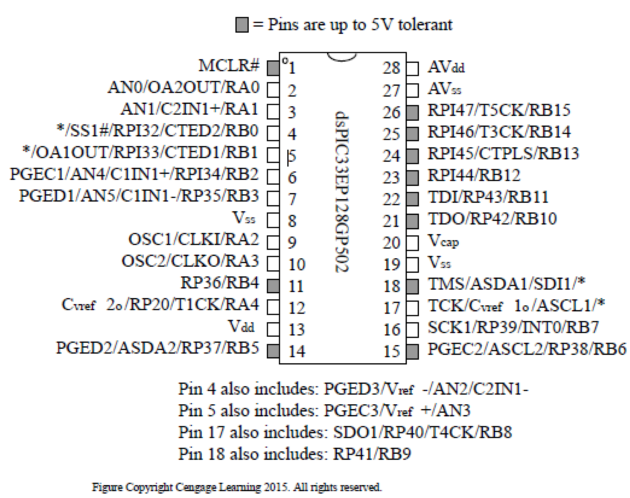


Figure 1. dsPIC33EP128GP502 microcontroller pinout.

The electrical schematic of the circuit you will be breadboarding is outlined below.

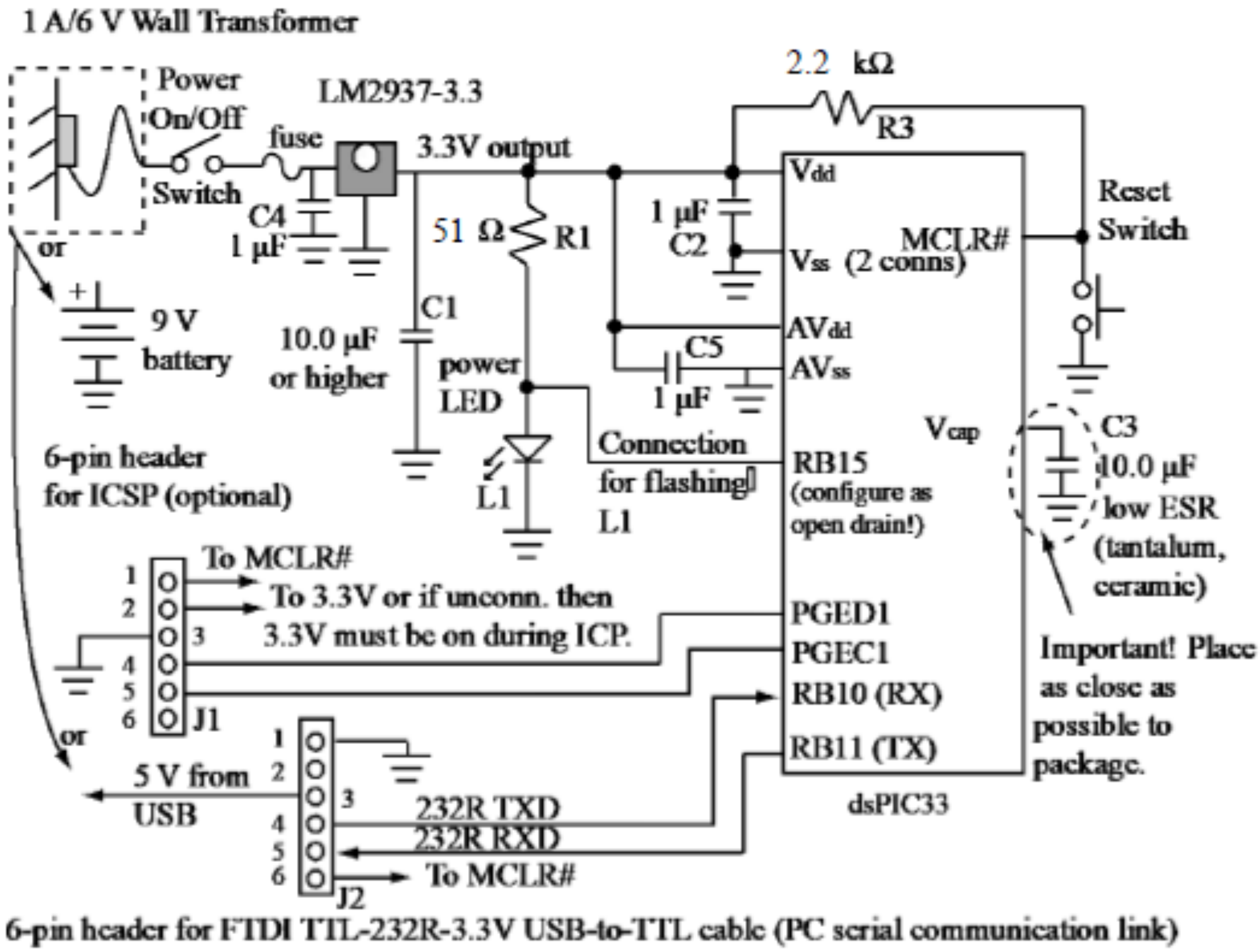


Figure 2. Startup schematic from the textbook that will be built as the platform for later labs in the course.

2. Power Distribution on the Breadboard

Both the microcontroller and other electronics you'll use in the course need to be conntected to a power source (most often 3.3V, but also 5V for some components) to operate correctly. To make it easy to connect components to this voltage supply, breadboards use rails that can be easily connected to distribute this voltage across large areas of the breadboard.

To start with this process, connect the power/ground rails as shown in Figure 3. Use wires to connect or "strap" all of the rails together. Do not connect the power rails on the breadboard to the power post.

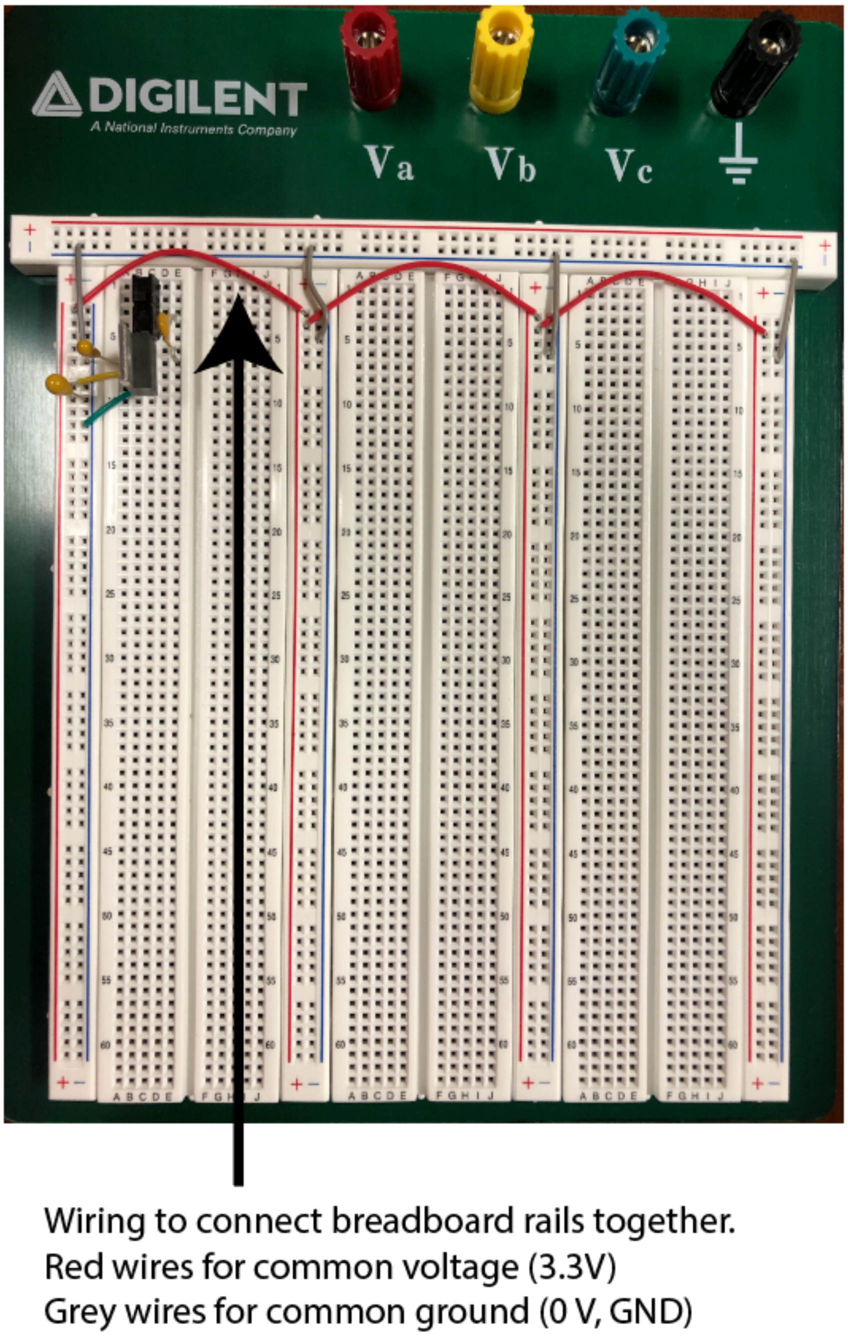


Figure 3. Power/Ground rail connections on breadboard.

A tighter zoom of the wiring of the voltage/ground rails is shown in Figure 4. For your initial setup, you do not need to have the voltage regulator, switch, and capacitors that are shown in the left of this figure (you'll add these later). For now, focus on the wiring of your voltage rails. Your breadboard and wiring may look slightly different based on which locations you use to wire, that is alright (as long as you meet the functionality requirements).

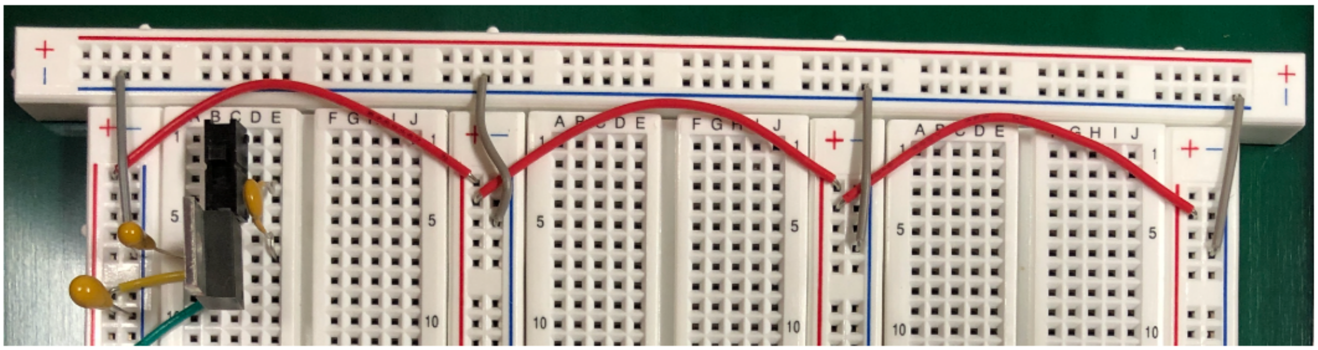


Figure 4. Tighter zoom of power/ground rail connections on breadboard.

After setting up these connections, both power and ground will be distributed around your breadboard for easier wiring when you start adding components. Remember that the vertical rails that you have wired together are all internally connected. Figure 5 highlights what this means for where power and ground are distributed for your use later.

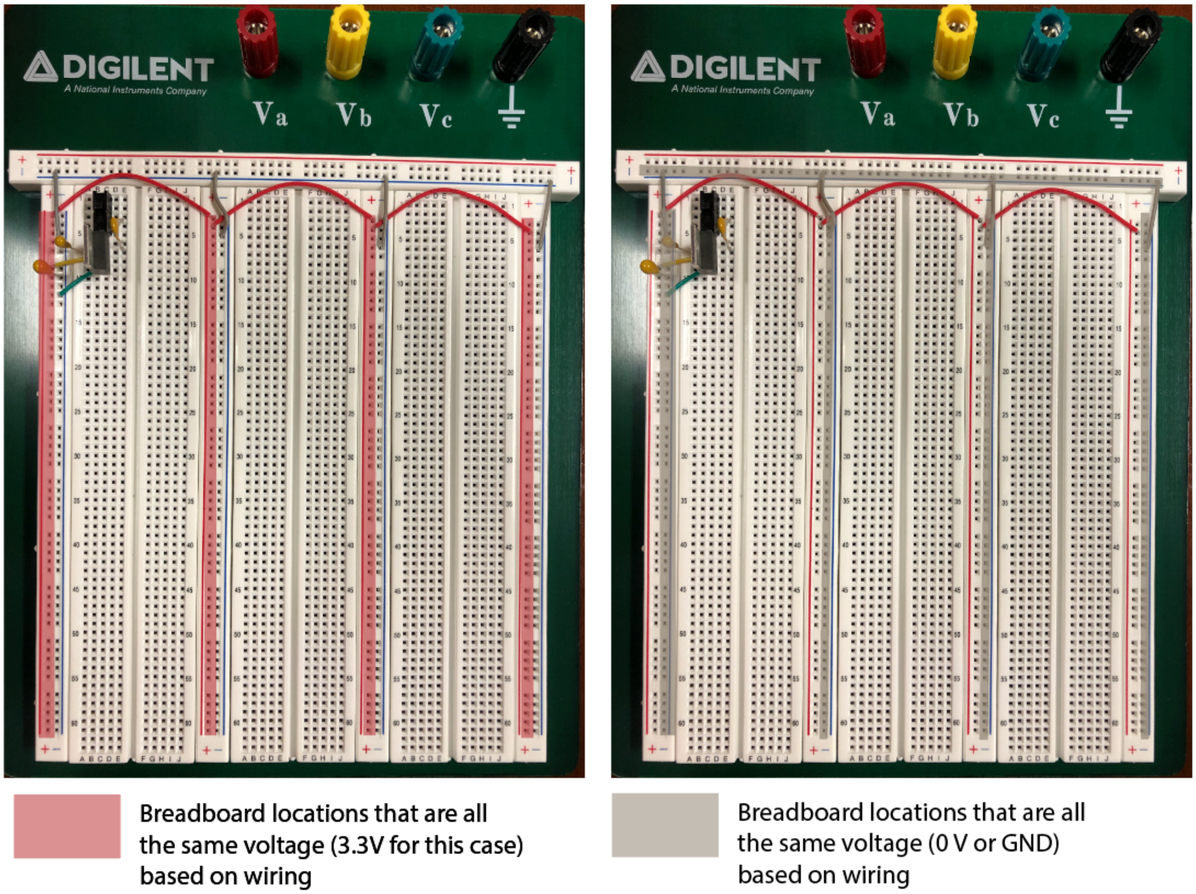


Figure 5. Distribution of power rail (3.3V) and ground rails (0V, GND) based on breadboard wiring.

<u>Deliverable 1:</u> Upoad a photo of your breadboard after you have setup the power and ground rails. In this photo, have a hand-written note that has your **name** and **CWID** visible. If working as a group, include the name and CWID of both partners. Photos without this detail will be awarded a score of zero.

3. Power Control and Regulation

To power the microcontroller (and other electronics in the course), a voltage of **3.3V** is needed. This 3.3V voltage will be generated using a **voltage regulator**. This component will convert an input 5V value (that's available from the USB port of a computer) to a "regulated" or tightly controlled 3.3V value. In addition to this regulator, you will wire in a power switch (that can toggle the voltage on/off) and a fuse (which will provide some protection from high currents damaging the electronics in case of wiring something incorrectly in this or later labs). A photo of the power switch, fuse, and voltage regulator and the electrical schematic of their connections are provided in Figure 6.

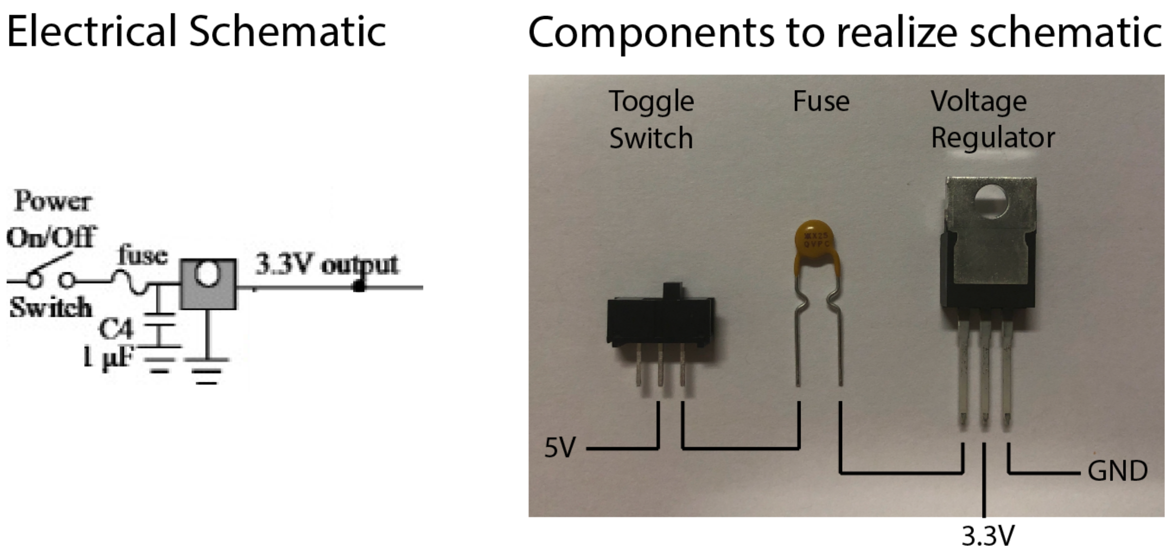


Figure 6. Power switch fuse, and voltage regulator electrical schematic and sample components with wiring.

Locate the **power switch**, **voltage regulator**, and a **PTC (self-resetting fuse)** in the parts kit. While we will provide basic descriptions of the componets in this walkthrough, more comprehensive details for every component is always available in the component datasheet. You can usually access these datasheets on DigiKey (or the supplier you purchased the parts from) or directly from the vendor websites.

The switch makes/breaks a connection between the middle pin and the outer pin(s). Place the **power switch**, **voltage regulator**, and **fuse** on the board as shown in Figure 7. Notice that the C1 and C4 capacitors are shown in Figure 7 (and in the original schematic given in Figure 2) but are not shown in Figure 6 (they were removed for clarity and focus on the switch/fuse/regulator).

These capacitors help "smooth" the voltage inputs/outputs of the regulator to improve its performance. Depending on the 5V source used as the input to the voltage regulator they are not always needed, but we suggest placing both C1 and C4 for your breadboard setup in this course. The C1 capacitor needs to placed across the power rails as shown. This capacitor is polarized (meaning that the orientation matters and a specific pin must be connected to the expected higher voltage) - the longer lead is the '+' terminal.

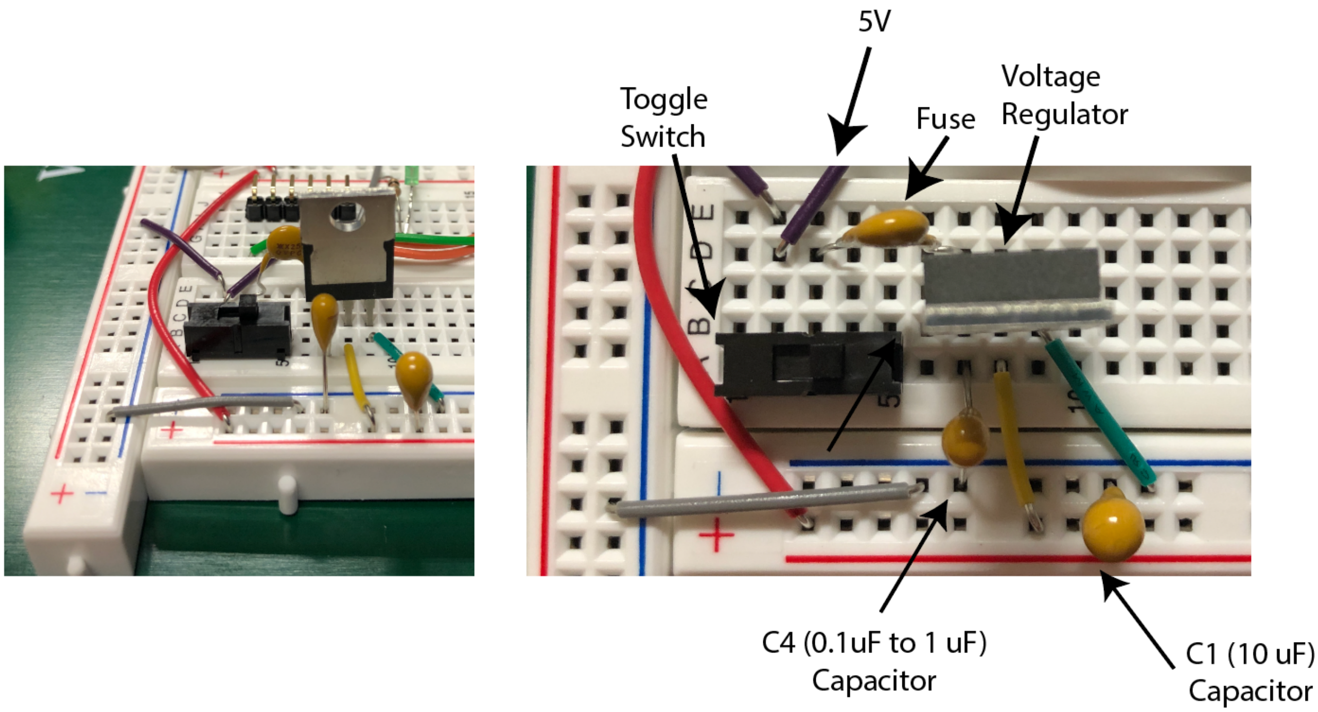


Figure 7. Side and top views of power switch, votlage regulator, fuse placement, and supporting capacitors on breadboard .

Note: Figure 2 with the original electrical schematic notes a "+9V input" you will be using a 5V input for your breadboard. This change is based on using USB supplied voltage instead of an external power supply.

Next, locate an **LED** and **51 Ohm** resistor in your parts kit. A sample of these components is shown in Figure 6 (but you may have a slightly different looking LED, don't worry if it doesn't look exactly the same or is a different color).

An LED is a "Light-Emitting Diode" and it will generate light when an appropriate voltage is applied to it. It will be used on the broad-board as a visual indicator that power is being applied (with a second used to confirm operation of the microcontroller). This is extremely helpful when working with electronics so you can quickly check that your system has power. Without it, you have to use a multimeter to measure the voltage manually.

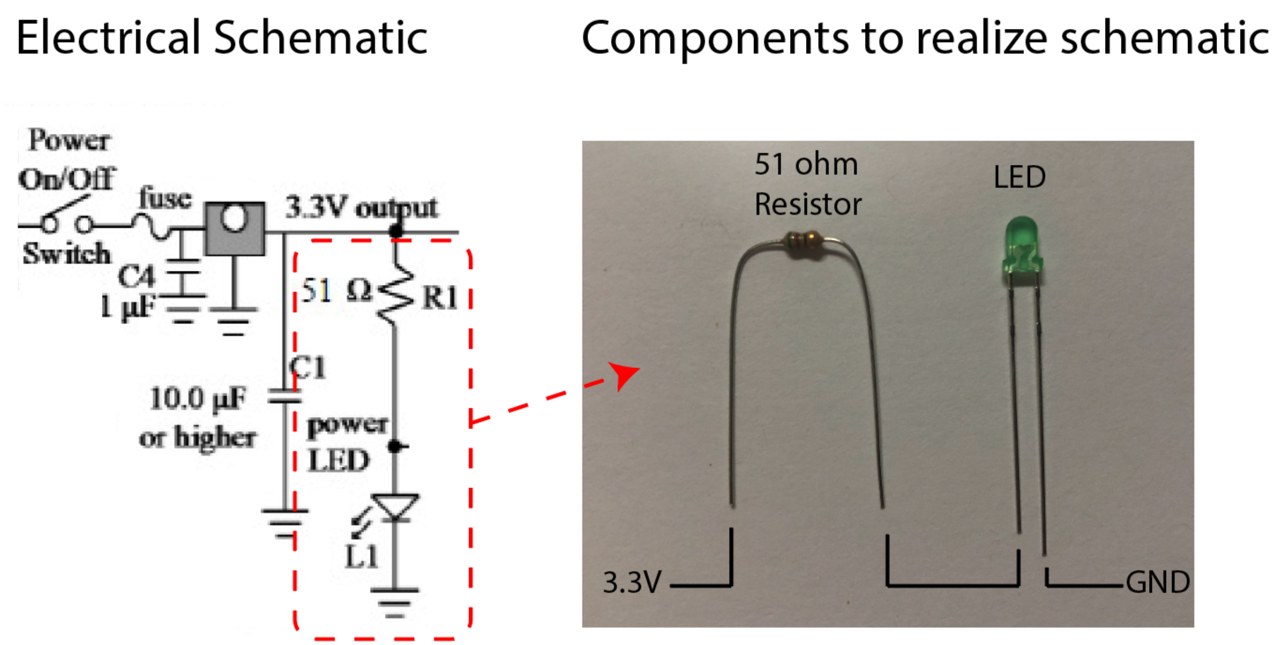


Figure 8. Power-on LED and a 910 Ohm resistor.

A sample of the resistor and LED electrical schematic and sampels of components for this wiring are shown in Figure 8. Place the power-on LED and the 51 Ohm resistor on the board, a sample of one way to do this is shown in Figure 9.

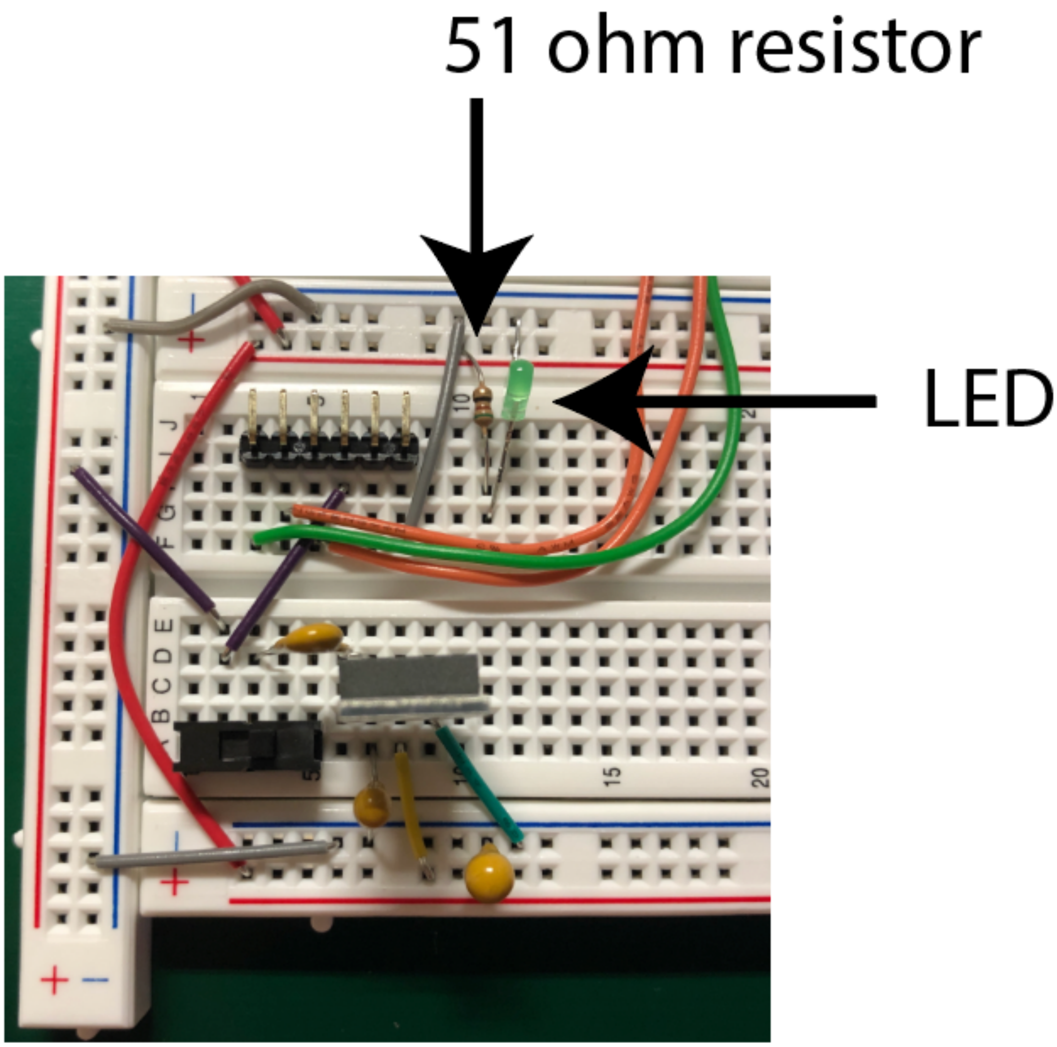


Figure 9. Power-on LED and a 51 Ohm resistor placed on breadboard to serve as visual indicator that power has been applied.

4. 5V Power Input

Before being able to toggle your LED on/off, you will need to connect **5V** to your breadboard. This 5V will be supplied from a computer through a TTL-to-USB cable (which will also be used later to program your microcontroller). We will first walk through the steps of connecting it to your breadboard to validate the operation of the voltage regulation circuit and LED before moving on.

Figure 10 shows the electrical schematic of the connection between the USB-to-Serial cable (when interfaced to the breadboard) and the power switch. The specific components (USB-to-serial cable and the 6-pin header / breakout pins for it to plug into on the breadboard) are also shown in Figure 10.

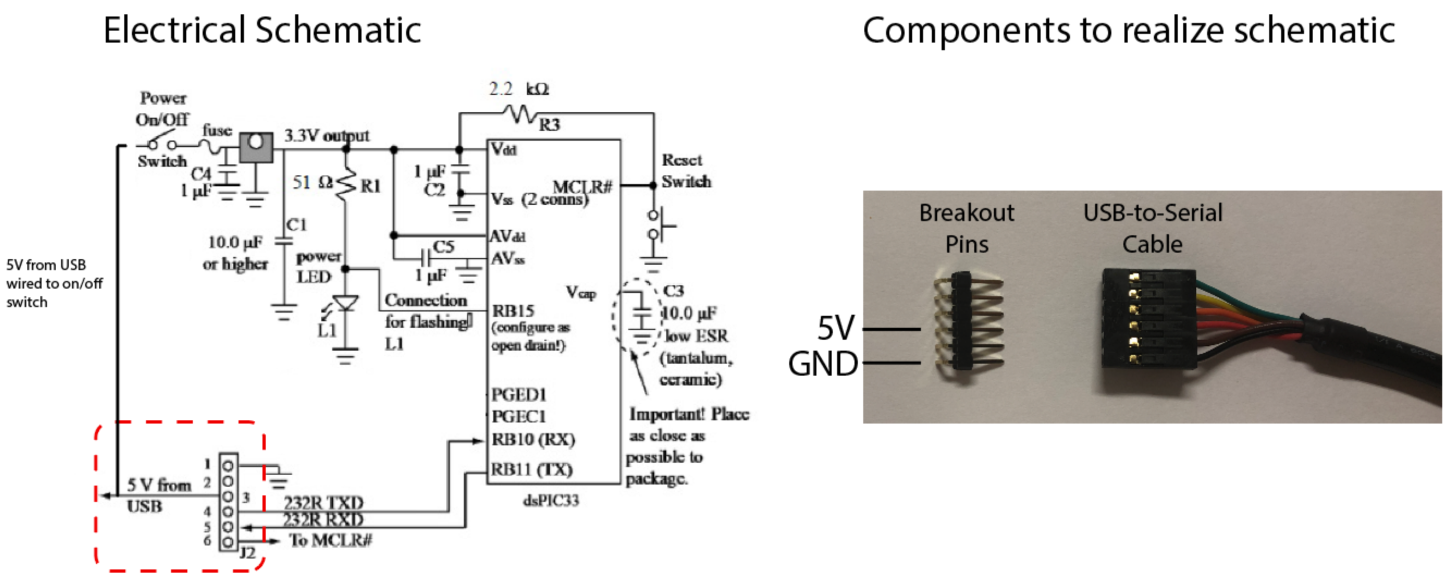


Figure 10. TTL-to-serial CMOS connection for serial port.

The two important connections for power are VCC (red wire) and VSS (black wire). The Vcc wire is 5V relative to the Vss (GND or 0V). Both wires MUST be connected correctly to your system for it to be powered. A very common mistake is to only connect the Vcc wire. Without the Vss or GND wire, the system will not be powered or operate.

Find the TTL-to-USB cable in your kit of lab materials. Connect the 6-pin right-angle header and TTL-to-serial cable to your breadboard as shown in Figure 11 (but do not yet connect the cable to a computer, keep the system unpowered while you are wiring).

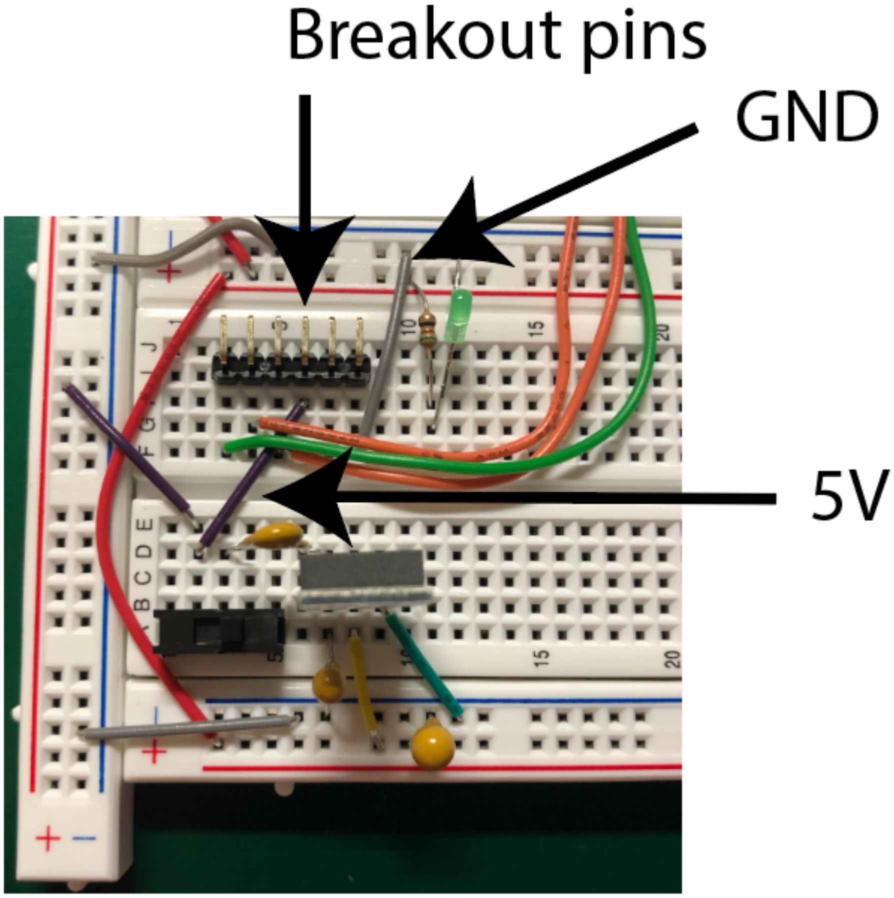


Figure 11. USB-to-serial cable connection to supply power (5V) from USB to breadboard.

Using a multimeter (which you learned how to use in Lab 2) measure the voltage of the VCC (5V) pin of the 6-pin right-angle header on the breadboard in reference to the GND pin when the cable is plugged into a computer. If you are not able to measure the 5V value there is something incorrect in your setup. Double check your connections and how you are measuring the VCC value with the multimeter. Check with the TA if you are not able to trouble-shoot this problem.

> **Deliverable 2:** Capture a photo that has both your breadboard and the multimeter voltage value confirming your correct setup and application of 5 V to your system. In this photo, have a hand-written note that has your **name** and **CWID** visible. If working as a group, include the name and CWID of both partners. Photos without this detail will be awarded a score of zero.

Next, toggle the power switch so that the 5V signal is passed to the voltage regulator. The LED you placed on the breadboard should now be illuminated. If it does not turn on with the power switch try reversing the orientation of the LED. Next, with the power applied confirm that the voltage regulator is generating the expected 3.3V value. Measure the output voltage pin of the voltage regulator (the pin that is connected to the 3.3V supply rails you set up earlier) with the multimeter.

> **Deliverable 3:** Capture a photo that has both your breadboard and the multimeter voltage value confirming your correct setup of the voltage regulator (and its 3.3V output) and the LED. In this photo, have a hand-written note that has your **name** and **CWID** visible. If working as a group, include the name and CWID of both partners. Photos without this detail will be awarded a score of zero.

5. Placement of the dsPIC33

The next steps are the placement of the microcontroller (dsPIC33) and the push-button, (additional) LED, resistors, and capacitors necessary for it's correct operation. The electrical schematic for the connection of these components is below in Figure 12.

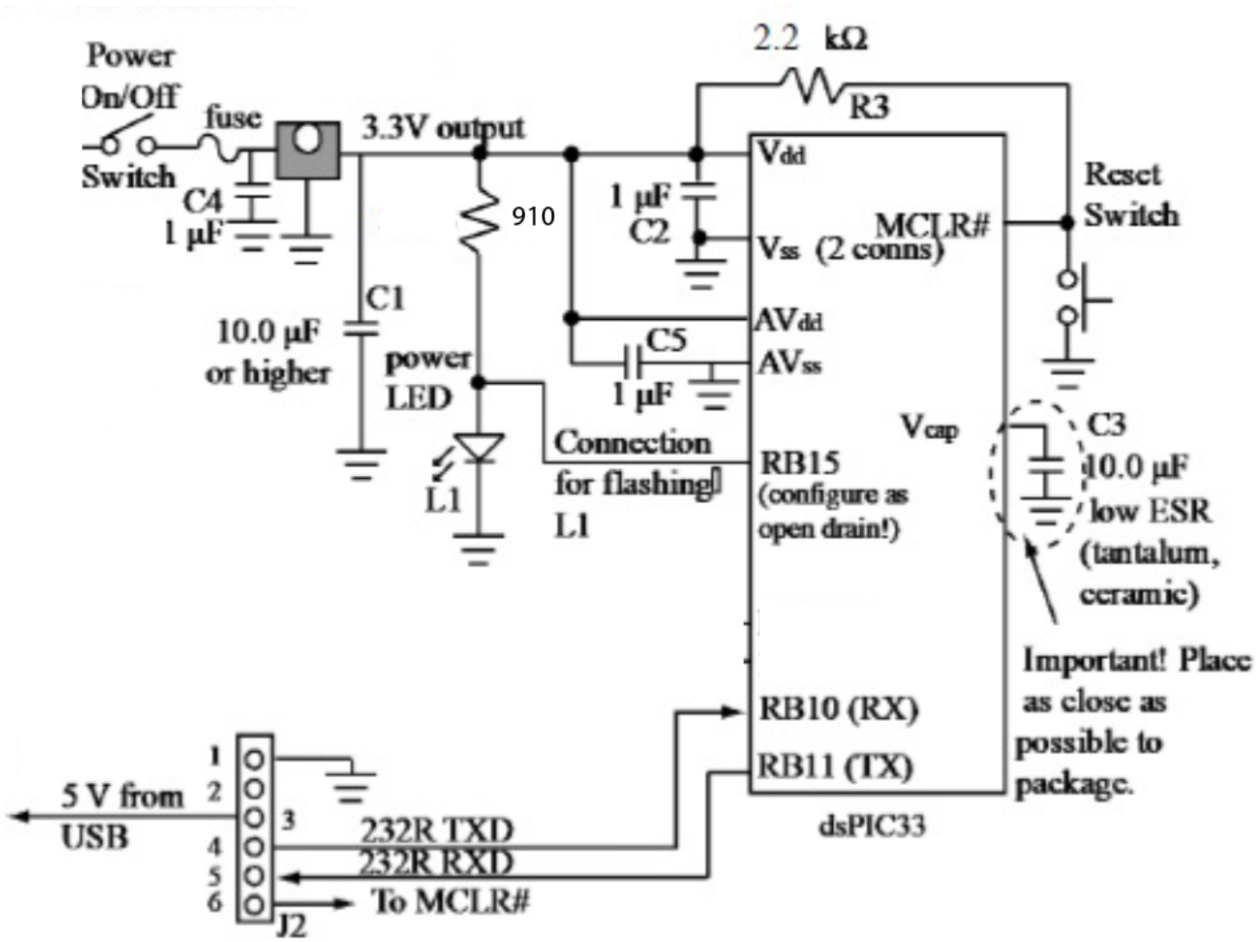


Figure 12.. Electrical schematic of dsPIC33 and connected pushbutton, resistors, and capacitors.

To help understand how the electrical schematic of the dsPIC33 translates to the physical components (and how they are connected), Figure 13 is included. The numbers in white beside the pins of the dsPIC33 in Figure 12 indicate the pin numbers from Figure 1 to help you match the pins to their datasheet referenced value.

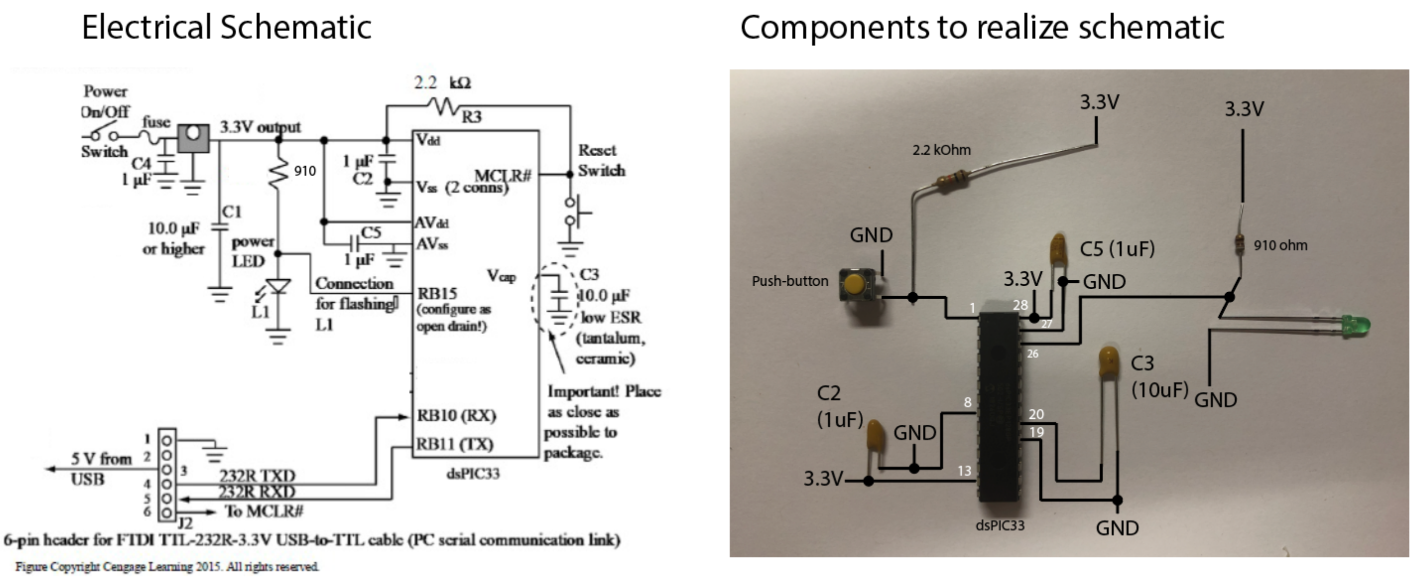


Figure 13. Additional components for reset circuit for dsPIC33, LED indicator, and power smoothing.

The additional capacitors for this setup help further regulate the voltage to the microcontroller (to help its proper operation). The resistor and push-button are added to be able to manually reset the circuit.

The 10.0 uF capacitor (C3), which will be connected to the Vcap pin, is a polarized capacitor. This means that the orientation of how it is connected is important (like an LED). The correct lead must be connected to the voltage (+) and the other to ground (-) or else this component could be damaged and not operate as expected.

WARNING: DO NOT CONNECT THE Vcap PIN TO 3.3 V - THIS WILL DAMAGE YOUR MICROCONTROLLER!

Notice that the Vcap pin on the dsPIC33 is connected to the long lead (+) of C3 only (not Vcc or any voltage value)

The other 1 uF capacitors (C2, C5) in your parts kit may be ceramic unpolarized capacitors (their leads will be the same length). Unpolarized capacitors are not orientation specific (like the polarized). Be sure to check these capacitors and use them appropriately if they are polarized.

To determine the correct orientation of the push-button for placement on your breadboard, you will need to use the multimeter to understand which pins connect to each other when it is pressed. This process is shown in Figure 14.

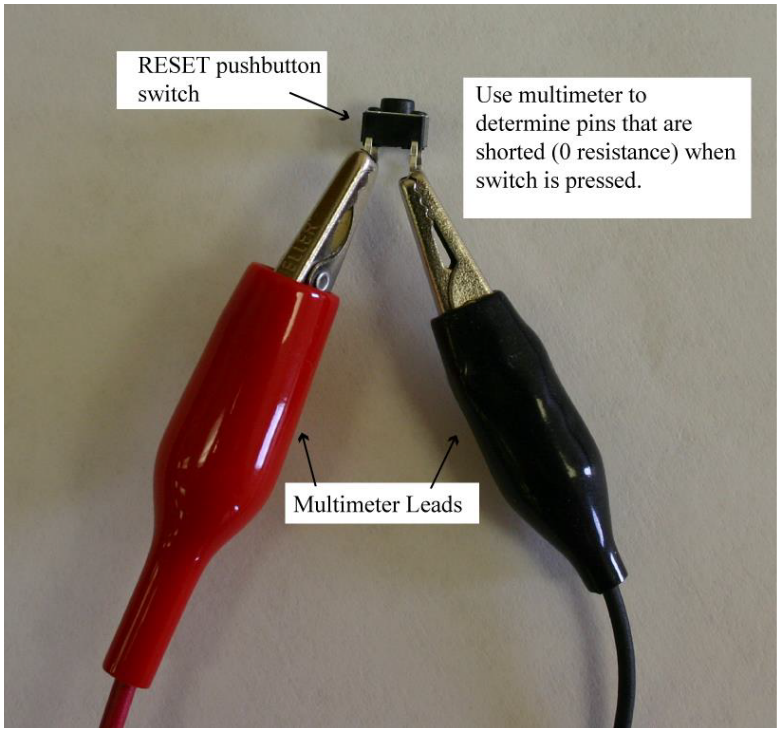


Figure 14. Measuring resistance of pushbutton switch.

With that orientation noted, you are ready to assemble the rest of the components. Figure 15 shows the placement of the dsPIC33 and the reset circuit on board. When placing the dsPIC33 the orientation matters so that your pins are connected to the correct external components. Check Figure 1 from earlier to identify pin 1 based on the half moon / pin 1 indicator that will be on the component. In Figure 15, pin1 is the top-left pin of the dsPIC33 shown.

Note that the dsPIC33 has three ground (2 VSS, 1 AVSS) connections, and two power pins (VDD, AVDD). **All of these MUST be wired to the correct supply rails or your microcontroller will not operate as expected.** VSS / AVSS are connected to the GND rail (0V) and VDD / AVDD are connected to the 3.3V rail. The specific detals of the pin numbers of these pins are given in Figure 1 earlier in these instructions.

Notice that there is a placement guide wire on the breadboard (orange jumpers above and below the dsPIC33). While this isn't needed for the design to function properly, it helps you to remember where to place the microcontroller if you remove it from the breadboard.

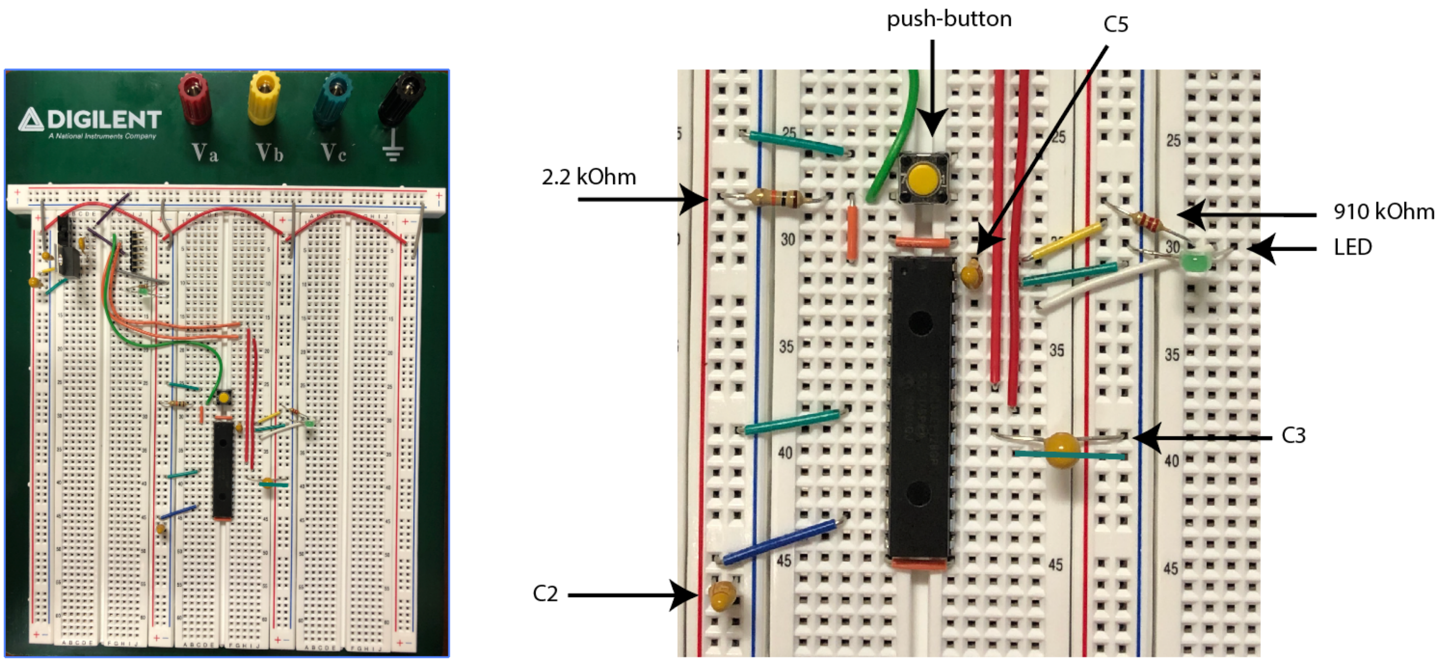


Figure 15. Complete breadboarded circuit (to show dsPIC33 relative to power circuitry) and tighter zoom of dsPIC33 wiring.

5. Connecting USB-to-Serial for Computer Communication

With the dsPIC33 all wired up, the net steps are to connect this component to the header that was setup earlier for the USB-to-serial cable. While this cable has been used to supply power for teseting, it is time to connect the wires so the dsPIC33 and a computer can communicate so you can download programs to your dsPIC33.

Figure 16 shows the USB-to-Serial cable and the 6-pin header for it to plug into on the breadboard. You previously wired this header to supply 5V and GND (not shown in Figure 16). Figure 16 presents only the additional wiring needed for communication purposes: **MCLR#, TX (RB11), RX (RB10)**. The TX and RX signals are the transmit and receive communication with a computer and MCLR# is the signal to reset the dsPIC33.

Before a device can be programmed it has to be reset (to prevent existing code from potentially interfering with this process). The MCLR# connected to the computer gives the computer the ability to do this reset before programming.

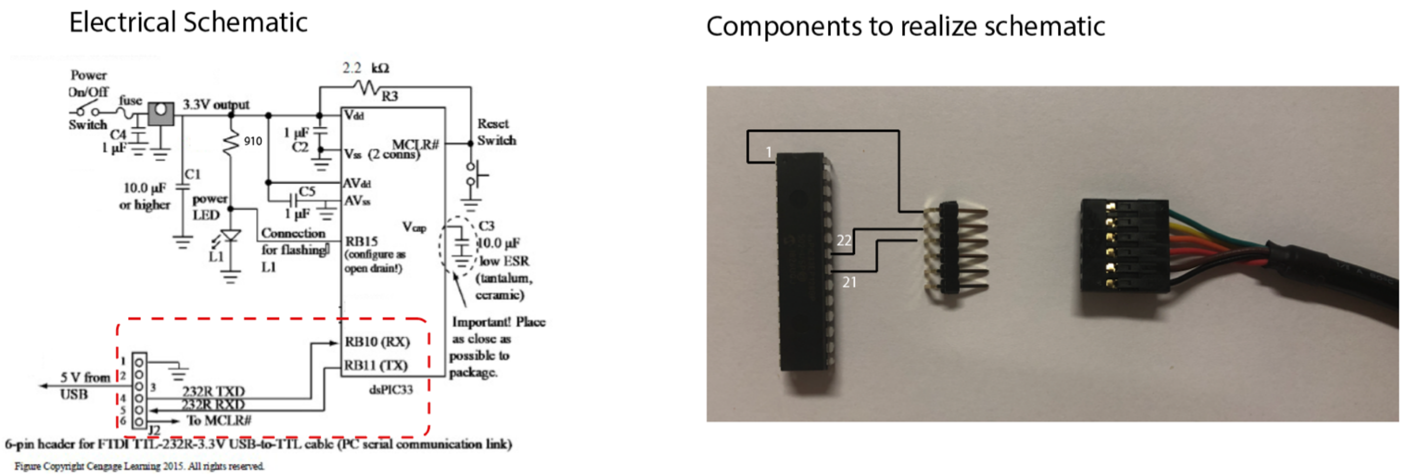


Figure 16. USB-to-serial connection for serial communication with computer.

Figure 17 shows the serial connection placed on the breadboard. Once completed

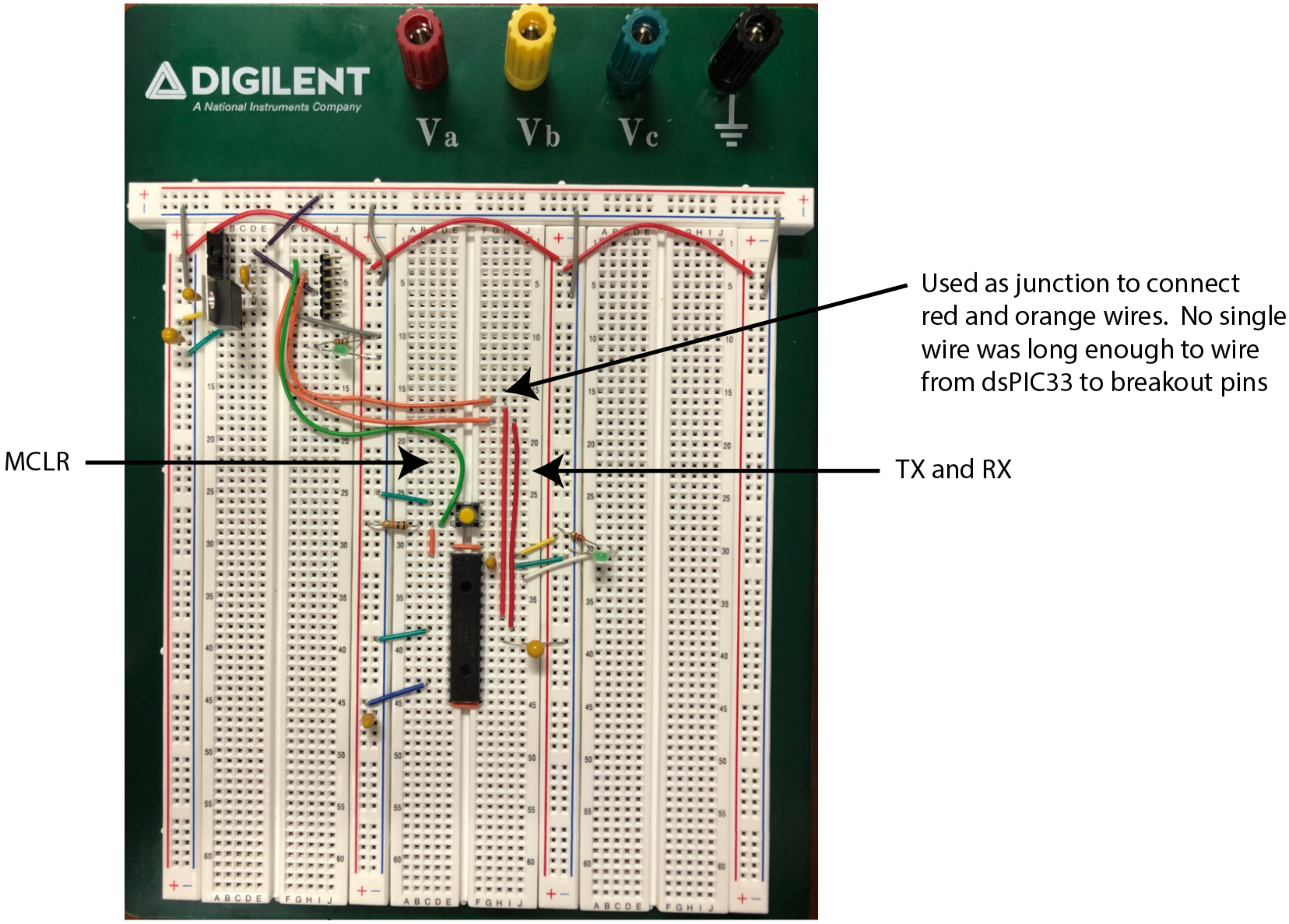


Figure 17. Wiring of TX, RX, and MCLR between dsPIC33 and breakout pins.

<u>Deliverable 4:</u> Capture a photo with your complete circuit built on your breadboard. In this photo, have a hand-written note that has your **name** and **CWID** visible. If working as a group, include the name and CWID of both partners. Photos without this detail will be awarded a score of zero.

6. Program Download

Now you are ready to download programs onto your dsPIC33 using a bootloader program that facilitates communication between a host computer and the dsPIC33 using the USB-to-serial cable. This program, "BullyCPP.exe" is available with the lab documents on Blackboard for you to use throughout the course. Download this program and run the executible. This program is for windows-only. After opening, you will see the program window of Figure 18.

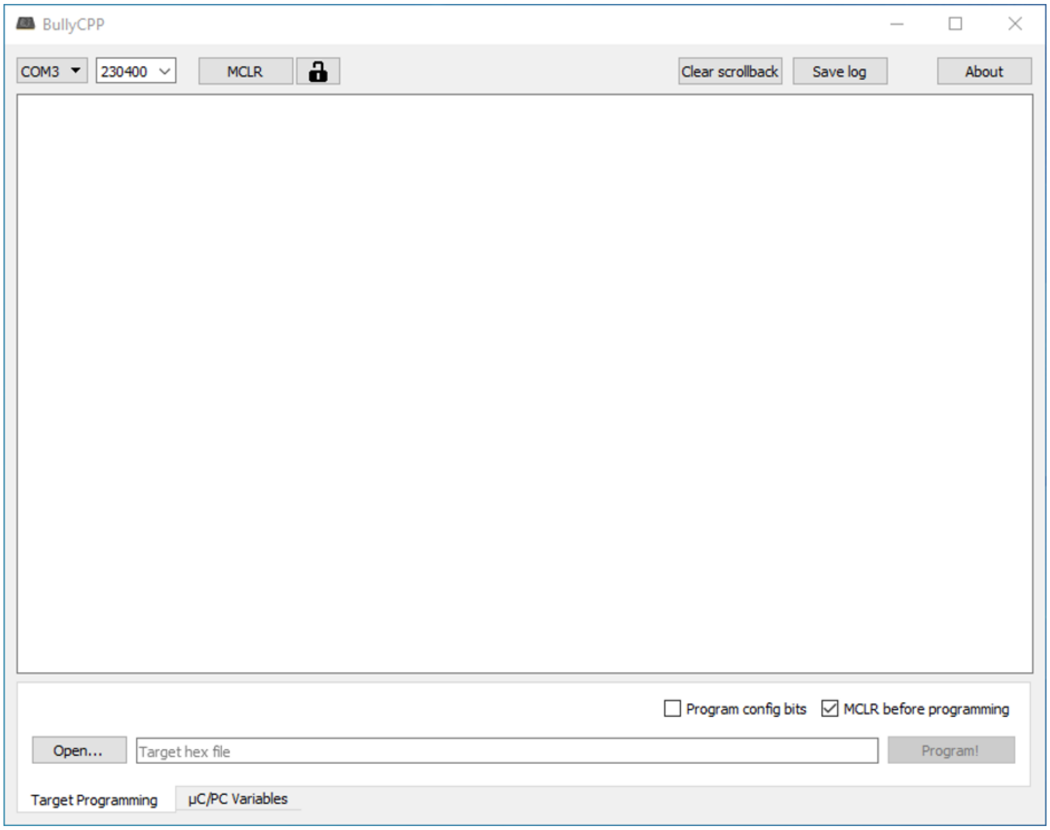
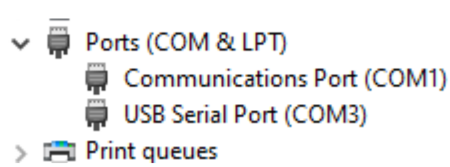


Figure 18. Sample of BullyCPP program after opening.

Before programming your dsPIC33 you will need to 1) set the correct communication port (COM1, COM2, etc.), 2) click the **MCLR before programming button**, and 3) select the target hex file to program the dsPIC33 with.

You can determine which serial port that the USB-to-serial cable is connected to by opening the Device Manager and checking the Ports section. As seen here, my cable is connected to COM3. Change the serial port in BullyCPP to this port and set the baud rate to 230400.



Next, Power on your breadboard circuit with the USB-to-serial cable connected. You may see some serial traffic and a header message in the BullyCPP window, like below. Click open and navigate to the hex file that you wish to download to the microcontroller. For this lab you will use a hex file that has already been prepared for you and is available with the lab files on Blackboard (**echo.X.production.hex**). After you have downloaded and selected this file (like shown in Figure 19) you are ready to program your dsPIC33.

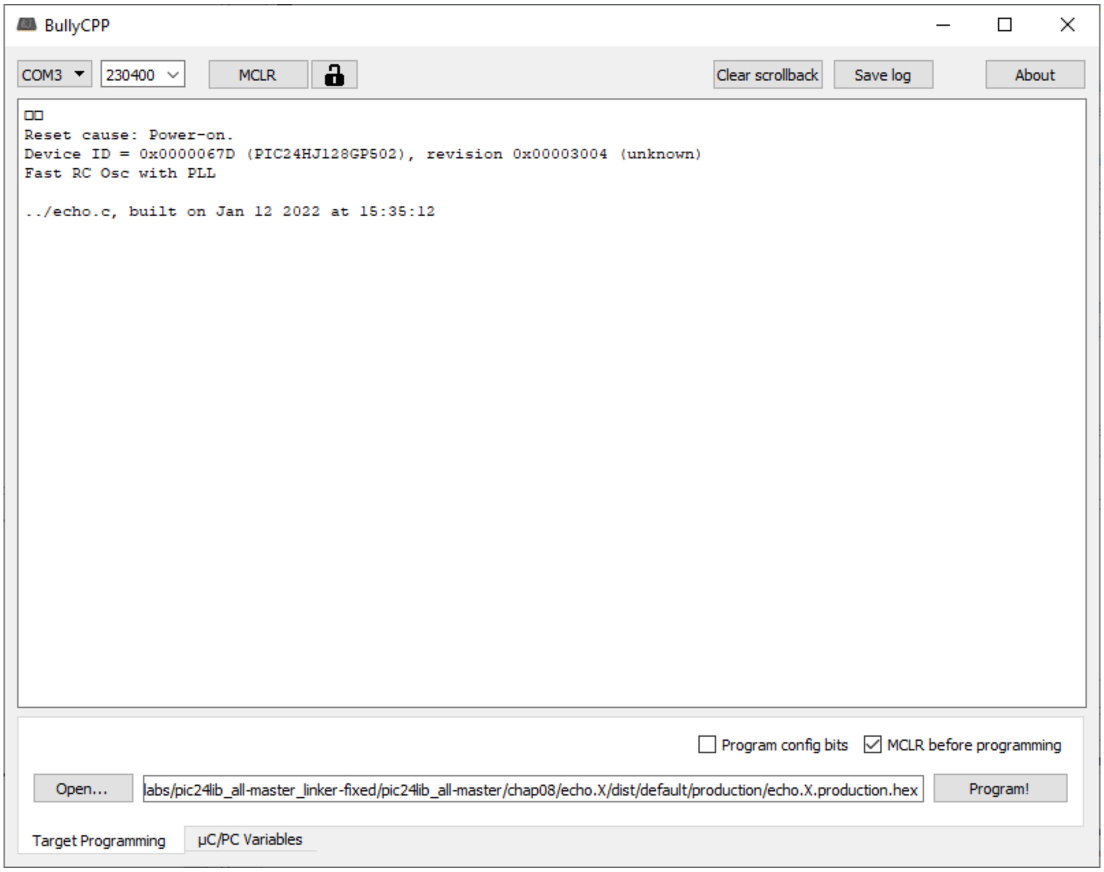


Figure 17. Sample of details received by BullyCPP from dsPIC33 after programming.

First, click the **"Clear Scrollback"** button to clear the communication window and then click **"Program!"**. After programing your device will report back details of the new file loaded to your device and the date it was built. If you receive this reporting, it confirms that your dsPIC33 is functioning correctly! This confirms that your breadboard setup is correct, the power supply is working, and the communication is working. From the program that is loaded, you should also notice that the LED connected to RB15 is blinking. This behavior is part of the program that is now running on the dsPIC33. You will learn more about how to blink LEDs and the code to achieve that later in the course.

Deliverable 5: Capture a screenshot of your entire desktop window that has BullyCPP shown (with the details after programming), the current date/time, and a notepad window open with your name and CWID visible. If working as a group, include the name and CWID of both partners. Photos without this detail will be awarded a score of zero.