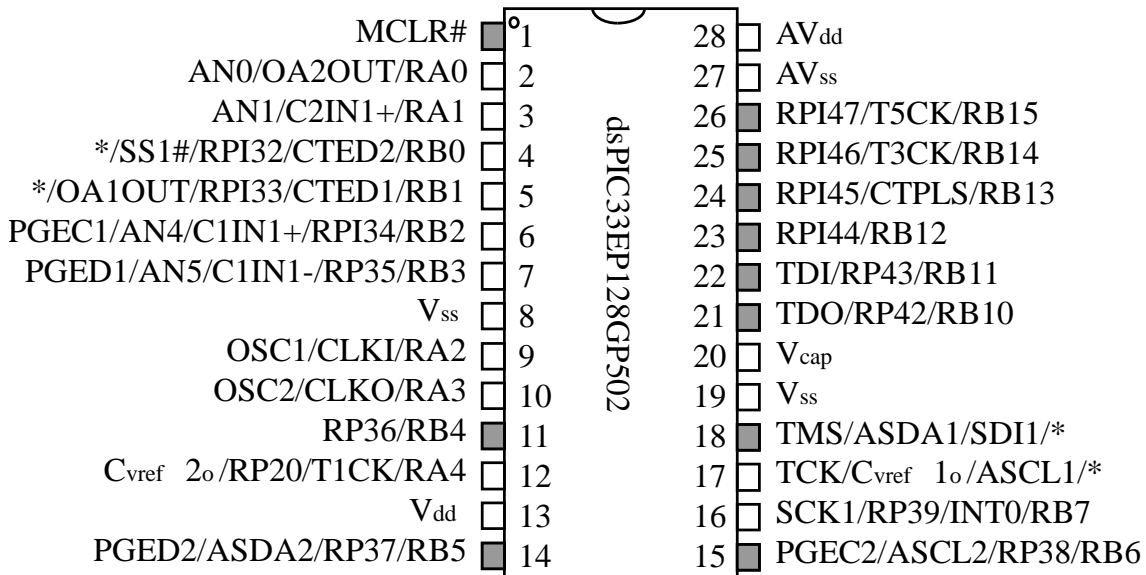


Prototyping Walkthrough for the dsPIC33 Startup Schematic

This prototyping walkthrough is meant to supplement the material in Lab 6, the dsPIC33 system startup. Figure 1 shows the pinout for the dsPIC33 microcontroller. This walkthrough implements the schematic shown in Figure 2.

Figure 1 dsPIC33EP128GP502 Microcontroller Pinout

■ = Pins are up to 5V tolerant



Pin 4 also includes: PGED3/V_{ref} -/AN2/C2IN1-

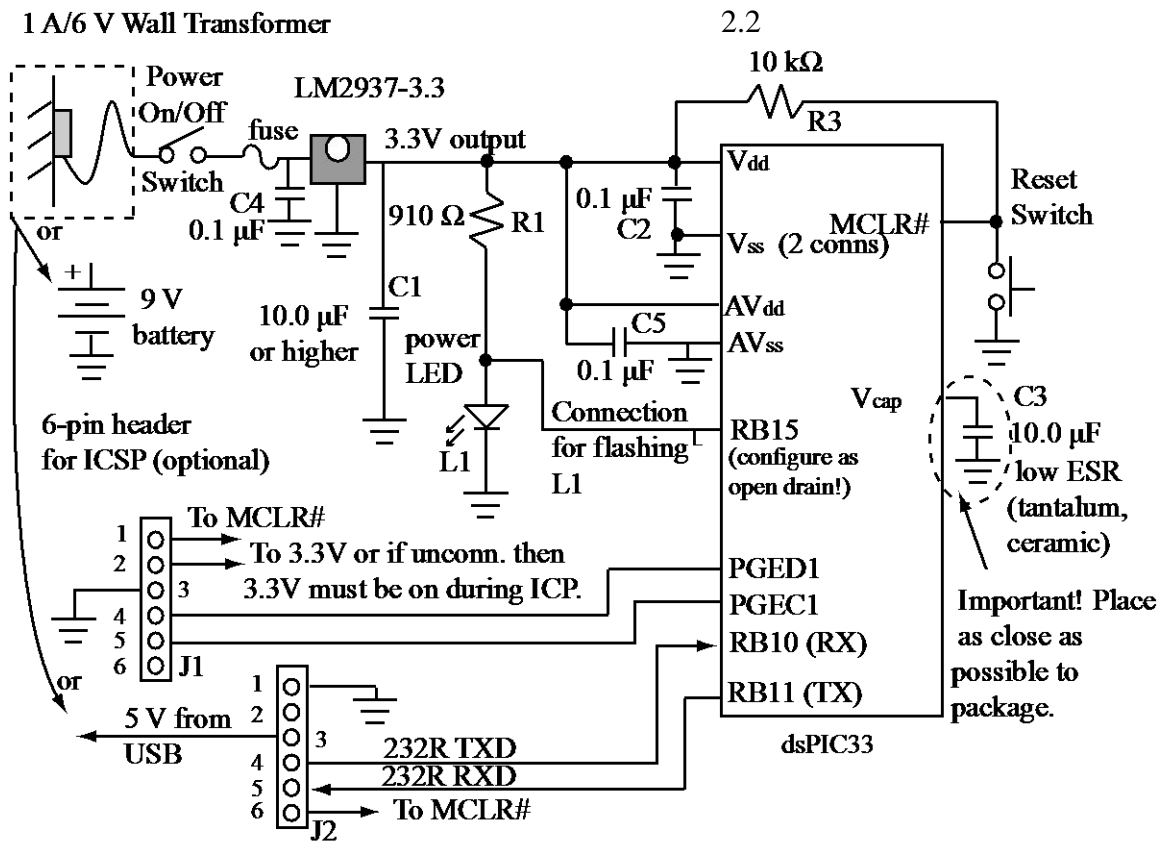
Pin 5 also includes: PGEC3/V_{ref} +/AN3

Pin 17 also includes: SDO1/RP40/T4CK/RB8

Pin 18 also includes: RP41/RB9

Figure Copyright Cengage Learning 2015. All rights reserved.

Figure 2 Startup Schematic

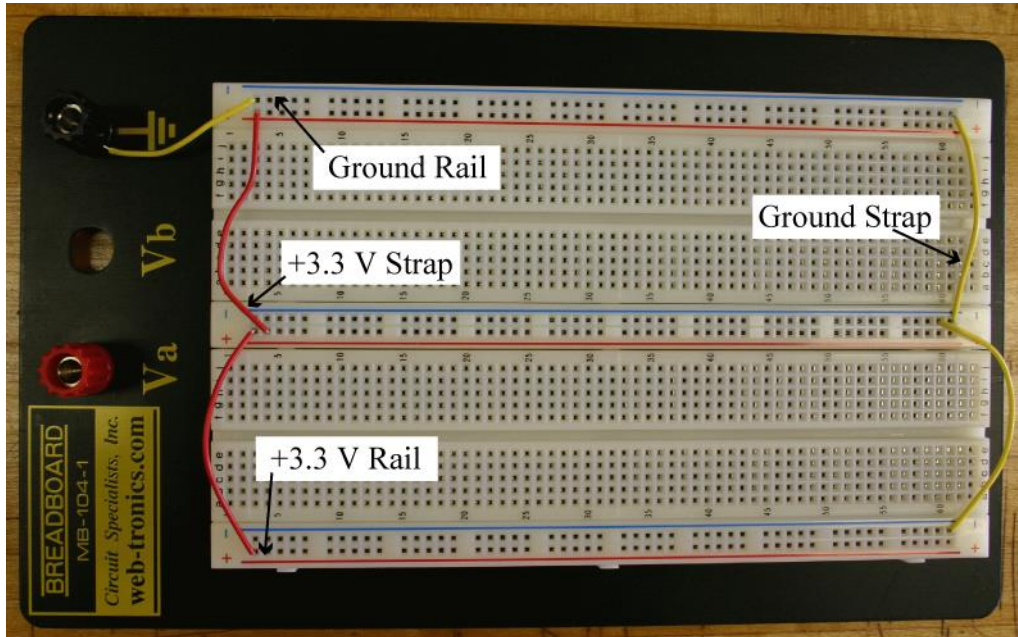


6-pin header for FTDI TTL-232R-3.3V USB-to-TTL cable (PC serial communication link)

Figure Copyright Cengage Learning 2015. All rights reserved.

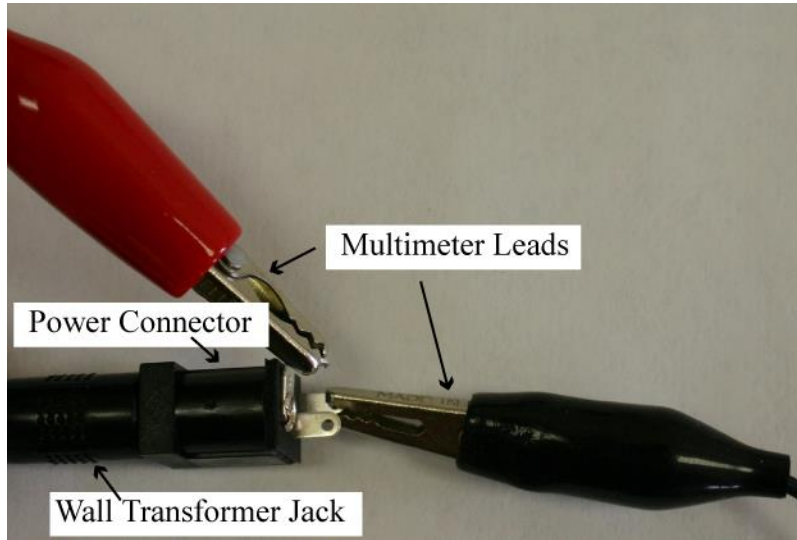
1. Connect the power/ground rails as shown in Figure 3. Use wires to strap all of the rails together. Do not connect the power rails on the breadboard to the power post; this post will be used for 15V to 6V power from the wall transformer. The breadboard ground rails should be tied to the ground power post.

Figure 3. Power/Ground Rail Connection



2. Plug the power connector into the wall transformer jack and use the multimeter to determine which two pins on the power connector provide +6 V and ground as shown in Figure 4. Each power connector has three leads, but only two are used.

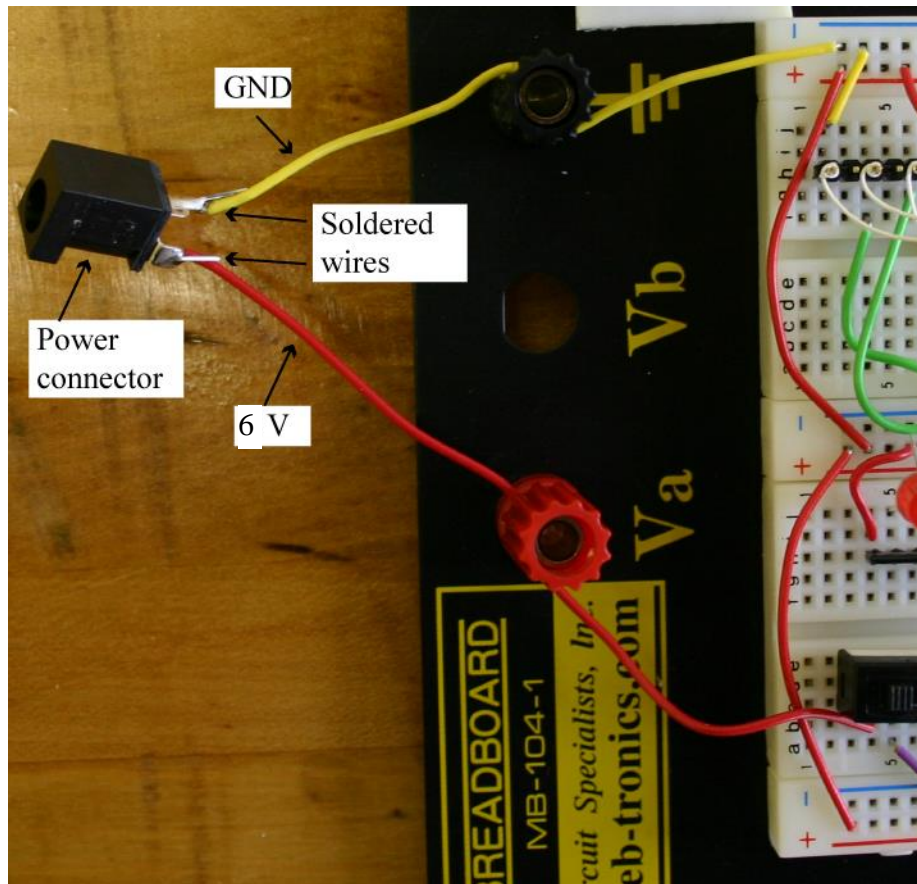
Figure 4. Measure power output of wall transformer



3. SOLDER wires to the power connector, and connect this to the power/ground posts on the breadboard. Figure 5 shows a closeup look of the finished board that shows how the power connector with wires is connected to the breadboard. The GROUND post is connected to the ground rail, but the +6 V power post connects to the input power switch.

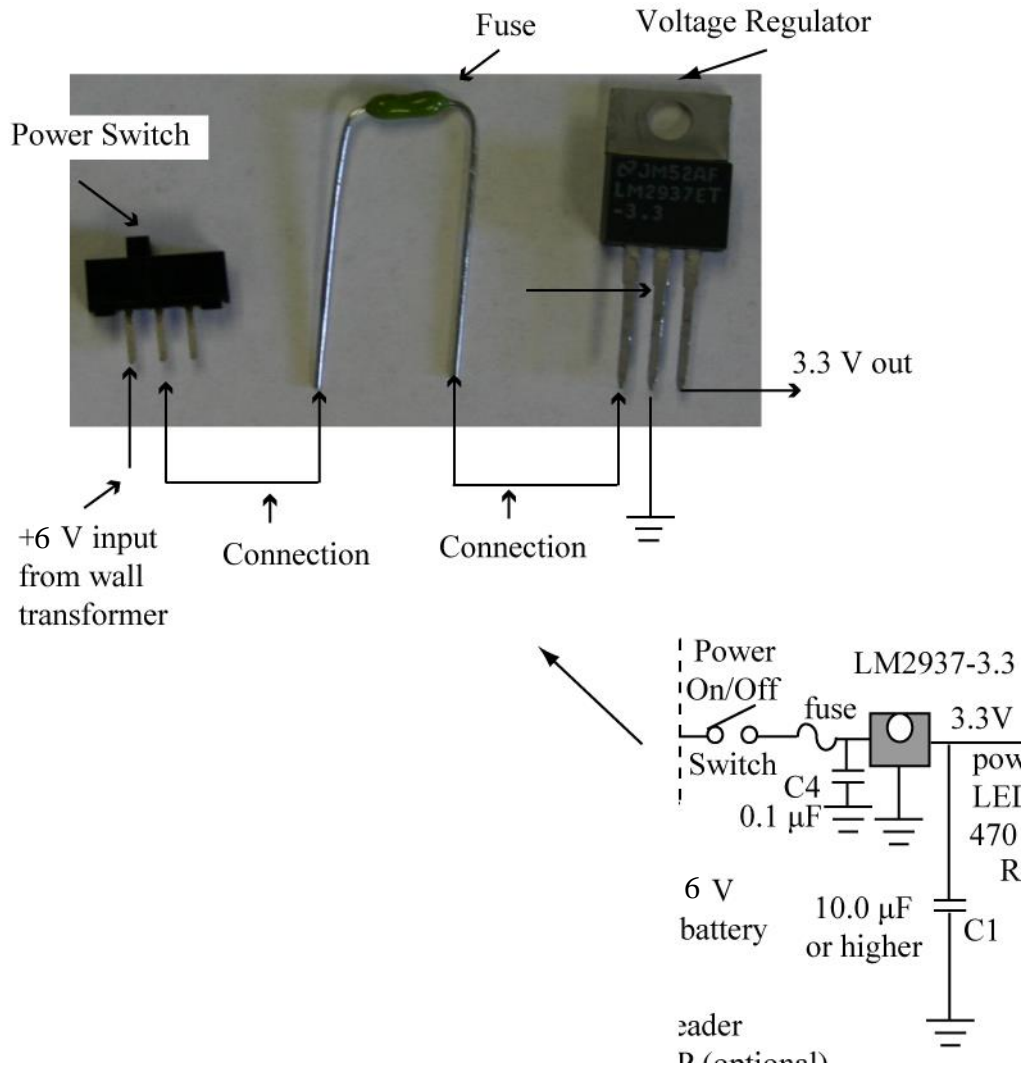
DO NOT just “crimp” the wires onto the power connector; this will create an unreliable connection. Please take the time to solder these wires. If your wires are not soldered, then the TA will not help you debug your board when problems occur.

Figure 5 Power connector attachment to board



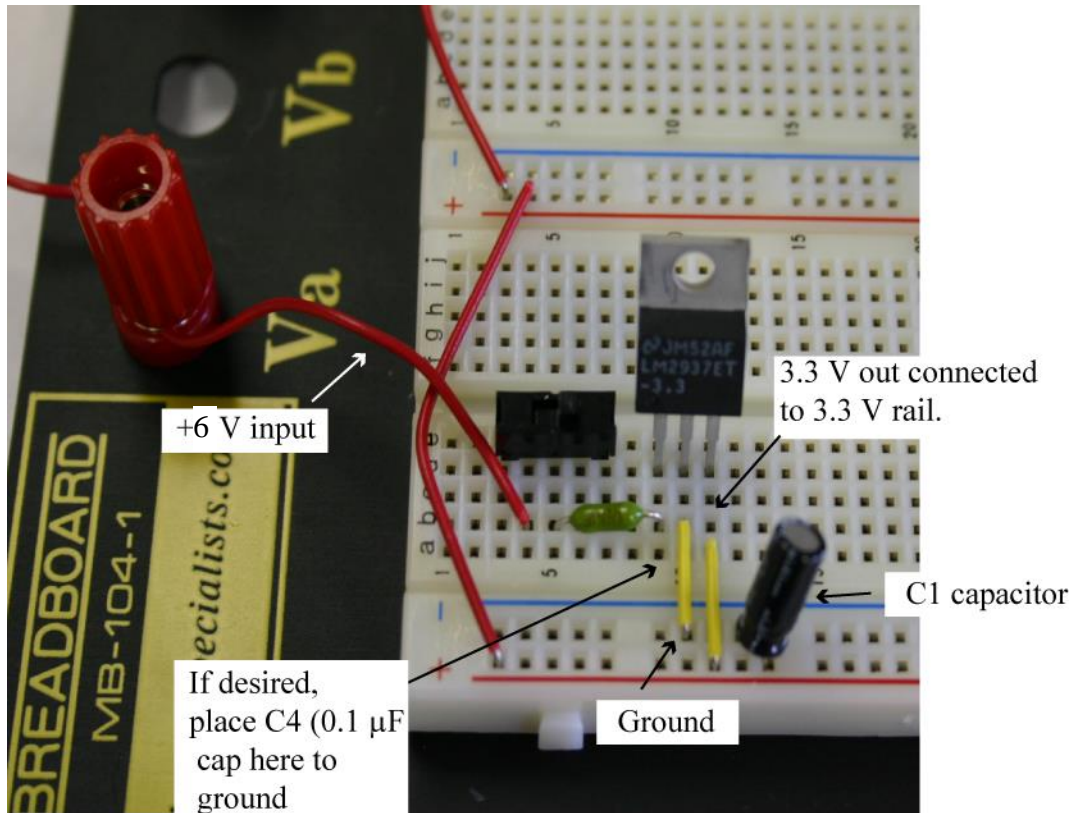
4. Locate power switch, voltage regulator, and fuse in the parts kit as shown in Figure 6. The switch makes/breaks a connection between the middle pin and the outer pin(s). The C4, C1 capacitors are not shown in Figure 5. The C4 capacitor is generally not required, but you need to place the C1 capacitor on the 3.3 V output as shown.

Figure 6. Power Switch, Voltage Regulator, Switch



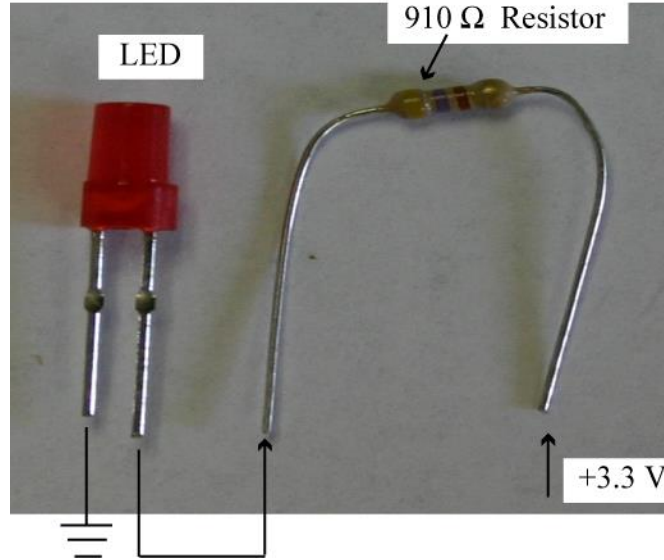
- Place power switch, voltage regulator, and fuse on the board as shown in Figure 7. The C1 capacitor simply needs to be placed across the power rails as shown. This capacitor is polarized (a 'can' capacitor) - the longer lead is the '+' terminal.

Figure 7. Power switch, Voltage regulator, fuse placement on board



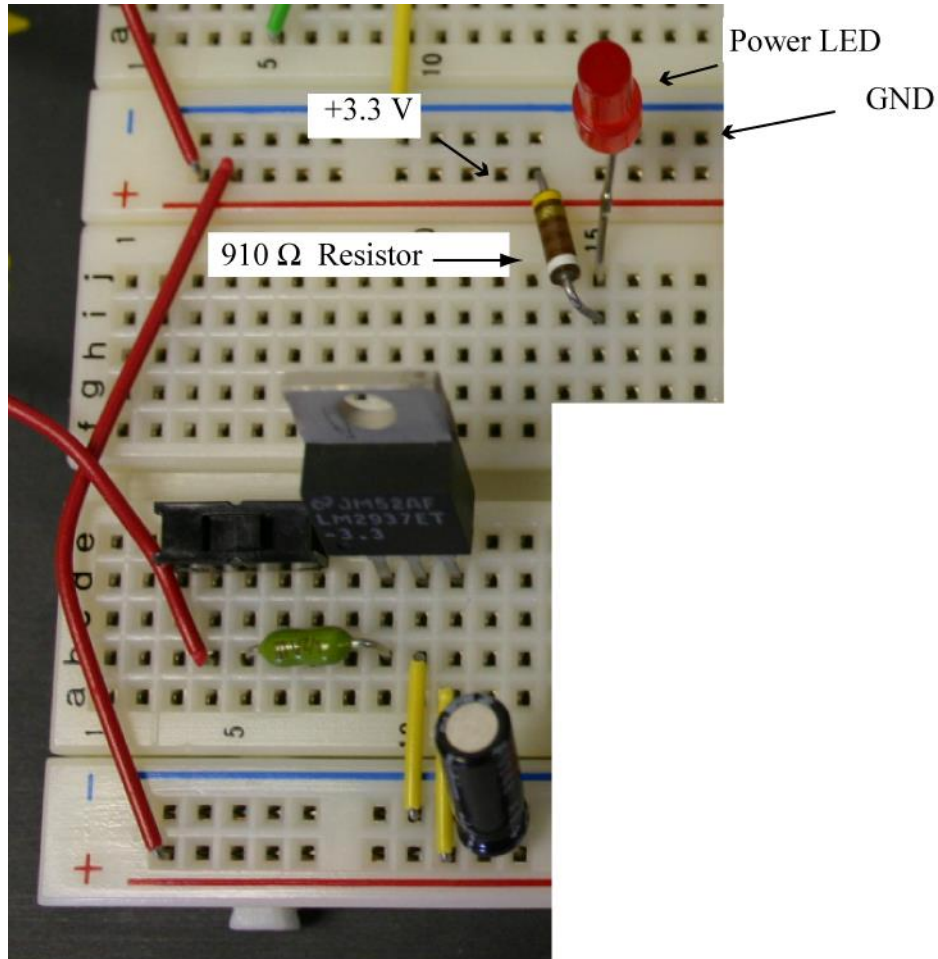
6. Locate an LED and 910 Ω resistor in your parts kit as shown in Figure 8; use this for a power-on LED. Use a multimeter to verify the resistance value.

Figure 8. Power-on LED and 910 Ω resistor



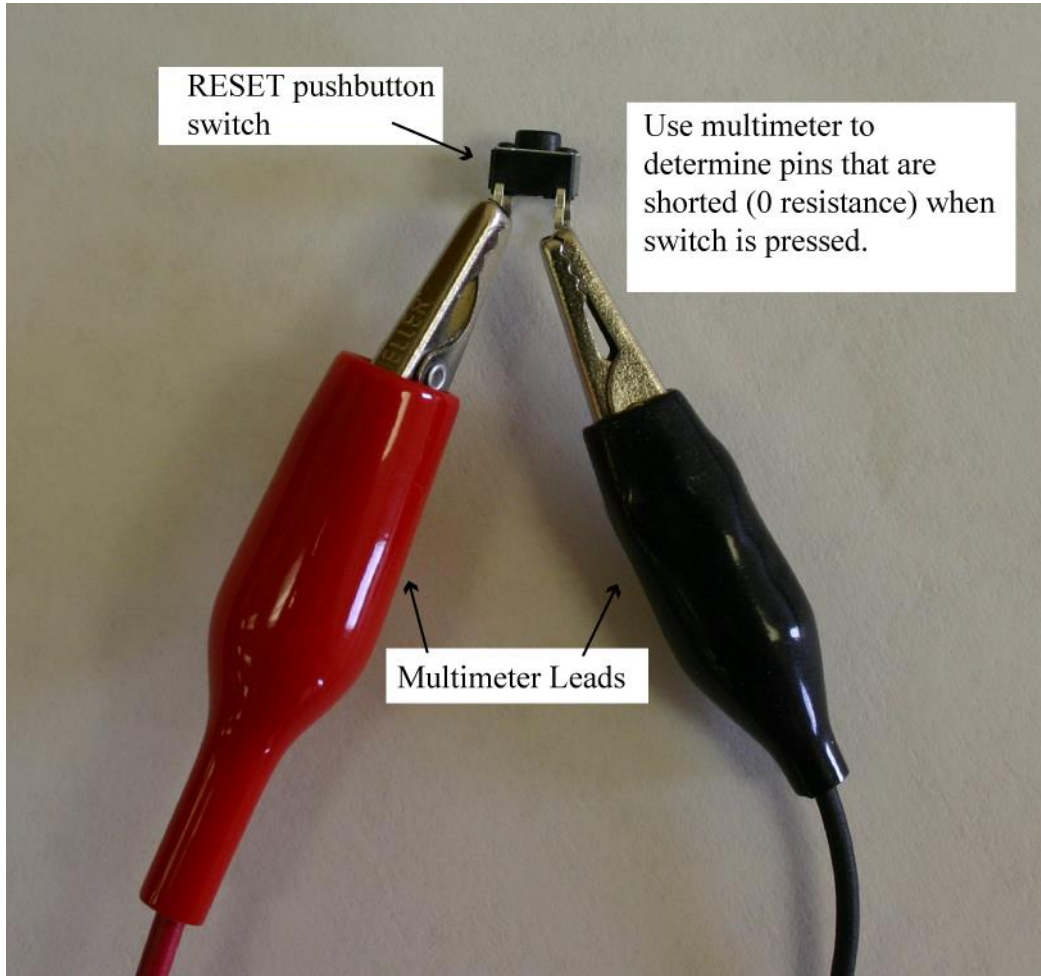
7. Place the power-on LED and the $910\ \Omega$ resistor on the board as shown in Figure 9; the LED should turn on/off with the power switch. If it does not, try reversing the orientation of the LED.

Figure 9. Power-on LED and $910\ \Omega$ resistor placed on board.



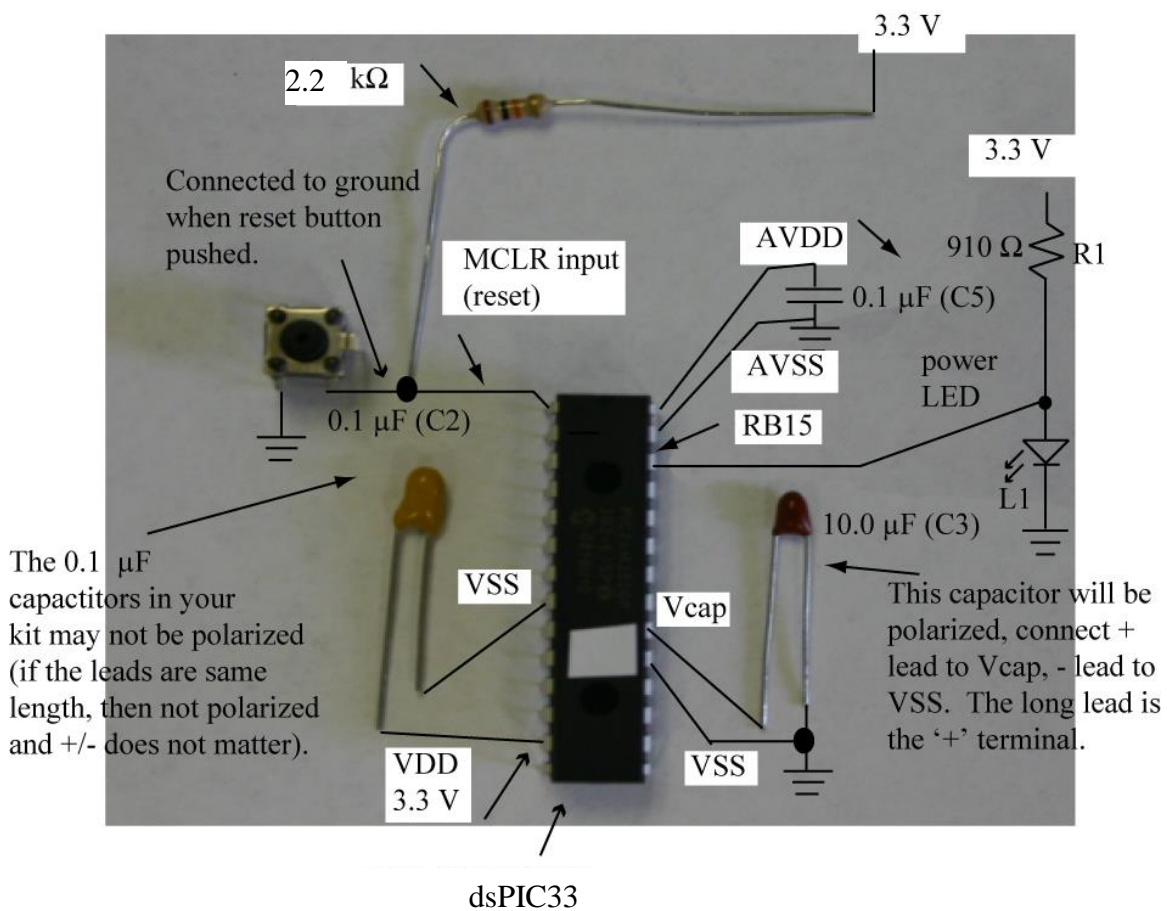
8. The reset pushbutton switch is shown in Figure 10; use the multimeter to determine the pins that are shorted when the pushbutton is pressed. The other two pairs of pins are permanently shorted together.

Figure 10. Measuring resistance of pushbutton switch



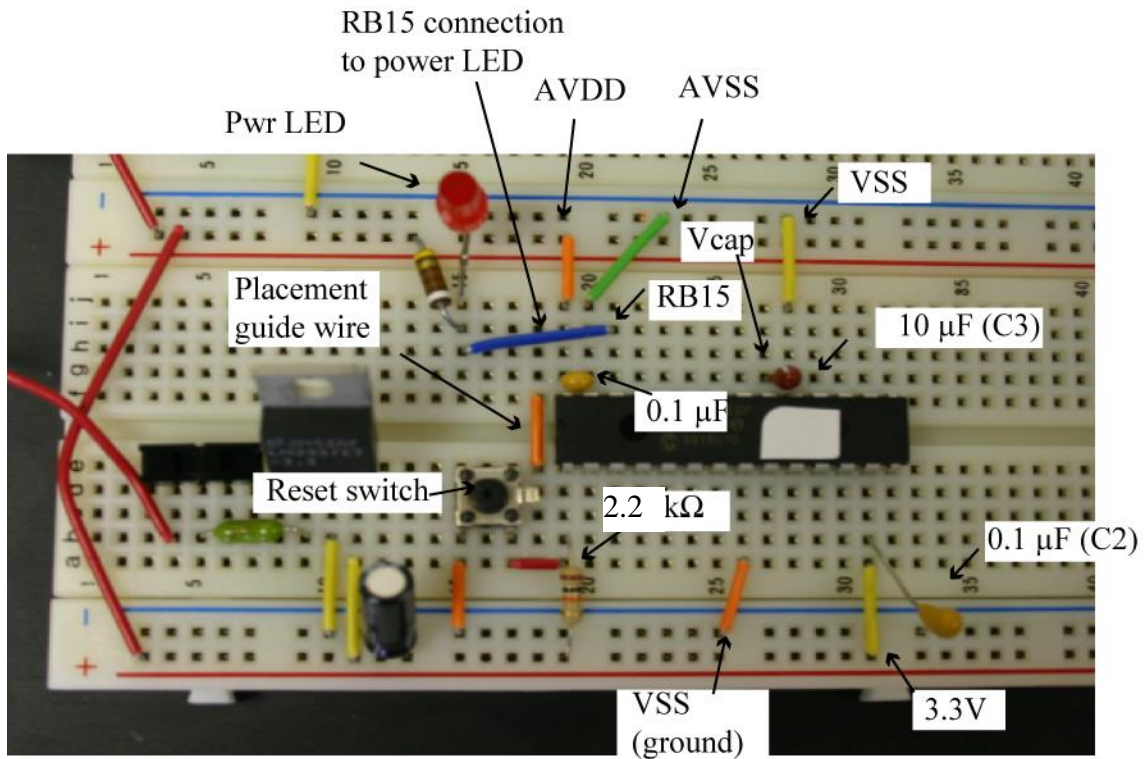
9. Figure 11 shows the 2.2 k Ω resistor and pushbutton switch needed for reset circuit. It also shows the LED connected to pin RB15, and the C2, C3, C5 capacitors. The 10.0 μ F capacitor will be polarized, and it will NOT be a 'can' type, but will be a tantalum capacitor. It should be the only tantalum capacitor in your parts kit. The other 0.1 μ F capacitors in your parts kit will be ceramic unpolarized capacitors (their leads will be the same length). **WARNING: DO NOT CONNECT THE VCAP PIN TO 3.3V - THIS WILL DAMAGE YOUR MICROCONTROLLER!**

Figure 11. Reset circuit components



10. Figure 12 shows the placement of the dsPIC33 and the reset circuit on board. Note that the dsPIC33 has three ground (2 VSS, 1 AVSS) connections, and two power pins (VDD, AVDD). The placement guide wire is important, it helps you to remember where to place the μC if you remove it from the board.

Figure 12. Reset circuit placement



11. Figure 13 shows the USB-to-Serial cable and the 6-pin header for it to plug into on the breadboard. Only four connections are required: MCLR#, TX (RP11), RX (RP10), and GND. Figure 14 shows the serial connection placed on the breadboard; however, **the MCLR# connection is not shown, though it must be connected.**

Figure 13. TTL-to-Serial CMOS Connection for Serial port

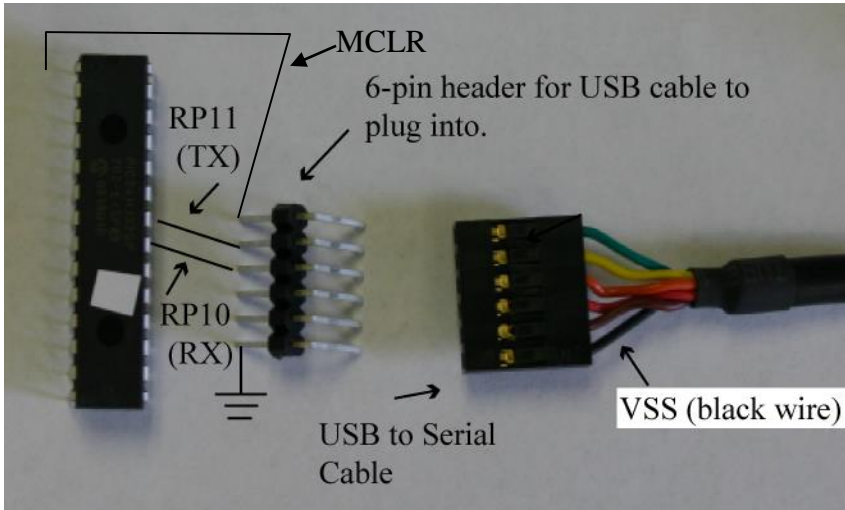
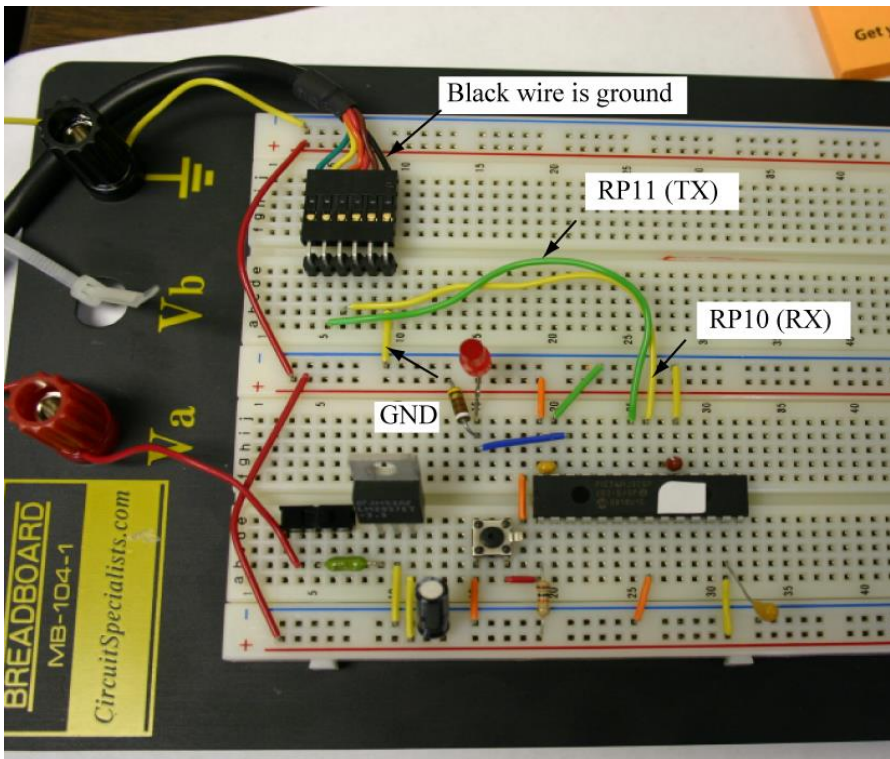


Figure 14. TTL-to-Serial CMOS Connection for Serial port on Breadboard



12. As stated in the previous step, be sure to connect pin 6 of the header (the topmost pin shown in Figure 13; it connects to the green wire on the USB-to-serial cable) to the MCLR# input of the PIC (pin 1 of the PIC), where the pushbutton is also connected. This allows use of the “MCLR# and Prgm” button on the bootloader by enabling the PC to reset the PIC.

At this point, you should be able to download programs into the system using the bootloader program, assuming your μC has been programmed with the bootloader firmware.