

SPIRIT

Phase 2 – Charging Circuit

Computer and Electronics Engineering

Purpose

You have learned about some very fundamental concepts of potential difference, (measured in volts) and current (measured in amperes). A very useful tool in analyzing any circuit is the digital multi-meter (DMM). This tool measures current, voltage, resistance, and in some cases many other metrics. Knowing how to use this tool is very important to being successful in troubleshooting problems with your TekBot.

Project Overview

During the course of this project you will perform the following tasks:

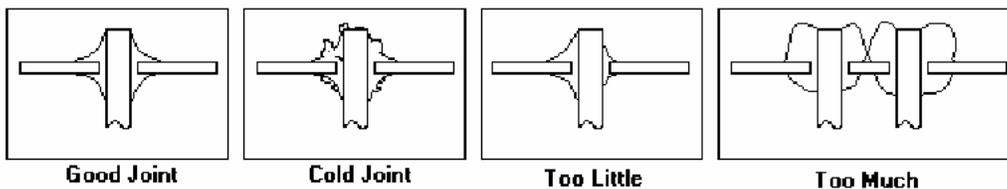
- Construct your charging printed circuit board
- Use your DMM to measure voltage.
- Connect your charger board to your TekBot base.
- Have a good time.

Pre-Project

1. Print out the parts list and silk screen documents.

A Guide to Soldering

Many of you have soldered once or twice before but most likely very few of you have ever been instructed how to recognize ‘good’ soldering from ‘bad’ soldering. There are a few basic concepts involved in soldering that we will detail here, the first being that solder itself melts with heat. Well duh. Part of this simple concept though is the key to making a ‘good’ solder joint. A good solder joint will conduct electricity well and prevent the component from falling out. Since solder melts with heat we need to be sure that we give enough heat to the solder to get it to melt uniformly, not in clumps. A solder joint that was made without enough heat is called a ‘cold joint’ (See the figure below). Another problem can be if there is too much or too little solder used for the joint. If too little solder is used the joint will not be strong enough and will likely break. If too much is used there is a risk of a ‘solder bridge’ being formed (See figures below). An ideal joint should appear ‘shiny’ even after it cools and should look like the solder is stretched from the pad to the wire.



Procedure

IMPORTANT: YOUR NICAD BATTERIES CAN SOURCE SEVERAL AMPS OF CURRENT. IF DURING THE CONSTRUCTION OF THE CHARGER BOARD YOU SHORT YOUR NICADS YOU WILL HAVE USELESS, BURNT, SMOKING, NICADS INSTEAD OF BATTERIES. YOU HAVE BEEN WARNED.

The charger board is a simple board that uses common components to create a way of charging our batteries and to current limit the output from our batteries to prevent accidental discharge. To construct your board we will need to learn about schematics and schematic symbols.

1. The parts list is a list of each of the parts that is used to construct the charger board. Each part has a quantity, name, reference number and associated figure. The reference

Below is a short list of various types of components and the reference designators commonly used.

Resistor	R
Capacitor	C
Inductor	L
Transformer	T or L
Connector	J
Jumper	J
Integrated Circuit	U
Switch SW	SW or S
Fuse	F
Diode	D

number(s) tell you where on the board to insert the components. These reference numbers are also shown on the schematic for each component. The schematic is a graphical representation of the circuit we are building and shows how the components are connected to each other. Each component has a unique shape depending on what it is and every component has a different reference number. The printed circuit board (PCB) silk screen is printed on the circuit board and shows where each of the components are located. It has the reference number and shows the orientation of those components which have to be placed in a specific direction.

Choosing a Component to Install.

When choosing a component to install there are many things you can consider when making the selection:

- Will the component cover another component that has not yet been installed?
- Is the component very fragile or sensitive so it should be installed last?
- Is the component very close to another component making it hard to install the other component?
- Do I have the component available to install?

part goes you should see ‘which way’ it goes into the board. Many components can be inserted in any direction, but others must be placed in a certain way or you can damage the circuit and component. The parts list shows which components must be placed in a specific direction. With component J9 we notice that it is an irregular shape and could only be inserted in one direction so place it on our board and solder it into place. It would look like Figure 1.

2. To assemble your charger board you should begin by choosing a component to install. For our board you can choose any component that you would like first, but your job will be easier if you use a system. Place the components on the components map referring to the parts list for reference number and shape information. This will verify that you have all the components. You then need to double check all of your sheets that you are installing the component correctly. For example: You decide to install the component J9 first (J9 should be installed last, not first). First you look at your **parts list** to see what the component looks like. Then you use your **PCB silk screen** to see where it goes on the board. Now that you know where the

Components that can be inserted in any direction.

Resistors, Ceramic Disc Capacitors, Inductors, Fuses

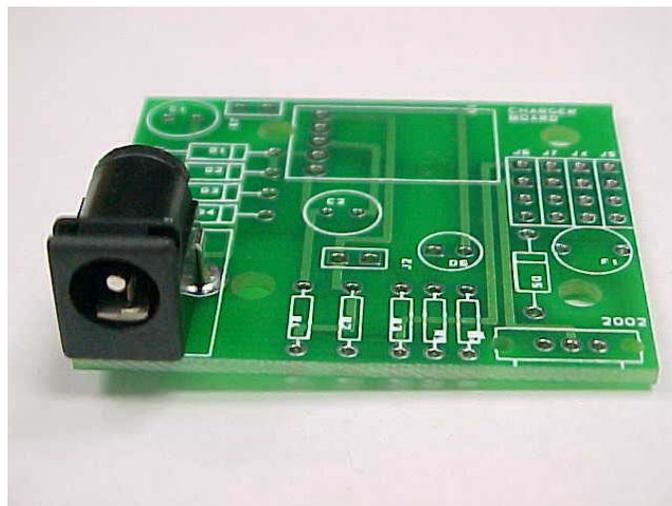
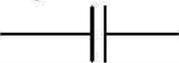
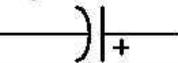
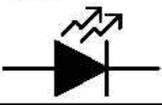
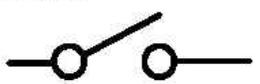
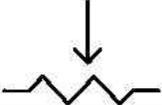
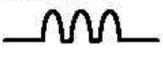
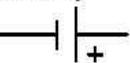


Figure 1. J1 installed, but don't install yours until all other components have been installed.

Common Schematic Symbols		
Below are some common schematic symbols you might see when examining a schematic.		
Diode 	Capacitor – non-polarized 	Capacitor – Electrolytic 
LED 	Switch 	Power Jack 
Potentiometer 	Fuse 	Ground 
Resistor 	Inductor 	Battery 

3. Another way of checking that you are inserting a component correctly is to use your **schematic** for reference with your PCB. Using this method ensures that you are correctly connecting pins on components. Let's use D3 as an example. D3 is a type of diode known as a Light Emitting Diode (LED). It is intended to work with current flowing through it in only one direction so it is important that it is inserted correctly. If we look at our **schematic** we can see that pin 1 of D3 is connected to pin 1 of J8. Pin 2 of D3 is connected to R7. Looking at our PCB however we can see that there is no marking telling us which way to insert D3. To figure it out, follow the copper trace from one of the holes for D3 to see what it connects to. If we follow the hole closest to the outside of the board we find that it connects to J8 pin 1. So we must need to insert D3 into the hole with pin 1 in this location. This brings up our next question; which pin of D3 is pin 1? To figure this out again consult your schematic. Notice that the schematic symbol for D3 has a unique shape with an arrow close to pin 1 and a vertical line close to pin 2. These are called the Anode and the Cathode respectively. For LEDs the longer lead on the component is the anode, or in our case pin 1. So we would insert D3 with the longer lead into pin 1 and solder into place.

Common ways to distinguish pins on components.

Electrolytic Capacitors: These components commonly have a small strip on one side that designates the negative lead on the component.

Diodes: Diodes (other than LEDs) will commonly have a stripe around them on the end that is the cathode.

LEDs: LEDs commonly mark the anode with a longer lead and the cathode with a small flat part on the side of the plastic housing.

I recommend installing the lowest profile components first progressing to the taller components. When you turn the PCB (printed circuit board) over to solder, the bench top will help hold the components in position. Start by placing R1 on the circuit board. **Measure the value of the resistor to verify you have the correct one.** It is much easier to double check the value than to try to unsolder and remove a component. A close-up photo of a completed board is on the course web site. Use it to verify the placement of components. Spread the leads slightly so the component does not fall out when you turn the board over to solder. After soldering use the side-cutters to trim the excess lead. **Wear safety glasses and hold the lead that you are cutting.** If you don't hold the lead, it will be launched upward toward your eyes and/or anyone else's in the vicinity.

Next place R2 and R8 on the board and solder. These are very close to each other on the board. Too much solder will bridge the gap between them and too little will result in a poor connection.

Finish the resistors with R3, R4, R5, R6 & R7.

Diodes are the next components to be installed. The orientation of diodes is important. If they are installed backwards, the circuit won't work. The cathode (-) end is marked with a band. The silk screen shows the band also. D3 is a light emitting diode (LED). See the discussion above. The longer lead is the anode (+) and is inserted into the hole adjacent to the + sign.



Ohm Meter Measuring Resistor

The longer lead is the anode (+) and is inserted

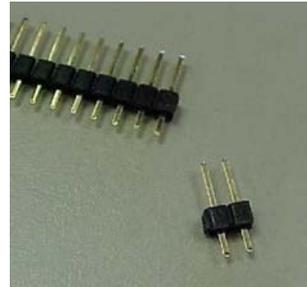
4. Go ahead and complete your charger circuit using the information above. You will want to be very careful when you insert U1. Putting it in backwards is bad news.
5. When you decide to insert J1 –J8 you will need to use the header strips in your kit and cut the amount of header you require from the strips. You will need one 2 pin male header (J8), one 2 pin female header (J7) and 2 six pin female header pieces. Note on the photo of the charger board how the two six pin headers are located.

Testing the System

Have your instructor check your system before continuing.

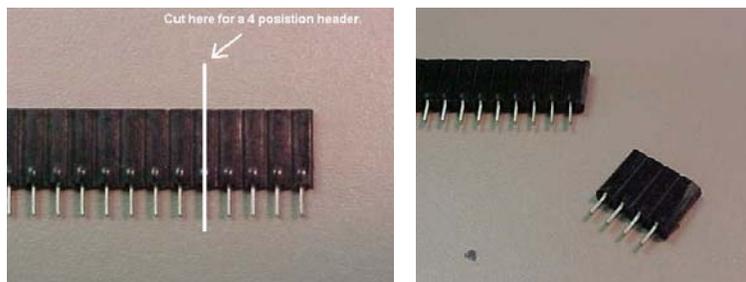
Making Male Headers/Jumpers

To make J2 you should find a long strip of ‘male header’. You simply need to use a pair of diagonal cutters to clip off as many pins as you need for the jumper.



Making Female Headers/Jumpers

All you need to do to make female headers is to ‘clip’ sections off of the female header in your kit. Simply count the number of pins you need and cut the next pin in half. Be careful to not cut too close to the piece you are keeping or it may break when you cut it off. A good tool for doing this is a pair of diagonal cutters. If you would like you can then clean the edges with a sharp knife



After you construct/design anything a crucial next step is to test what you have done. The more rigorous the testing the more certain we can be that our system works. Now we are going to use our Digital MultiMeter (DMM) to check our system with voltage and current measurements.

1. Check the voltages of your circuit first. To do this first select the ‘voltage setting’ on your DMM. We want the range to be set to the 20V setting. We want to be measuring DC voltage as well, not AC voltage. In Figure 2 we can see a picture of one DMM you might use. Most DMMs are very similar so yours will probably look very similar.



Figure 2: The settings dial on a common DMM.

AC vs. DC Measurements.

Most DMM allow for you to measure both Alternating Current (AC) and Direct Current (DC) voltages. To allow for this functionality most meters have either a switch to change settings or two separate voltage scales. More advanced meters will automatically detect which type they are measuring and respond correctly. Your meters have a switch.

- Now you want to make sure that the 'probes' on the DMM are in the correct locations for measuring voltage. The two positions are the 'common' and the 'V' position. Figure 3 shows the correct placement on the sample DMM.

Voltage vs. Current Measurements.

Most DMMs allow you to measure the current flowing in a circuit and the voltage across components in a circuit. These two function though are fundamentally different so many DMMs make you change where on the meter your probes are plugged in when you switch from one measurement to the other. Advanced meters may not require this switching. Again ask your instructor what the functions of your meter are.



Figure 3. The positioning of the probes on a DMM for measurement.

- Now that our DMM is set up we are going to take our first measurement. Go ahead and plug your DC adapter into the wall and then into your charging board. Be careful to not set your charger board on anything metal that might short out the board.

The first measurement we want to take is to measure the incoming voltage to the voltage regulator portion of the charger board. To do this you will need to place your probes with the

'common' probe on a ground, and the 'V' probe on the test pad T2. Notice from the schematic that T2 is connected to the node with C3 connected to it. Also note that there is no test point marked as ground. You can place a wire in one of the female sockets connect to ground on J1-J6. If your meter is turned on, you should see it display a voltage between 8 - 16V. Enter your measurement in the space below. If your voltage is 'negative' double check that you have the common probe and the V probe in the correct locations and not reversed.

Voltage at T2: _____

Now measure the voltage the charger board which will be used to charge the batteries. This is the voltage at test pad T1. It should be about 8-9 volts. Write your measurement in the space below.

Voltage at T1: _____

(Make sure that your jumper J3 is shorted with the included shorting block, or you won't get the correct voltage.)

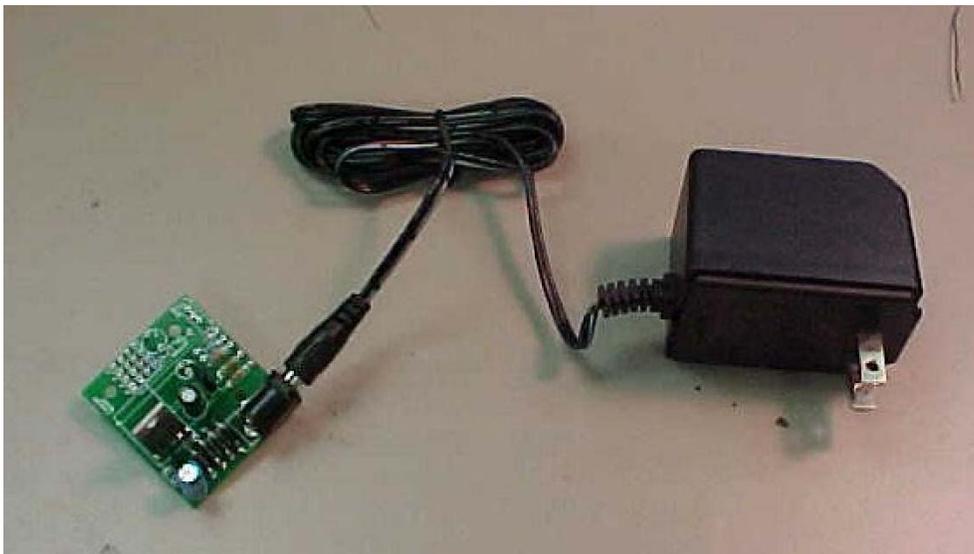


Figure 4. DC adaptor plugged into the Charger board.

Installing your charger board on your robot

1. Use 3 4-40 ½" metal machine bolts, 3 4-40 round spacers, and 3 4-40 metal nuts to mount the charger board to your aluminum base. Align your charger board as shown below and attach it. **DO NOT MOUNT THE BOARD DIRECTLY TO THE METAL BASE; USE THE 4-40 SPACERS.** Figure 5 shows an attached charger board.

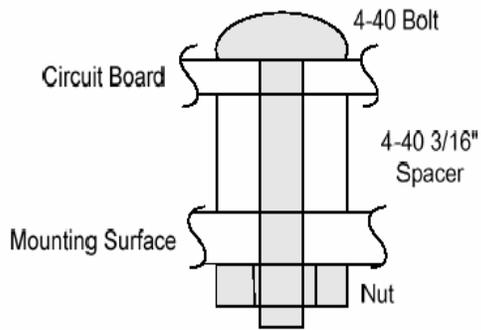


Figure 5: Diagram of spacer usage and base with connected charger board.

2. Now you need to connect your batteries to the charger board. Notice on the silk screen there is a small '+' symbol. The red lead from your batteries should be closest to this symbol when the connector is inserted. If you reverse this plug you **WILL DESTROY** your board and you will need to purchase a new one. Plug the battery connection into your charger board.

Copyright C 2006
University of Nebraska

This document is based on documents that are the property of Oregon State University and the School of EECS. Limited use is allowed according to the following criteria; materials are free to use except for cost of reproduction but must bear this statement. Materials created using this information may not be labeled as TekBots materials with out the prior written consent of Oregon State University and the School of EECS.