

2.996/6.971 Biomedical Devices Design Laboratory Lab 1 and 2

Distributed: Sept 5, 2007

Due: Monday, Sept 17, at 5pm

Staff availability: Sept 10 and 17, 2-5pm

Goals

- Understand the standard block circuit
- Develop a design of your custom block or use the reference design
- Understand electronics packaging
- Make sure the components in your design are available and can be hand soldered
- Learn to use the PCB Artist software
- Create a library of components for your design
- Create the schematic diagram for your design

Conceptual design of your custom block

The goal in the first part of the lab is to develop a conceptual circuit that will provide inputs or outputs to the main circuit blocks. Input blocks can include components such as buttons, magnetic sensors, accelerometers, light sensors, optical encoders. Output blocks can include components such as LED's, relay drivers, or simple motor drivers. Of course, one form of output is transmitting data through the USB port, which is already implemented by the standard block.

Please research the components that you will use carefully. Download the datasheet from the manufacturer or the Digikey website. Determine all the necessary supporting circuitry. For example, the standard block uses a 3.3V power supply, and if the components you need require a different power supply voltage, than a voltage regulator is needed. See staff regarding help in this regard.

The total cost of the components used to make this circuit should be less than \$15 and the components must be in-stock and available in individual quantities. If the components are found on Digikey, we will purchase them for you. Parts not available at Digikey are your responsibility. Often times, it is easier and faster to obtain parts as samples directly from the manufacturer. However, you must make sure that the manufacturer's lead-time is compatible with the schedule of the lab and you should place your sample order as soon as possible.

Please note that the purpose of the lab is to take you through the process of creating a working PCB, and not to develop a new product (that's for the project part of this class). Therefore, it is important to keep the complexity of your design to a minimum in order to ensure that you have sufficient time to create a working design and complete the lab, which will be useful for your project. All designs will be subject to approval by staff. Be sure to speak with one of us if you have questions in this regard.

For folks who are new to electronics design, an example design has been created for your reference. It is entirely appropriate to copy some or all of this design as your custom circuit block. The reference design, shown in Figure 1, includes a 3-axis MEMS accelerometer and several

LEDs controlled by the microprocessor. The LEDs are driven from a hex-inverter powered at 5V in order to allow it to drive blue and white LEDs. The hex-inverter is from the HCT logic family, which means that it can be switched using 3.3V logic. It is represented on the schematic by six (“hex”) logical NOT (“inverter”) gates, U9a-U9f. When the input is low (0V), the output is high (5V), when the input is high (3.3V) the output is low (0V). Although it has several symbols on the schematic, the component itself is a single IC, as you will see in the PCB layout lab.

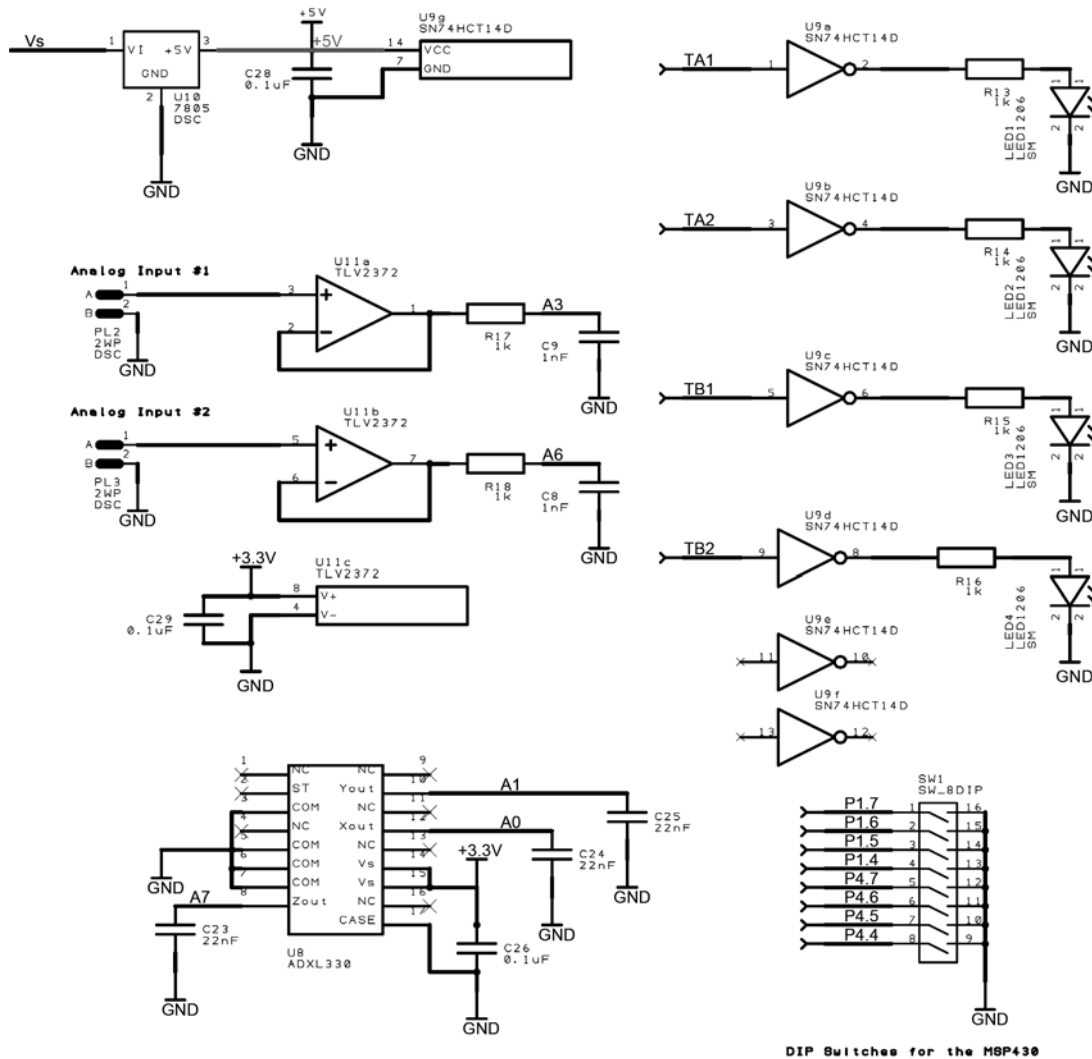


Figure 1: Custom block reference design

Selecting component packages

Once your a conceptual design has been formulated and you’ve checked that the components can be obtained within a reasonable amount of time, the next step is to select specific packages of the devices that will be used. There are two main classes of packages in electronic components: Through-Hole (TH) and Surface Mount (SMT). TH packages contain wire leads that are designed to be soldered into metal-plated holes on a PCB. SMT packages have leads that lay flat on the PCB. SMT packages are smaller than TH packages and are generally found on newer devices. In


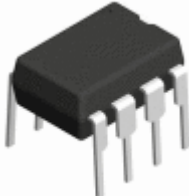

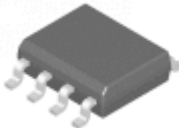
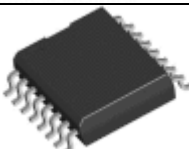
fact, many new devices are exclusively packed in SMT. We will use SMT packages whenever possible because they are also easier to solder and de-solder.

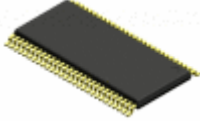
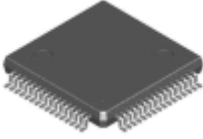
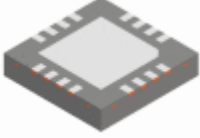
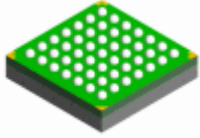
The SMT packages for passive components, such as resistors and capacitors, are designated by a number that describes their size. For example, a common package for resistors is 1206, which is 120 mils long by 60 mils wide. Common sizes include 1206, 0805, 0603, 0402, and 0201. Note that these numbers are sometimes given in metric. The standard sizes with their metric equivalents listed in parentheses are 1206 (3216), 0805 (2012), 0603 (1608), 0402 (1005), 0201 (0603). We will generally use 0603 for most resistors and capacitors in this class because this package is a good compromise between size, solderability, and having a wide range of available values.

The SMT packages for active components, such as transistors and integrated circuits, come in a variety of sizes and shapes. A summary of different SMT packages are shown in the table below. Note that some components are difficult or impossible to solder by hand and therefore should be avoided in your designs.

Selected Electronic Packages

(Source: <http://www.national.com/packaging/parts/>)

| Package Name | Picture | Comments |
|-----------------------------|---|---|
| Through-hole |  | |
| DIP – Dual Inline Package |  | Standard old school through-hole IC package |
| SIP – Single Inline Package | | Typically used for connectors |
| SOT-23, SOT-223, SC-70 |  | SMT packages for transistors and single op amps |
| SOIC |  | The largest SMT package for op amps, easy to solder |
| SSOP |  | Easy to solder |

| | | |
|------------------------------|---|--|
| TSSOP – thin version of SSOP |  | Easy to solder |
| TQFP – Thin Quad Flat Pack |  | Difficult to solder, but can be done |
| LCCC |  | Difficult to solder. Hand soldering requires larger than normal pads |
| BGA – Ball Grid Array |  | Un-solderable by hand, require reflow oven |

PCB design workflow and “PCB Artist” software

The PCB design workflow generally involves three types of files: schematic, layout, and library files. The schematic is the logical diagram of the circuit components and connections. The layout includes drawings of the metal, soldermask, and screen printing patterns of the PCB. Library files links the schematic and layout together to enable layout-versus-schematic (LVS) checking to ensure that logical connections are implemented correctly. LVS check is critical to creating the PCB since it is very difficult to verify the design of large scale metal patterns by eye. However, in order for LVS to work correctly, the library files must be error-free.

We will be using “PCB Artist” to create the our PCBs. PCB Artist is a free software package created by Advanced Circuits, a prototype PCB manufacturer in Denver, Colorado. Please download the latest version of PCB Artist from Advanced Circuits’ website (www.4pcb.com). This program is also available in the Der Torossian computer lab (3-462). Please work through the tutorial to familiarize yourself with the program.

Create a library of components

Once you have completed the PCB Artist tutorial, it is time to create library entries for the components in your design. The libraries in PCB Artist are structured as three different files including a schematic library, a PCB library, and a component library, as shown in Figure 2. The schematic library contains the schematic symbol and a numbering of the connections to the component. The PCB library contains drawings of PCB footprints for the packages with each pin in the package identified by a number. The component library links the schematic symbol with the PCB footprint and names each pin. Since one component can come in many packages and standard packages are shared by many components, this scheme enables maximum reuse of both schematic symbols and PCB footprints.

PCB Artist comes with a good selection of standard libraries. Browse through the component library to find your components. We have created a set of libraries featuring many of the components in the standard block identified by the filenames: **BDDL07.SSL**, **BDDL07.PSL**, and **BDDL07.CML**. The standard libraries will not have all the components that you will need. You will create your own set of schematic, PCB, and component libraries. Fill your library by first determining the PCB footprints that you will need. If the components in your design use standard footprints, it is very likely that these footprints are already available in the standard libraries. For example, if you are using an op amp in 8-pin SOIC package, you can get this footprint in the PCB library – **PROLIB.PSL** listed under “SOIC8”. Whenever you are using an existing PCB footprint, you should double check the pitch of the pads to make sure that the footprints that you are using haven’t been mislabeled. Using incorrect footprints is one of the quickest ways to make a non-working PCB. If you cannot find an existing PCB footprint for your part, you should go to your PCB library and create a new footprint by going through the wizard using the specifications provided in the datasheet of the component. There are two very important things to make sure of when doing this: 1) The correct units (mils or mm) are used to create the footprint; and 2) the pins are numbered in the same way as the drawing in the datasheet. Some components require additional pads to mechanical fixturing (see the USB-MINI-B footprint in the class PCB library. Add these pads in the PCB footprint and label them after the highest available pin number.

Once you have created or found the PCB footprints that you need, the next task is to create the schematic symbols. Standard symbols, such as op amps, can be copied into your own library. If you are drawing a schematic symbol from scratch, make sure the symbol matches that of the schematic symbol in the datasheet.

After both PCB footprints and schematic symbols are created, you are now ready to associate them using a component library. This part should be very straightforward. After you have completed this step, enter manufacture and distributor information for each component by clicking on the “Values...” button while inside tab for the component library. This information will be useful when creating a bill of materials for your PCB.

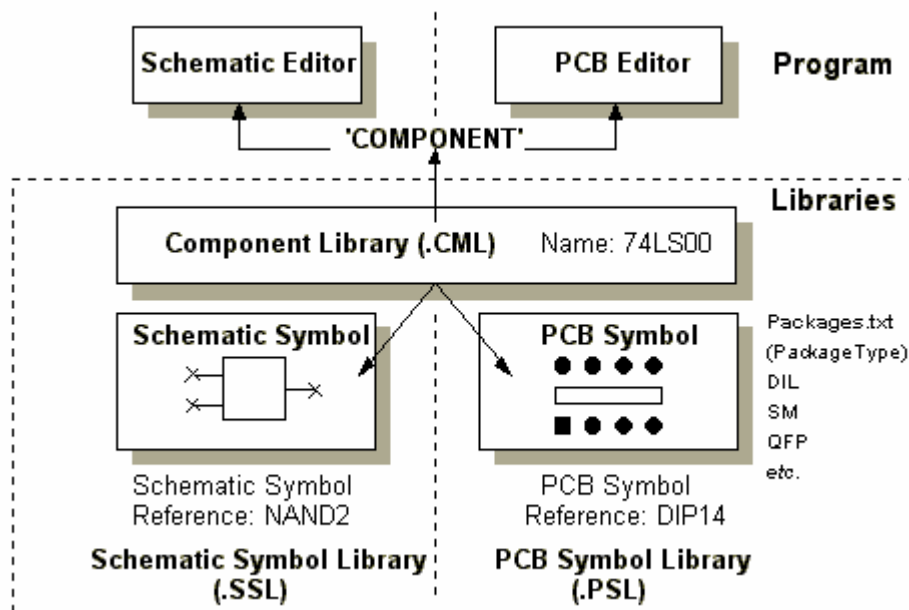


Figure 2: Library structure in PCB Artist

Create the schematic diagram

Once your component library is complete, you are now ready to create the schematic diagram. The schematic for the standard block should be copied exactly. It is important to include the breakout connectors for the microprocessor, which will allow the PCB to be modified and adapted with ease. Also, notice the zero ohm resistors that are designed to allow the communications channels to be reconfigured. There are a number of important things to keep in mind when drawing the schematic:

- Each active component (e.g. op amp, transistor, microprocessor) must be accompanied by a 0.1 μ F ceramic capacitor located near its power supply pin. We will discuss the reason for this capacitor specifically in lecture.
- Check that the power and ground connections for each active component are connected. There's nothing more embarrassing than forgetting to hook up power. A good check is to highlight the power and ground nets with a different color (see Fig. 1).
- Look for potential bad connections on the schematic. See examples in Figure 3. You can use the "Unconnected Pins Report" to aid your search.
- It is best to protect the digital connections between the microprocessor and other chips using a current limiting resistor
- Make sure logic levels between devices are consistent. 5V output to a 3.3V input = KaBoom!
- Double check that the schematic symbol and the PCB footprints are consistent.

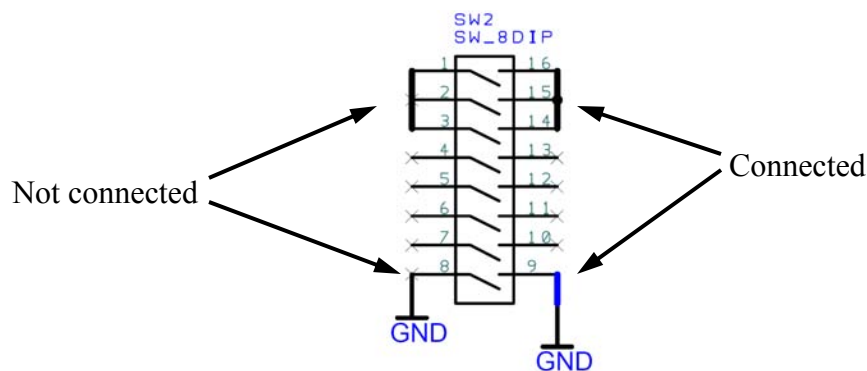


Figure 3: Common hookup errors in PCB Artist

Lab Check-off

All designs will go through a 5-minute design review with staff. Your design must be approved by staff before you can move onto the next lab.

Lab Write-up

The lab write-up should be brief and informative. The purpose of the write-up is to record information about the design for future use. Please include the following in your lab write-up:

- Description of your circuit
- List your component choices and explain the reasoning behind each one. For example: value of the current limiting resistor for an LED, components that you used for level shifting, extra power supplies that you've added
- Schematic diagram
- Bill of materials
- Estimate the total current draw of your circuit, list contributions from each major component. Confirm these requirements are met by the voltage regulators.