Lab 8g Design and Layout of an Embedded System

Goals • To design an embedded system,
• To study issues of power, clock, reset, and programming for a embedded system,
• To layout a PCB board.

Review • Data sheets for your microcontroller,

Starter files **EE345L** library for PCB Artist, **Lab8BOM.xls** **(see the latest version)**

Background

You will use the CAD program **PCB Artist** to layout an embedded system. The software to activate the interface, hardware construction, and device testing will be performed in Lab 11. There will be an open house (like a science fair) to demonstrate your Lab 11 systems to the academic community. The design of the system must satisfy certain requirements (things it must do) within a set of constraints (limitations dictated by the realities of the project). Rather than simply redesigning one of the previous labs like you did in Lab 6, this embedded system must do something useful. There are some options listed below, but you have flexibility to define exactly what it is to do.

Option 1. You can make a digital hearing aid. Basically you need to interface an electret microphone, sample data, perform signal processing, perform noise cancellation, output to a DAC then amplify and filter the signal to head phones. For more information on this product, see.

http://users.ece.utexas.edu/~valvano/EE385J/HearingAid.pdf
http://focus.ti.com/lit/an/spra657/spra657.pdf

Option 2. You can make a hand-held game. You can use either LEDs or an LCD as the display. Inputs can come from switches, accelerometer, joystick or slide pots. Sound output will add an important quality to the game.

Option 3. You can make a voice recorder. You will interface a microphone, switches and a DAC to the microcontroller. When you press the record switch, the analog voltage is sampled by the ADC and restored in the system memory. You can use either internal RAM or internal flash EEPROM to store the data. When you press the other switch, the previously recorded waveform is output using the DAC. One option in this project is a heart sound analyzer created by placing the electret microphone inside a stethoscope.

Option 4. You can propose to your TA to design, implement, and test any microcontroller-based system demonstrating the educational objectives of this class. You will need specific approval from your TA for this option.

Requirements

• A microcontroller must be embedded into the system,
• A PCB layout of the system must be used, having been created with **PCB Artist**
• Each group of two will produce one PCB layout (a SCH plus a PCB file),
• There must be at least two inputs, two outputs, and two interrupt service routines,
• The final system (Lab 11) will be an actual device with chips soldered onto the PCB,
• The system should perform something useful similar to the above options.
TAs will judge if the project is sufficiently complicated.

Constraints

• Each **PCB** must be done using the $33 student service (we actually pay $25 plus $16 shipping),
• There will be a specific list of parts that we will be willing to give you for building this system,
• You must purchase, borrow, or get free samples for any additional parts that you require.

Implementation 1. We expect most students to layout the I/O part of the system on the PCB and connect it to your TechArts 9S12DP512 board using two Samtec SMH-125-02-G-D connectors. If you have not ordered the free Samtec parts, try to get them from another student. If you have extras, feel free to give them to your classmates. If your I/O devices require less than 100 mA, then you can route power from the Tech Arts board into your I/O board. If your system requires between 100 and 500 mA, you can place a 78M05 regulator on your I/O board and route power from your I/O board into the Tech Arts board. If your system requires more than 500 mA, see your professor.

Implementation 2. If you have a 9S12C32 board leftover from EE319K, you can place the Nanocore module on your I/O board. Special considerations about power and programming must be addressed if you plan to use this option, so please discuss this with your professor or TA. Connect a DB9 connector so you can run the debugger.

Implementation 3. You can implement a complete system putting the 9S12 itself onto the PCB, like Lab 6. This option requires some delicate surface mount soldering, so please discuss this with your professor or TA. This lab should not be the first surface mount component you have soldered.

Implementation 4. You can implement a complete system using another microcontroller as long as the entire system is soldered onto one PCB. This option requires developing a plan about how the system will be programmed and debugged, so please discuss this with your professor or TA. The TI MSP430 is a good option. In particular if you choose develop your system using a MSP430F2012 or MSP430F2013, we will give your team a MSP430 development kit donated by Texas Instruments. For more information on the development kit, see

http://focus.ti.com/docs/toolsw/folders/print/ez430-f2013.html
http://focus.ti.com/lit/ug/slau176b/slau176b.pdf

To get the free development kit donated by Texas Instruments, please show the professor your pre-preparation. The MSP430F2012 (10-bit fast ADC) and MSP430F2013 (16-bit slow ADC) have 2 KiB of EEPROM, 128 bytes of RAM, and 10 I/O pins. The actual project can be created with all through-hole soldering, because these microcontrollers can be obtained as free samples in a 14-pin PDIP package.
http://focus.ti.com/lit/ds/slas491d/slas491d.pdf
If you choose this project, please sample one of these four (N stands for plastic DIP package). If you wish to use surface mount versions of the MSP430, please get approval from your TA.

MSP430F2012IN, MSP430F2012TN, MSP430F2013IN, MSP430F2013TN

If you choose this project, please also sample a connector allowing you to interface your PCB to the debugger

SLM-104-01-G-S

There will be a “Science Fair”-like public demonstration for Lab 11. We will present special awards to the team of two with the best design. The preliminary round will be judged by your TA, and the final round will be judged by an independent panel (e.g., Daryl Goodnight, Paul Landers, Carole Bearden, and Perry Durkee.) Some students will put extra electronics off the PCB, because it doesn't all fit on the PCB. If you do have off-board electronics, then you will need a connector or something to create the bridge. You can get good grades in Labs 8 and 11 with off board electronics, but you will not be eligible to win "best design". In particular, your grade depends on if the required tasks are completed on-time, and if your eventual project (I/O, microcontroller, and software) works. However to win "best design" you will need to meet the following restrictions:

All electronics (resistors, capacitors, ICs) are on the PCB with or without a TechArts board
LCD displays, switches, LEDs, speakers, keypads and microphones can be off the PCB
Your team of 2 spends less than $30 on extra components (which are readily available to all students).

The file Lab8BOM.xls lists parts you can find in the professor’s office. For example:

resistor 1/8W 5% 2.7K OHM [Jameco 108062] $0.0069
resistor 1/8W 5% 10K OHM [Jameco 108126] $0.0069
0.47 uF Tantalum 35V 10% caps [Jameco 332137] (0.1in footprint) (tantalumCap.pdf) $0.23
0.1uF ceramic capacitors, bypass cap for each chip, (0.1in footprint) $0.05
B3F tactile push button switch (B3f-1059.pdf) $0.17
1.6V 1mA HLMP-D150 LED (LEDHLMP-D150.pdf Digikey 516-1323-ND) $0.173
2-pin header with 2-pin jumper AllElectronics SBC-2 and SBH-2 , $0.90
2-pin cable for TechArts power plug, [CON-242], $0.70
Test point, black, [Digikey 5001K-ND], $0.29
Test point, red, [Digikey 5000K-ND], $0.29

It is possible to have external I/O devices, like speakers, switches, thermistor, and/or an LCD off the PCB and still win "best design. In particular, you can use a LCD that you check out from the second floor, but materials checked out from the second floor must be returned. If you want additional components that I do have (LEDs, switches, thermistors, connectors, resistors, capacitors, speakers, boxes) you need to come to my office and show me your SCH file, circling or listing the needed components. Components from professor’s office do count against your maximum of $30 additional components (but you do not have to actually pay for them.) Free samples, components in the original bag of parts, the PCB, batteries and components checked out of the second floor do not count towards your $30 limit, but should be listed with fair market price in Procedure e)

Pre-preparation (your TA will announce due date for this part)

Part a) Write a one-page **Requirements Document** for the system. Refer back to Lab 6 for general information about requirements documents. Please email this to your TA (due date announced in class). Please use this outline

1. Overview
- **1.1. Objectives:** Why are we doing this project? What is the purpose?
- **1.2. Roles and Responsibilities:** Who will do what? Who are the clients?
- **1.3. Interactions with Existing Systems:** Include this if you are connecting to Tech Arts board

2. Function Description
- **2.1. Functionality:** What will the system do precisely?
- **2.4. Performance:** Define the measures and describe how they will be determined.
- **2.5. Usability:** Describe the interfaces. Be quantitative if possible.

3. Deliverables
- **3.1. Reports:** Simply state the reports for Labs 8 and 11 will be written
- **3.2. Outcomes:** Simply copy/paste the Lab 8 and Lab 11 deliverables.

Preparation (do this before your lab period)

Part b) Create a **bill of materials** and physically collect all components required for the system. This includes switches, ICs, LEDs, LCDs, resistors, capacitors, and connectors. It is not necessary to put the system in a box, but if you plan to use a box, you should have the box available at the start of the lab. It is not necessary to power the system using batteries, but if you do plan to use batteries, you should have the necessary cables and/or battery holders.

Part c) Draw the circuit diagram (SCH file) using PCB Artist. Refer to the suggestions and guidelines described in Lab 6. Be careful not to load the CMOS outputs of the 9S12. The data sheets of many ICs offer suggested values and placements of capacitors, in order to improve performance. Add features such as test points and labels that will make it easy to test the hardware.

No net names that begin with **N0**;
All components need labels (e.g., U1 R1 C1 J1 etc.), shown both on the board and the circuit diagram;
Each IC should have a bypass capacitor, placed on the PCB as close to the chip as possible;
For resistors, specify wattage (1/4 or 1/6 watt) and tolerance (5% carbon or 1% metal film);
For capacitors, specify type as 5% C0G ceramic, 10% X7R ceramic, 20% Z5U ceramic, or 10% tantalum.

Part d) Within PCB Artist, execute *Tools->SchematicPCB* and select *Translate to PCB*. Place all components inside the PCB area. See
http://users.ece.utexas.edu/~valvano/EE345L/Labs/Fall2010/PCBOrderProcess.pdf

You should have a TA or professor certify your SCH circuit is valid before you begin putting traces onto the PCB. Neither the TAs nor the professor have the time or ability to verify accuracy of your PCB. However, we should be able to advise you on your I/O circuits.

Procedure (do this during your lab period)

Part a) Please build and test on a protoboard a prototype of your hardware circuits. Although you can adjust resistor and capacitor values in Lab 11, it will not be possible to add or change ICs. Write just enough software to verify the functionality of the hardware. You must be able to verify there are no major design flaws in your system.

Part b) Please finish the PCB layout using PCB Artist. The SCH-PCB file combination must pass all the design rule checks. The size limit is 30 in^2. You may not create two designed on one PCB and cut it in half.

Make sure the Snap to *Grid* mode is active (experiment with different settings of the snap)
Add Top Silk labels for your initials, your TA's initials, the date, and the purpose of the board,
Place all though-hole components on the top side (surface mount components can go on either side),
If possible align all chips in the same direction,
Configure the board so that all though-hole soldering occurs on the bottom side,
Add Top Silk labeling to assist in construction and debugging,
Add test points at strategic points to assist in debugging,
- Either by placing two 0.029 in holes 0.1 in apart then soldering a U wire into it,
- or by making a 0.043 in hole then soldering a test point into the one hole

Avoid 90-degree turns, convert them to two 45 degree turns,

Before you submit the board to your TA, you should go through all the steps leading up to actually buying it. I.e., register an account, upload your board, but do not give them a credit card. If you could have purchased your board for $33 plus shipping, then we will be able to have it built. See the **PCBOrderProcess.pdf** instructions.
Part c) You should print both sides of the PCB layout (with the mask). See Figure 6.3. Create a mirror image of the bottom layer and glue/attach the two pieces of paper to Styrofoam, cardboard, or wood. You will have to punch or drill holes in order to place components on this simulated "PCB". Create a complete 3-D mockup of the system placing the actual components on this Styrofoam/paper/cardboard "PCB". Do not place the statically sensitive parts on the mockup. See Figure 8.1. 3-D spacing will be critical if the system will be placed in a box.

Part d) Please estimate or measure the supply current required by the entire system, including microcontroller and I/O devices.

Part e) Please estimate cost of the entire system, this will include fair market cost of parts given or lent to you by UT, parts you use that you already owned, fair market cost of any free samples, the cost of PCB manufacture ($25 for board+$16 shipping), and any parts you purchased. Use www.Octopart.com as a quick way to get fair market cost of your free samples. Remember to see the latest copy of starter file **Lab8BOM.xls**.

Deliverables (exact components of the lab report)
A) Objectives
    1-page requirements document
B) Hardware Design
    Regular circuit diagram (SCH file)
    PCB layout and three printouts (top, bottom and combined)
C) Software Design
    Include the requirements document (Preparation a)
D) Measurement Data
    Give the estimated current (Procedure d)
    Give the estimated cost (Procedure e)
E) Analysis and Discussion (none)

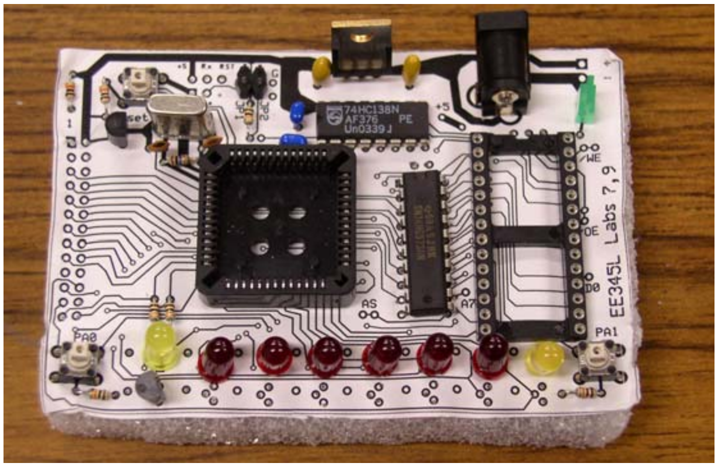


Figure 8.1. Example mockup of layout using Styrofoam (a game of 1-D pong will not be complex enough).

Checkout (show this to the TA)
Show the 3-D mockup of the system to your TA. Explain how the system will be powered. Discuss your estimation the current required by the system? Discuss the testing features of your design. If you plan to incorporate an enclosure, show how the connectors, I/O devices, and PCB fit into the box.

Triple-check everything. It is much easier to fix a mistake before PCB manufacturing than after.

Hints:
1) There are two types of LEDs you can have. Low current red/yellow/green HLMP-D150 LEDs can be connected directly to a microcontroller output using just a 1 kΩ to 2.7 kΩ resistor. The other colors and sizes that I have

require 10 mA and will need an interface (like a 2N2222 and a 220 Ω resistor.) You should test the LED/resistor circuit on a breadboard to make sure the brightness is acceptable.

2) My advice is to do a little bit of Lab 8, then have someone check it. DO NOT DO THE COMPLETE DESIGN SCH/PCB THEN GET IT CHECKED. To have Lab 8 checked, you can contact your TA, or email the SCH and PCB files to your professor. We will evaluate your

SCH files for gross design errors in the I/O interface
PCB files for style (line width, corners)

3) The datasheets for the components used in this lab can be found on the datasheets page
http://users.ece.utexas.edu/~valvano/Datasheets/

4) A very popular item students have bought is LCD1030 from www.bgmicro.com. It is a 128x64 Graphic LCD. There is starter code for this LCD. See AGM1264_DP512.zip at http://users.ece.utexas.edu/~valvano/Starterfiles/
Get this connector Samtec BCS-120-L-S-TE

5) If you plan to put the system in a box, you should create a 3-D mock-up of the system including the box during Lab 8. Starting to think about squeezing all the components into the little box once you get to Lab 11 will be difficult. Placing components in the proper place on the PCB during Lab 8 will greatly simplify the box-building process.

6) PCB Artist has a feature that allows for ground planes (copper pour). I STRONGLY SUGGEST you do NOT use this feature. Ground planes are useful for high frequency and/or low noise systems. The ground plane makes it much harder to visually see what wire connects to what pin, it makes it much harder to cut/add traces in Lab 11 to fix mistakes, and it makes it harder to create good solder joints without using a high-temperature soldering iron.

7) One common mistake new PCB layout designers make is placing two wires too close to each other. Subsequently, during fabrication, these two wires may become shorted because of the tolerances of the manufacturing process. A general rule of thumb is that you should allow enough space between two wires to fit the smallest allowable trace between them. For this PCB manufacturer, separate all traces by at least 0.007 inch.

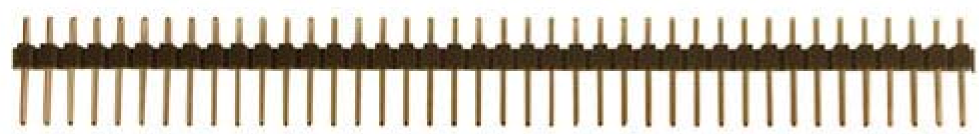
8) Here is a list of parts potentially available in the instructor's office. This information is also included in an excel sheet called **Lab8BOM.xls**. This will help you create a bill of materials. Number in [] is the distributer part number. pdf file refers to data sheet at http://users.ece.utexas.edu/~valvano/Datasheets/ There are some resistors and capacitors available.

http://www.bgmicro.com

Black case, 7" long x 5-13/16" wide x 2" tall, [CAS1007], $1.00
16 by 2 LCD with back light, [LCD1031], $5.95

http://www.allelectronics.com/

5 pin, 3" jumpers with matching pcb mount, [CON-55], $1.35
20K LINEAR SLIDE POT, OPEN FRAME, [SP-20K], $0.33
32-ohm speaker, [SK-230], $0.50
Electret microphone, [MIKE-74], $0.50
9V battery snap, T-type, [BST-61], $0.50
2 to 40 pin Headers, [SHS-40], $0.02 per pin



2-pin jumper, [SJ-1], $0.10

https://www.jameco.com

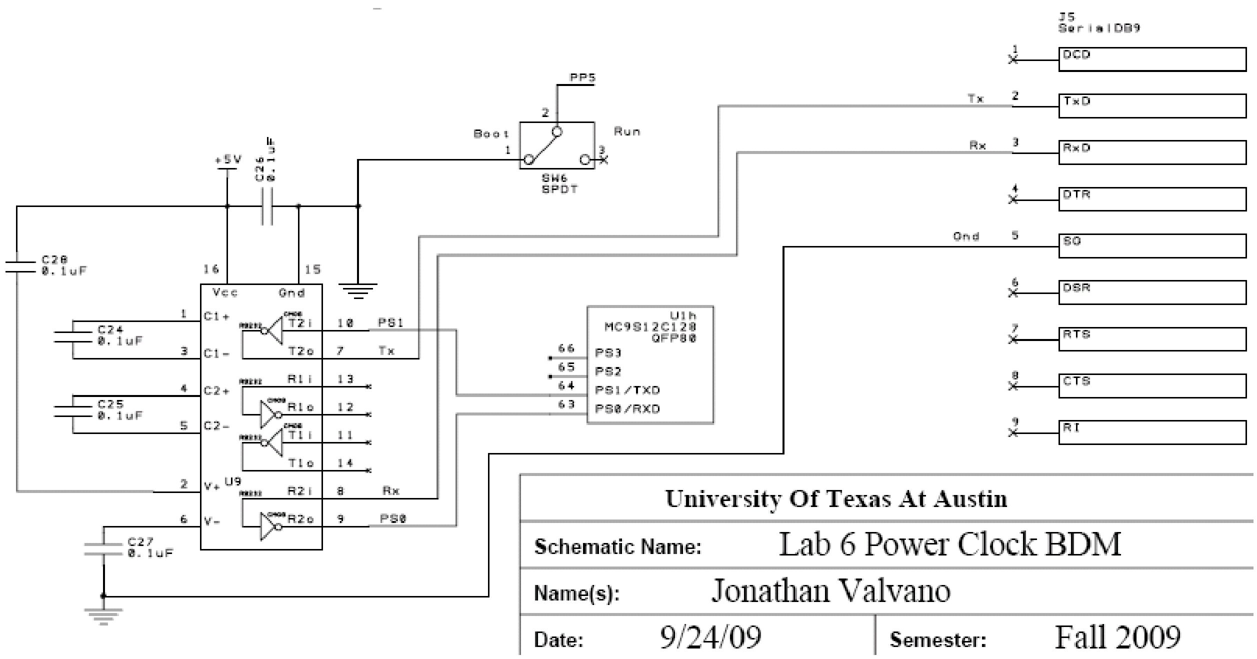
78M05 0.5 amp +5V regulator [192233] (78m05.pdf) $0.19
JACK DC Power MALE 2.1mm [101179] (2.1mmJack.pdf) $0.55
MC34119 audio amp, [316865], MC34119.pdf, $2.25
4.7uF Tantalum 16V 10% cap, [94035] , $0.22
1 uF ceramic mono cap 20% [81509], $0.35

http://www.questcomp.com

ADXL202JQC, 2-axis accelerometer, $6.50

www.Digikey.com

Test point, white, [5002K-ND], $0.29
Red LED 1.6V 1mA 5MM HLMP-D150, [516-1323-ND], $0.29
Yellow LED 2mA 5MM DIFF LOW CURR, [516-1326-ND], $0.29
Green LED 2mA 5MM DIFF LOW CURR, [516-1327-ND], $0.29



Add a Max232 RS232 driver and a run mode switch if you plan a MC9S12C128