Lab 7 Design and Layout of an Embedded System

This laboratory assignment accompanies the book, Embedded Systems: Real-Time Interfacing to ARM Cortex M Microcontrollers, ISBN-13: 978-1463590154, by Jonathan W. Valvano, copyright © 2013.

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, **Lab6_artistStarter**, **Lab8BOM.xls (see the latest version)**

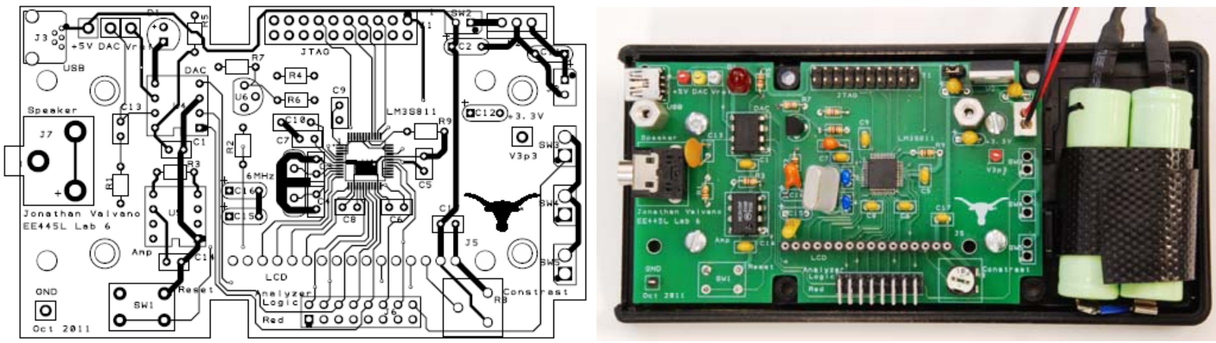


Figure 7.1. Example mock-up and eventual Lab 11 system.

Background

You will use the CAD program **PCB Artist** to layout an embedded system. The software to run the device will be performed in Lab 8. The hardware construction, final testing, and performance measurements 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. If you look at Sparkfun.com you will see lots of ideas of I/O devices you could attach to the system. 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 an external secure digital card 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 and soldered onto the PCB you produce,
• 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 (plus $20 shipping),
- See **Lab8BOM.xls** for 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 all students to design a complete system putting the LM3S811 itself onto the PCB, like Lab 6. If this is the first time soldering surface mount components, we expect you to work closely with the professor, your TA, Daryl or Paul during soldering of the LM3S811. In order to salvage your project and grade during Lab 11, we suggest you add large 56 mil vias (hole=29mil) to all I/O pins used in the project. This way if the LM3S811 microcontroller never boots up, you can unsolder the LM3S811, attach individual wires to the I/O pins and complete the project with your LM3S1968 board.

Implementation 2. 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).
MSP430F2012IN, MSP430F2012TN, MSP430F2013IN, MSP430F2013TN
If you wish to use surface mount versions of the MSP430, please get approval from your TA. If you choose this project, please also sample a connector allowing you to interface your PCB to the debugger: SLM-104-01-G-S

Implementation 3. Design and commercialize a Booster Pack board for the EK-LM4F120XL launch pad. For this approach you will need to conceive of something potentially attractive for the launch pad community. There are lots of MSP430 booster packs that will give you an idea of possible projects. Choosing this option will require a business plan (estimation of a realistic price that could be charged, manufacturing cost analysis, and a marketing plan). The approach will not involve layout of the microcontroller board; it will use an EK-LM4F120XL.

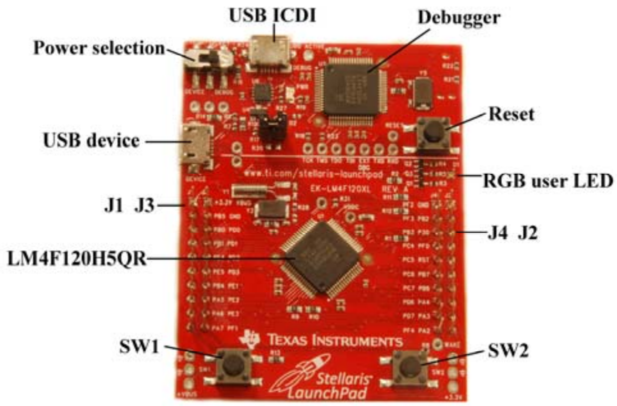


Figure 7.2. EK-LM4F120XL LaunchPad.

We strongly discourage using any of the 100-pin LM3S chips (like the LM3S1968). Basically, we have not found a 2-layer PCB layout or a soldering process that reliably creates a main oscillator clock. In contrast, our success rate with the LM3S811 is over 95%, while the success rate for 100-pin LM3S systems has been less than 50%. If you need more pins, think about using two microcontrollers or using serial shift registers.

You may use your regular EE445L development board if the system involves two microcontrollers, and at least half of the hardware/software exists on the newly designed PCB. An example of this configuration is a wireless link.

If you are using a processor other than the LM3S811 or the MSP430F2012TN, you must sample it or buy it. It must arrive before your final PCB files are submitted. For example, if you turn in your final PCB files before the parts arrive, it might arrive too late to complete the lab and you may not be able to finish. We will have enough samples of both the LM3S811 and the MSP430F2012TN to satisfy.

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, Mark Innmon, Carole Bearden, Sharon Bressette, 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 7, 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, microcontroller) are on the PCB
LCD displays, switches, sensors, 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. 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 the professor's office and show your SCH and PCB files, circling or listing the needed components.

There are two system costs you will add up. The first cost is the sum of all the components regardless of where the part was obtained. This would be the cost to manufacture the system, Procedure e). The second cost is the sum of all the parts you purchased to put into the device. Components from professor's office do not count against your maximum of $30 additional components. Free samples, 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). If you get a part from the second floor, you will have to return it. If you wish to use a part you already own, or a part given/lent to you by another, then you must find the part in stock and report the cost as part of your $30 limit.

Pre-preparation (see course syllabus for due dates)

Part a) Write a one-page **Requirements Document** for the system. Refer back to Labs 3, 4, and 5 for general information about requirements documents. Please email this to your TA. 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 another 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 7 and 11 will be written
- **3.2. Outcomes:** Simply copy/paste the Lab 7 and Lab 11 deliverables.

Preparation (show BOM, SCH, and PCB to your TA at the start of your lab period)

Part b) Create a **bill of materials** and physically collect all components required for the system. Copy and edit the **Lab8BOM.xls** file. 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. Other options for power include a USB cable or a wall-wart. Connectors and cables should be included in the BOM. Record your group on a shared Goggle Doc. For example see

Spring 2012 https://docs.google.com/spreadsheet/ccc?key=0ArNoO7MMFPE0dE1CX01XUEJVbVNMWWpVTm9WaFRWUFE#gid=0
Fall 2012 https://docs.google.com/spreadsheet/ccc?key=0AuBzSLNB7L15dE52SVRFdTk2SnQyVGFOMWljM0EtYWc#gid=0

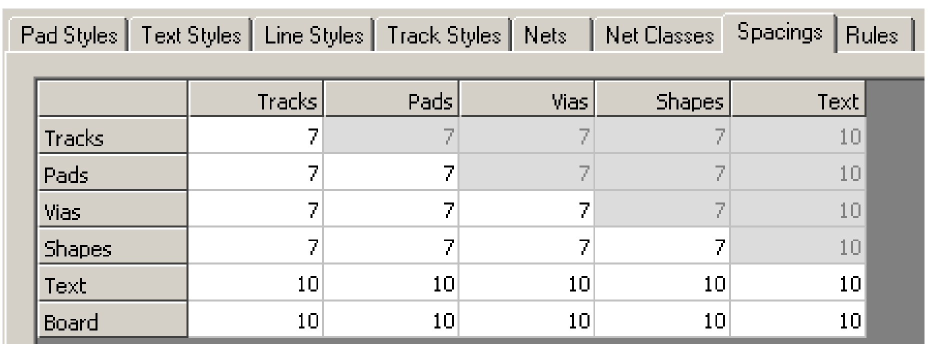
Part c) Draw the circuit diagram (SCH file) using PCB Artist. Refer to the suggestions and guidelines described in Lab 6. We suggest you begin with the Lab6_artistStarter.sch and Lab6_artistStarter.pcb. This way the crystal and JTAG will be connected properly. Be careful not to load the CMOS outputs of the microcontroller. 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 *ForwardDesignChanges*. Place all components inside the PCB area. Follow the first seven steps of
http://users.ece.utexas.edu/~valvano/EE345L/Labs/Fall2011/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 professors have the time or ability to verify accuracy of your PCB layout. However, we should be able to advise you on your I/O circuits.

Common mistakes which will delay the boards for the entire class
1) Having traces/holes/spacings too small (verify settings as shown in Figure 7.3)
2) Having drill holes closer to each other than 10 mil (verify settings as shown in Figure 7.4)
3) Asking for bottom silk, but not giving any bottom silk (verify settings as shown in Figure 7.5)
4) Not asking for bottom silk, but giving a bottom silk layer (verify settings as shown in Figure 7.5)
5) Board name does not match Blackboard submission (see Figure 7.6)
6) Entering the wrong size in Blackboard (see Figure 7.7)



	Tracks	Pads	Vias	Shapes	Text
Tracks	7	7	7	7	10
Pads	7	7	7	7	10
Vias	7	7	7	7	10
Shapes	7	7	7	7	10
Text	10	10	10	10	10
Board	10	10	10	10	10

Figure 7.3. Execute Settings->Spacings… and verify your settings match these numbers.

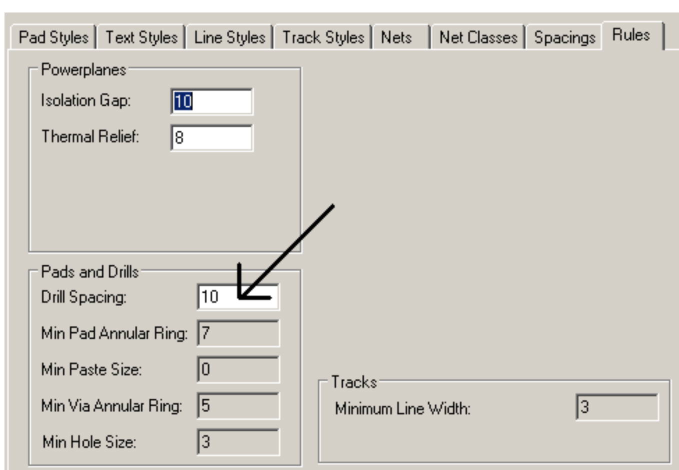


Figure 7.4. Execute Settings->Spacings… click the rules tab and verify "Drill Spacing" is 10 mil.

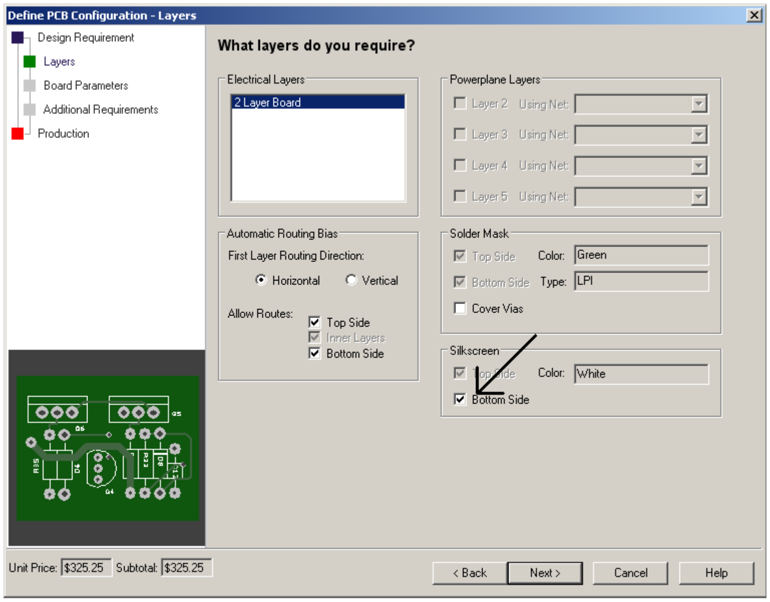


Figure 7.5. Execute Settings->PCB Configuration…(second screen). If you have bottom silk then check the "Bottom Side" button. If you do not have bottom silk then uncheck the "Bottom Side" button.

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 60 in^2. You may not create two designs 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 and bottom silk labels that will assist you in construction and debugging,
Avoid 90-degree turns, convert them to two 45 degree turns (enable the Miter option),
Avoid loops (circles in the PCB traces), because they are a source of noise.
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

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 7.1. 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. You may use measurements from Labs 3, 4 or 5 as estimates.

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 ($33 for board+$20 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**.

Part f) **Uploading the solution.** There is a hard deadline for PCB submission (**see course syllabus for due dates**). You can submit your files before, but no later than then. **If you do not finish by this time, it will cost you about $54, because you will have to order it on your own, and pay $33+shipping.** Here is the procedure to follow before deadline when submitting your PCB files.

0) Get an **xxx** part number from your TA (UTX-2014S**xxx)**. This part number will exist in 4 places.

a) You will place it as silk screen on the PCB

b) You will specify the **Board Part Number** inside the PCB file (See Figure 7.6)

c) Your SCH and PCB files will bare this part number. E.g.,

UTX-2014S100.sch

UTX-2014S100.pcb

d) You will include your **Board Part Number** as part of the Blackboard submission

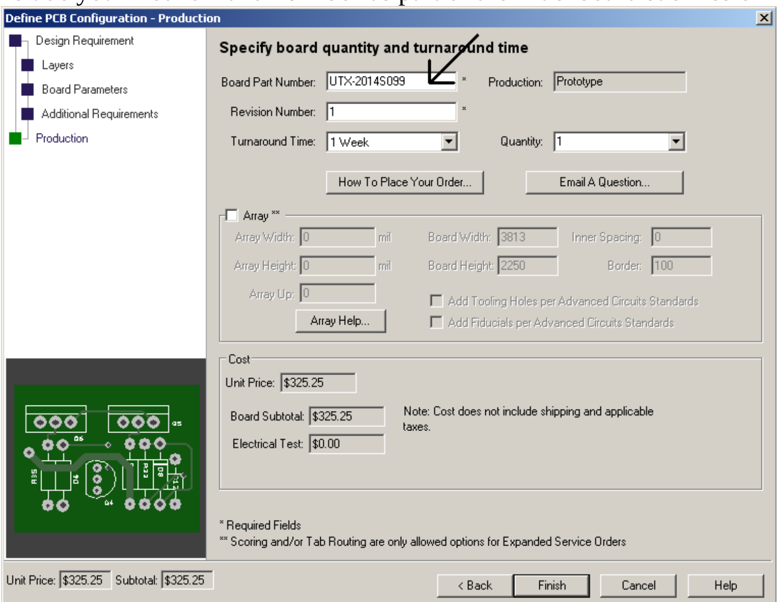


Figure 7.6. Execute Settings->PCBConfiguration… and in the last screen, specify your part number.

1) Follow the instructions in: http://users.ece.utexas.edu/~valvano/EE345L/Labs/Fall2011/PCBOrderProcess.pdf
You can get to that menu from the PCB view, and go to the Settings Menu, and select PCB Configuration. Follow the instructions on the PDF. If you select bottom silk, please add something in the bottom silk layer. If you say there is a bottom silk on the configuration, but the design does not actually have bottom silk, then your board is delayed by 1 day while we get an email warning to confirm or ignore this error. Make sure you select 1 week and quantity 1. Once you've clicked on Order. PCB artist will generate a .fab file. If Advanced Circuits allows you to purchase it for $33 plus shipping, then we can purchase it for you.

2) When you submit on blackboard, please enter the following information as part of the submission on blackboard, comma-delimited:

- Student Name 1
- Student Name 2
- Your TA's Initials
- Project name
- Your **Board Part Number**, e.g., UTX-2013F101
- PCB Filename, e.g., UTX-2014S101.pcb
- X dimension in inches of your board (largest width)
- Y dimension in inches of your board (largest height)
- Area in Square inches of your board (X*Y, even if the corners are nibbled out)

Example: Josh Apple,Drew Sens,CM,Alchemy,UTX-2014S101,UTX-2014S101.pcb,5.35,2.95,5.7825

You will also attach your sch and pcb files. Please **don't** zip them up.

To view your X and Y dimensions, click on your board outline, and at the bottom right of the screen you will see "Size: 123 x 456". 123 is the X size, 456 is the Y size, Area is the product of X size times Y size. **The total area must be less than 60 sq. inches. Make sure you have Your names, Project name, **Board Part Number** on your board in silk screen. The TAs will have to compile all this information and all of your files so we can send it off for manufacturing. If you have questions or aren't sure how to make sure you deliver the files in the correct way, ask before the deadline. Any fixes I have to do to your files/naming/anything after the deadline will result in 10 points being deducted. This includes missing board outlines and deleting internal routes (NOT ALLOWED, use pads/vias to make holes.) **If you don't follow these directions, you will lose 10 points on Lab 7.**

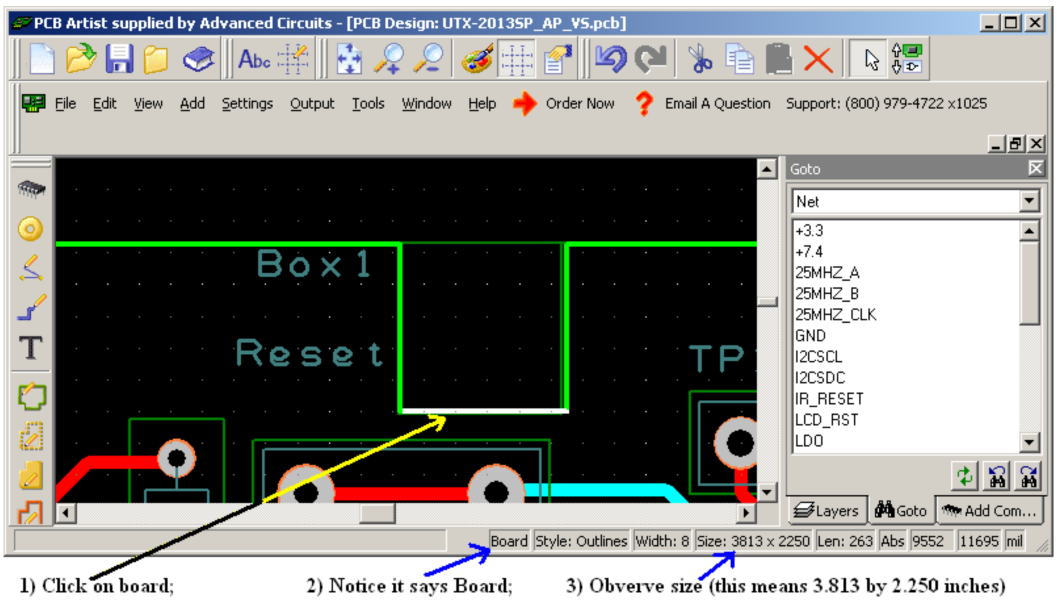


Figure 7.6. Process for determining board size.

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)

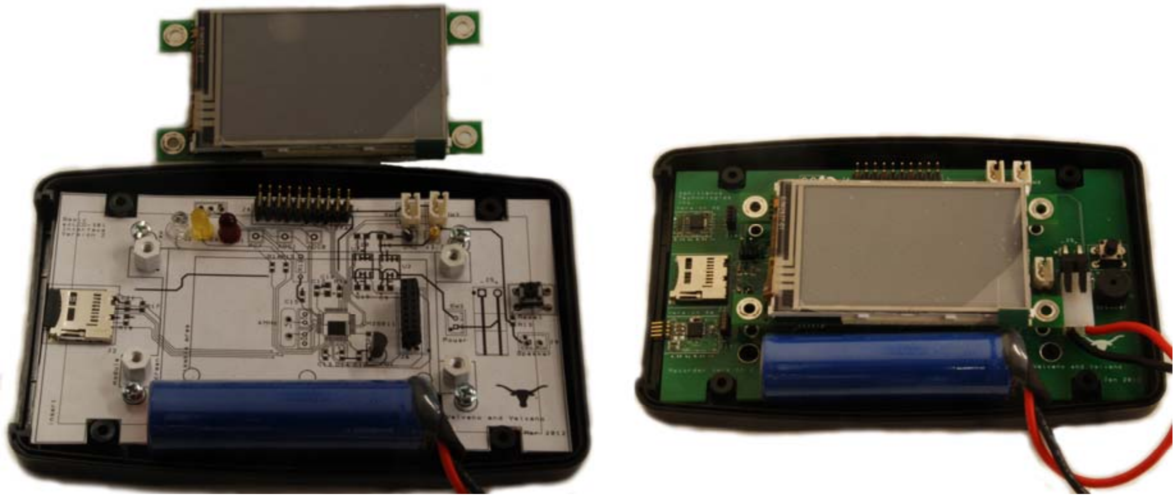


Figure 7.7. Example mockup of layout using paper printout glued on cardboard (LM3S811 into PacTec box).

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:

0) Other than the LM3S811 and the ADXL202JQC you should collect ALL parts of the design while you are doing Lab 7, and you should finish collecting all parts by the time the final PCB is delivered.

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 680 Ω or 1 kΩ resistor. The other colors and sizes that I have require 10 mA and will need an interface (like a 2N2222 and a 100 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 7, then have someone check it. DO NOT DO THE COMPLETE DESIGN SCH/PCB THEN GET IT CHECKED. To have Lab 7 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, mitered 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 is a 128x64 Graphic LCD. This LCD can be purchased at Sparkfun as part number LCD-00710. If you use this display, sample this Samtec connector, BCS-120-L-S-TE. There is starter code for this LCD. See http://users.ece.utexas.edu/~valvano/arm/AGM1264_1968.zip This LCD can be purchased at Sparkfun as part number LCD-00710.

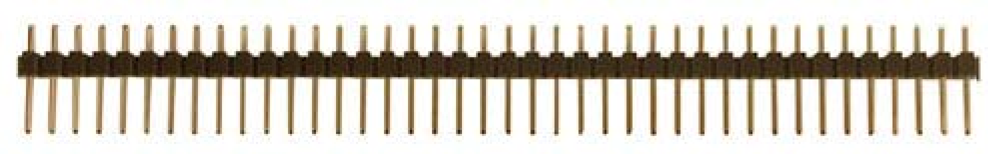
5) If you plan to put the system in a box, you should create a 3-D mock-up of the system including the box during Lab 7. Starting to think about squeezing all the components into the little box once you get to Lab 11 will be difficult. Placing components in the proper place on the PCB during Lab 7 will greatly simplify the box-building process. See Figures 7.1 and 7.5.

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) A list of parts potentially available in the instructor's office is included in an excel sheet called **Lab8BOM.xls**. This will help you create a bill of materials. There are some character LCD displays that are available, in PCB artist it is part LCD1034. It has a 16 pin DIP PCB footprint and requires a 16-pin ribbon cable. This photo shows a straight header with 0.1in separation. This can be broken into any number of pins and used for connectors or mode selection.



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

http://www.questcomp.com
ADXL202JQC, 2-axis accelerometer, $6.50
A computation music game, do a web search for Otomata:
http://earslap.com/projectslab/otomata/?q=5s6x3m2g402z4o6k4q8k480z6g512x3p1z4t7k44
https://www.youtube.com/watch?v=lHCdHh1eSi0

9) Web site references
===== Enclosure Manufacturers =====
http://www.OKW.CO.UK/ OKW
http://www.TEKOENCLOSURES.COM/ TECO
http://www.PACTECENCLOSURES.COM/ PACTEC
Most box manufacturers will not ship samples to students.

==== General Purpose Awesomeness ====
http://www.SPARKFUN.COM Spark Fun - Transducers, Buttons, Displays, etc. Lots of cool stuff.
http://www.allelectronics.com All Electronics - All sorts of random stuff
http://www.bgmicro.com BG Micro More sorts of random stuff
http://www.koaspeer.com/index.asp KOA Speer- will sample surface mount and regular passive components

==== LCDs ====
http://www.newhavendisplay.com/ New Haven Displays - LCD Manufacturer. Has a bunch of displays around $10
http://www.varitronix.com/ Varitronix - LCD Manufacturer.
http://www.crystalfontz.com/ Crystalfontz - good quality and price LCD's
http://www.sparkfun.com/products/710 $20 Sparkfun LCD-00710 64 by 128 LCD
http://www.circuit-ed.com/128x64-BLWH-TOUCHSCREEN-GLCD-P146.aspx $26 64by128 LCD touch screen
http://www.sparkfun.com/commerce/product_info.php?products_id=8977 Touch Screen

==== Batteries ====
http://www.powerstream.com/ PowerStream - Batteries
http://www.batteryjunction.com/ Battery Junctiton - Li-Ion packs

==== Industrial Suppliers ====
http://www.digikey.com/ Digikey - Useful parametric search. Lots of standard components. This is the easiest way to find multiple manufacturers of something common like connectors

http://www.mouser.com/ Mouser - Pretty much any standard chip you want is available here. Ships from Dallas, so usually faster than Digikey, but sometimes a bit pricier.

http://www.avnet.com/ Avnet - Some higher-end stuff and hard to get chips can be found here.

http://www.newark.com/ Newark - Similar to Avnet

==== Misc Info ====
http://www.ladyada.net/library/procure/samples.html Getting Samples - How to get free samples

For surface mount parts available in professor's office, see sheet 2 of the Lab08BOM.xls

Lab 7 grading (different from labintro.pdf)
Lab 7 is the first of three parts to your own project. The grading rubric for this lab will be different from the one mentioned in the labintro.pdf document."

****

Pre-preparation (10) - Requirements Document
You will be judged on the clarity of thought you have about your project. At a very abstract level you should be able to explain your TA what you intend to do and how you intend to do it. You can go through your Requirements document while you explain it to the TA. This will be like a MRD (Marketing Requirements Document) presentation to your TA.

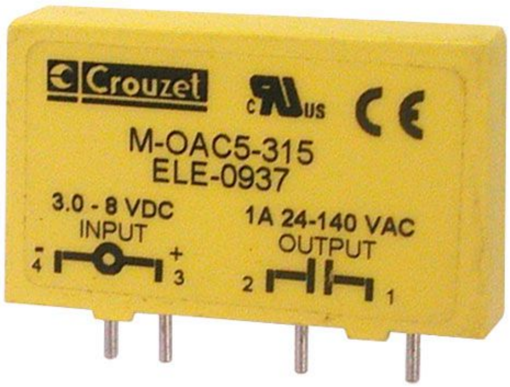
Preparation (10) - Schematic
By the prep-day you should have a very clear idea about your project. So you should be able to describe the lower level interface of your system. You can go through your schematic and BOM while you explain it to the TA. While you do this you would also like to point out your hardware and software design boundaries. This will be like a PRD (Product Requirements Document) presentation to your TA.

Check out (20) - PCB layout
Complexity (10) - You should be able to convince your TA that you have taken up a challenging project. Indicate the cool features of your project and the effort you will have to put in to make it work. Be as specific as possible.
Planning (10) - For every big project it is important to plan things out ahead of time. Break up your work into smaller steps and assign a deadline to each one. Remember to adhere to them.
Prototyping - Although as part of this lab you need to prototype your system before you finalize your PCB layout, your grade on this will be reflected in Lab 8 where you design the software for the system and do a much more precise prototyping.

Oral Questions (10)

Timely submission (10) - The PCB order process has to be done in bulk (for the discount) and on time (so that we get back the PCB on time). So it is very important you adhere to the deadline.

Report (40)
Requirements Document (10) - Due with Pre-Prep
Schematic (10) - A significant amount of this grade will be on how you plan to debug your board
Test points
Proper use of the logic analyzer
PCB Layout (10) - Due with Checkout
Measurement data (10) - Due with Lab submission along with a copy of the above



CAT# SRLY-19 www.Allelectronics.com
http://www.crouzet-ssr.com/english/products/download/M_output_modules.pdf

84 by 48 LCD, Nokia 5110, www.Sparkfun.com part10168, $9.95, Nokia5110RedSparkFun

18-bit color, 128*160, 1.8" TFT LCD display, Sitronix ST7735R, www.Adafruit.com part number 358, $19.96