Lecture 13 objectives

- Lab 6 overview
- Requirements document
- Power budget
- Design for test
- Layout

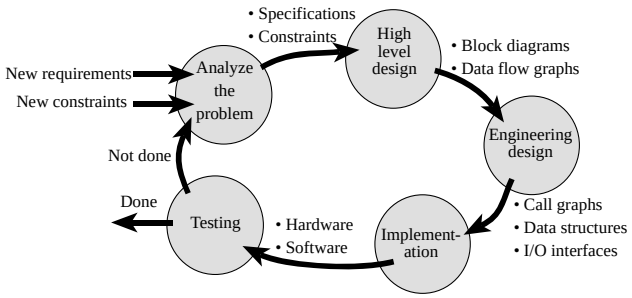


Figure 6.1. Product development cycle.

Requirements Document.

states what the system will do.
serve as an agreement between you and your clients
legally binding contract
easy to read and understand by others
unambiguous, complete, verifiable, and modifiable

It does not state how the system will do it.

Sample Outline

1. Overview
- **1.1. Objectives:** Why are we doing this project? What is the purpose?
- **1.2. Process:** How will the project be developed?
- **1.3. Roles and Responsibilities:** Who will do what? Who are the clients?
- **1.4. Interactions with Existing Systems:** How will it fit in?
- **1.5. Terminology:** Define terms used in the document.
- **1.6. Security:** How will intellectual property be managed?

2. Function Description
- **2.1. Functionality:** What will the system do precisely?
- **2.2. Scope:** List the phases and what will be delivered in each phase.
- **2.3. Prototypes:** How will intermediate progress be demonstrated?
- **2.4. Performance:** Define the measures and describe how they will be determined.
- **2.5. Usability:** Describe the interfaces. Be quantitative if possible.
- **2.6. Safety:** Explain any safety requirements and how they will be measured.

3. Deliverables
- **3.1. Reports:** How will the system be described?
- **3.2. Audits:** How will the clients evaluate progress?
- **3.3. Outcomes:** What are the deliverables? How do we know when the system is done?

The following requirements are added to the design

The system will be controlled by the 80-pin QFP MC9S12C128
All components must be soldered to a single PCB board.
You must hand place and hand route the PCB
(you can not use *Autoplace* or *Autoroute* features)
The system must fit neatly into a box
The system will be battery powered,
run for 24 hours (4 hours if stepper lab), and
the battery must also be in the box.

Power Budget

Average Current must be less than $\mathbf{E/t_{life}}$

Show the Li-ion 7.2V battery

Size, weight, voltage, energy
How long will it run your stepper motor?

Bill of materials

Show **Lab8BOM.xls**

Do the Lab 6 preparation

0) Download Lab6h_artistStarter.sch Lab6h_artistStarter.pcb
1) Rename starter files *and* update Title Block on Schematic
2) Copy of your lab SCH file from lab 3,4,5
3) Delete 9S12 and all nets that connect to 9S12
4) Copy circuit, paste into starter file SCH
5) Connect to 9S12, name all the new nets
6) Execute Output->Unconnected Pins Report
7) Execute Tools->UpdateComponents->AllComponents
8) Execute Tools->SchematicPCB->ForwardDesignChanges

Build a part

0) Schematic Symbol
Power on Top
Ground on Bottom
Inputs on Left
Outputs on Right
1) PCB Footprint
Follow mfg spec for *YOUR* part (look in the datasheet!!)
CHECK AND DOUBLE CHECK pin numbers
2) Component (Footprint + Symbol)

Incorporate part

0) Add it, or change one of the existing parts into it
1) Execute Tools->UpdateComponents->AllComponents
2) Execute Tools->SchematicPCB->ForwardDesignChanges

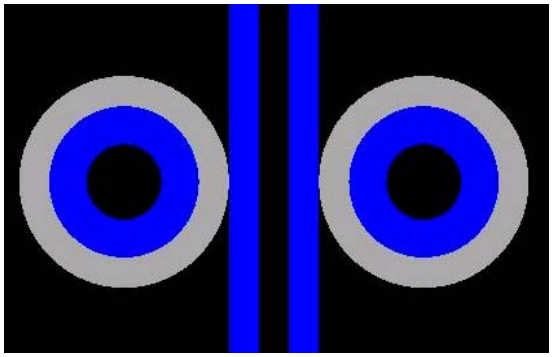
Length	Temperature Rise	Current	Thickness	Resistance	Trace Width
5"	1 C	100 mA	1 oz/ft^2	1 Ω	2 mil
5"	1 C	200 mA	1 oz/ft^2	0.47 Ω	5 mil
5"	1 C	500 mA	1 oz/ft^2	0.13 Ω	20 mil
5"	1 C	1 A	1 oz/ft^2	0.05 Ω	50 mil
5"	1 C	2 A	1 oz/ft^2	0.02 Ω	120 mil

Table 6.1. Minimum trace width for various current levels

Design Rules
Our Manufacturing Specification

Blue is copper, ignore the gray

Minimum trace width = 7 mil
Minimum trace clearance = 7 mil
Minimum drill size = 15 mil
Maximum board size = 30 in^2
Smallest via is 25 mil pad, 15 mil hole
(because minimum annual ring 5 mil)



Layout PCB

0) Change size of board

1) Show miter corners

While drawing a trace, right-click, SegmentMode, Miter

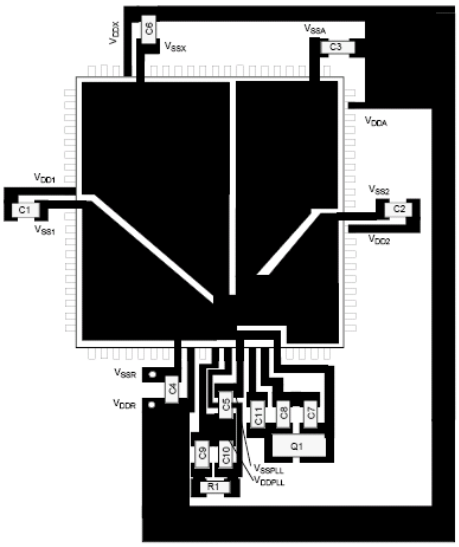
2) Place all fixed location parts first

Carefully consider interface placement within the mechanical boundaries of the box

Think about how connectors will plug in

3) Place parts near where they are connected

4) Place bypass caps as close as possible



http://users.ece.utexas.edu/~valvano/Datasheets/MC9S12C128_V1.pdf

Check the datasheet for specific design recommendations

5) Execute Tools->DesignRuleCheck

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 holes 0.1 in apart then soldering a U wire into it,

or by making a 0.090 in pad with 0.043 in hole then solder a test point into the one hole

Each IC should have a bypass capacitor, placed as close to the chip as possible,

All components need labels (e.g., U1 R1 C1 J1 etc.), shown both on the board and the circuit diagram,

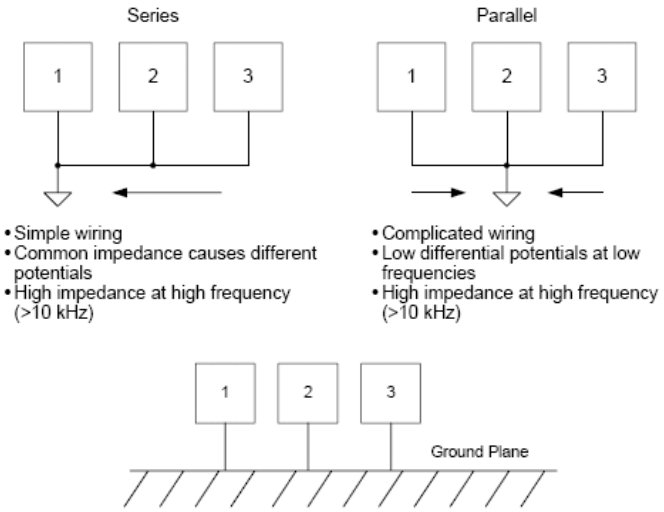
Avoid 90-degree turns, convert them to two 45 degree turns,

One way to make it all fit is to go left-right on one side and up-down on the other side

6) I route ground first, then power

L

S



Show how to set up PCB