Chapter 11 Wk 6 Monday and Wednesday: Design Brd 2

This is the beginning of the design process for brd 2. You got a taste of the board design for brd 2 in the measurement of switching noise on our version of brd 2. Now you get to design your own version.

11.1 Brd 2 POR

In this second board design project, you will get started practicing the best design principles in a simple 2-layer board. The goal is to design and build a board which will:

1. *Convert 5 V in to 3.3 V*
2. *Create a clock signal of about 500 Hz and about 50% duty cycle*
3. *Drive four of the inputs of two hex inverters used to demonstrate good layout and bad layout*
4. *Use red LEDs and 50 ohm resistors as the load to three of the switching outputs of each hex inverter*
5. *Estimate the current you expect to draw from the inverter for the LED and 50 ohm resistor load.*
6. *Stretch goal: if you measure the voltage on the output of an inverter with the LED and resistor compared to the inverter with no load into the scope, you can extract the Thevenin resistance of the inverter output.*
7. *Connect the output of the fourth switching inverter to a test point to act as a trigger for the scope*
8. *Set up one output of each hex inverter as a quiet HIGH and one output as a quiet LOW*
9. *Demonstrate indictor LEDs, test points and circuit isolation switches as appropriate*
10. *Engineer the layout on one side of the board with best design practices and the other side of the board with bad layout practices.*
11. *Keep the part placement and routing identical for the two regions of the board, except for the location of the decoupling capacitor.*
12. *Plan test points for at least:*
    a. *The scope trigger output*
    b. *The quiet high*
    c. *The quiet low*
    d. *The 555 output signal*
    e. *The 3.3. V rail on the board*
    f. *The 5 V rail on the board*
13. *Use best scope measurement practices when you test your board*

You will use this board example to step through the entire board design process.

11.2 The schedule for Design Assignment Brd 2

We will complete the design steps in these phases:

Wk 6 : Monday

- Create POR
- Create schematic, starting from your brd 1 design.
- In-class lab on Monday will be schematic design reviews.

Wk 6: Wed

- In-class lab on Wed will be to finish your layout and complete a CDR of your layout before release to fab
- Create the design files and sign off by your TA brd design files should be submitted by Wed, Sept 29, 9 pm
- Your brd design files will be ordered on Thurs, Sept 30.

Wk 9: Wed

- *When the boards are back, you will characterize the noise on the good layout section and bad layout section.*
- *The report for brd 2 is your midterm, due Monday Oct 25, at the latest.*

11.3 New components for brd 1

11.3.1 The LDO

We usually supply power to a board from some external DC voltage. This can be a battery or an AC to DC converter. On the board, there will often be other Voltage Regulator Modules (VRM) to create other voltages, or provide more stable voltage supplies.

An alternative DC voltage can be provided by a linear regulator. These devices typically need to have supplied a DC voltage higher than 2 V above the required output voltage.

When a low current and very stable voltage is required, and the input voltage is less than 2 V above the required output voltage, it is common to use a low drop out (LDO) voltage converter. The voltage accuracy of its output is typically 1% absolute accuracy.

For example, if the input voltage is 5 V and the output voltage required is 3.3. V, then a linear regulator can't be used and only an LDO would work.

An LDO is generally very easy to us. There is often an enable pin. This should be tied high to the input voltage to enable or turn on the output voltage.

There is a feedback circuit inside the LDO which monitors the output voltage and adjusts the internal resistance of a series MOSFET to keep the output voltage within a specific range, compared to an internal reference voltage.

There is some response time to this feedback circuit. Sometimes, if there is a noise fluctuation that causes the output to vary, the feedback loop can go into oscillation. To prevent this, the output voltage fluctuations must be slowed down. This is done by adding a ***filter*** capacitor to the output of the LDO. It's value is not critical. Any value in the 1 uF to 22 uF range will filter the high frequency noise to prevent oscillation.

This filter capacitor should be placed in close proximity to the output of the LDO, to provide a stable voltage at the output, so that noise fluctuations do not cause the LDO to go into an unstable state. This is NOT a decoupling capacitor. It is a filter capacitor associated with the LDO.

If you are interested in a cool experiment, use a 1206 filter capacitor so you can manually take it off your board and then an isolation switch to isolate your 3.3 V LDO from the rest of the circuit. With the capacitor removed, you may see the oscillations at the output of your LDO. With the capacitor in place, there should be no oscillations. The oscillations will be large voltage fluctuations at about 1 MHz.

The LDO we have in the LCSC library is this one: LCSC Part# C6186. You can do a google search and find the datasheet for this part.

11.3.2 A 555 timer chip

The 555 is a powerful component from which we can build a variety of oscillators and frequency related functions. This is the part commonly used.

There are a number of versions of the 555 timer. We have two versions in our integrated library:

LCSC Part# C90760

LCSC Part# C7593

Which one will you use?

This is where design tradeoffs come in. Your criteria for which one to use might be based on:

1. *Cost: is it an extended or basic part? Check the JCLpcb web site: https://jlcpcb.com/parts and search on the LCSC part number.*
2. *If the unit cost is < $0.5, the part cost will be in the noise.*
3. *Is it in stock? Check the same web page as above.*
4. *Can it operate at the required 3.3 V of the power rail? Or should it be driven by the 5 V rail?*
5. *Can it handle the current it needs to source or sink to drive the hex inverter inputs?*
6. *Can it switch fast enough for the application?*
7. *Are there any other features we need for the part for this application?*
8. *Does one part have more or less risk that the other part?*

You decide which part you want to use, based on your analysis. There are no wrong decisions, if you can justify your decision. Sometimes it is a personal preference. That is an ok answer as well.

For me, I am a speed freak. I like faster parts and I am willing to pay a little extra for faster parts. I also like to push parts to their limits and see if I can break them, so don't always do what I say, unless you are wearing safety glasses.

This analysis and your part selection should be stated in your POR, which will be in your final report.

11.3.3 Hex inverter

The Hex inverter, 7414, is a very simple chip. It has 6 inverters available, each with an input and an output. An inverter circuit component will take an input digital level and output the opposite. If a 3.3 V signal is at its input, its output will be 0 V. A 0 V signal at the input will result in a 3.3. V signal at its output.

NEVER use an inverter with an input pin that floats. The input impedance of an inverter is generally very high. This means stray electric fields as from static charges, AC pickup or cross talk from a nearby trace, may change the voltage at the input and cause the output to switch all by itself.

You can do a cool magic trick with an inverter with inputs that float. Connect the outputs of a few inverters to LEDs and current limiting resistors. Rub your hair or your clothes a little or sit in a cloth chair to build up some static charge. When you waive your hands over the board, in the vicinity of the input pins of the hex inverter, the LEDs will flash on or off. Your static fields are inducing a HIGH or LOW signal at the gate inputs of the MOSFETs of the inverters.

Tie all connections that might be open to a pull up resistor to 3.3.V. This way the outputs will be 0 V. What value of pull up resistor should you use? I recommend a pull up resistor of about 10k. What influences this value?

There are many versions of the hex inverter. We have two in the LCSC library:

LCSC Part# C5482

LCSC Part# C6065

Which part will you use in your board?

You can use the same criteria as for the 555 to make a decision.

For our application to show off the noise, we want the fastest switching part so that we can see the most noise. The switching noise varies with dI/dt. Of course, if you wanted lower switching noise and the rise time of the signal was not important, then you would want the slowest part.

Wouldn't it be great if the datasheet explicitly offered the rise and fall times? If it doesn't list this important parameter, and you wanted the fastest part, how will you decide which part to use?

Just for your reference, the letters, LVC, or HC, or AHC in the part name refer to the IC technology for which the parts were built. This also affects the switching rise or fall time. Generally, an LVC part is faster than an AHC part, which is faster than an HC part.

Do you want to run your hex inverters at 5 V or at 3.3 V? If you are not sure, you can add a 3-way switch to select the 5 V rail or the 3.3 .V rail for each inverter chip. This is one way of selectively turning the chip off or on and isolating it from the rest of the circuit. This approach of providing an option to the voltage rail of a device is sometimes an important feature to test in the evaluation or prototype phase of a circuit design.

Whichever part you decide to use, it is very important that you:

- ✓ *Use the same part for BOTH the good and the bad layout circuits.*
- ✓ *Use the same power connections for both parts*
- ✓ *Make note of the part you are using so you can include it in your POR and your final report.*
- ✓ *Maybe find another classmate using a different part from you so you can compare notes after you get your boards back.*

11.3.4 LED circuits

We will use an LED primarily as an indicator of a line being high. This can be for a data line or a power line. There are five different color LEDs in the LCSC integrated library. They are all in an 0603 part size. Each color will have a slightly different forward voltage drop.

For the LEDs that the HEX inverters drive, we are using them to drive a lot of current and as indicators so we can see the inverters are being driven. For these applications, use a red LED so that we have the lowest forward voltage drop and can get about 30 mA of current switching. Very important to use the same LEDs for the good and bad layout circuit. After all, we want the only differences between these two circuits to be the layout.

For the other indicators, just be sure we can drive them with the 3.3 V signal and a current limiting resistor. What is the criteria for selecting the current limiting resistor for general indicator lights?

As an indicator, how much current do you want to use in the LED? From your practice board, you will see that even 3 mA, as when you used a 1k resistor, was plenty bright.

You can decide in your project which colors will correspond to what message or indication.

Of course, it does not matter which order the LED or resistor are in the circuit. However, if we want to have the option of measuring the current through the LED, one approach could be to measure the voltage across the resistor.

If the resistor is on the high side of the LED and the LED’s cathode is connected to ground, then we would need a differential probe to measure the voltage across the resistor to get the current.

While this is possible, it is sometimes easier to just use a 10x probe which is ALWAYS reference to ground. In this case, the low side of the resistor should be connected to ground. To make this possible, the preferred order is the LED then the resistor.

11.3.5 Test points

All the test points will be used for the 10x probe. This part is found in the manual integrated library.

The signal pin of the tip of the 10x probe is inserted into pin 1 of the test point. The small ground spring of the 10x probe will fit in the far right hole. The other two ground holes are there because I have some other high bandwidth probes that use the adjacent ground hole. This makes this test point a “universal” test point.

11.3.6 Isolation Switches

To isolate one region of the circuit from another, use a 2-pin header pin as a switch. These can be found in the JLC library. We will let JLC do the assembly for us. You will just need to add the shorting flag as needed.

You can sue isolation switches anywhere in your circuit to isolate some parts of your circuit from others, except between the IC power pin and its decoupling capacitor. Why is this not a good idea? If you want to isolate the power from a specific IC, where do you place the isolation switch?

Feel free to add any isolation switches anywhere else in your circuit you may choose to help you debug your board when it comes back.

11.4 Design Assignment, brd 2, wk 5 Monday, before you come to the lab: create POR

Write up a brief summary outline of you POR. The POR is a living document. It may change as you learn more about the design. It will also be part of your final report.

Spending a few hours upfront, before you begin your project to think through the details will be a good investment to help you identify potential risks and how to mitigate or avoid them as early in the design cycle as possible.

Be sure to include:

1. *Rough schematic or block diagram*
2. *List of the significant parts*
3. *Find datasheets for them*
4. *Definition of what it means to “work”*
5. *The schedule*
6. *The test, characterization plan*
7. *The power budget and how you will supply power*
8. *Potential risk sites and how you might avoid them*

11.5 Design Assignment brd 2, wk 6 Monday: complete schematic

You can start with your brd 1 schematic. In addition, use the latest released libraries, the 2021-08 versions for all additional parts.

Try to use only basic parts if possible. This will keep the cost down. If you need extended parts, just justify your reasoning.

Use the 10x probe test points.

Plan to present your design in the in-class design review at the end of the lab period and to review others' designs.

11.5.1 Design reviews

In class on Monday and Wednesday, we will have peer design reviews both as a class and individually in pairs. In a design review be sure to always be respectful.

You will learn as much when you review someone else's design as when your design is reviewed by others.

When you are reviewing a design, be sure to look for good features that you might want to include in your design, and features that are done well.

Look for hard errors that will cause the board to not work.

Look for soft errors that might not reduce the noise to as low a level as possible.

Look for cosmetic errors that might not be good habits and might result in more difficulty (read, higher risk) when assembling, testing or using the board.

Listen to the rationalization the designer used to make their personal preference decisions. Maybe you would want to follow those guidelines as well.

The design review is a discussion. While another pair of eyes looking at a design is always a good thing, sometimes both the designer and the reviewer will learn new insights in the discussion, when forced to articulate their thinking behind a decision.

11.6 Design Assignment brd 2 wk 6 Wed: complete the layout

Use any board size you want, but keep its dimensions below 3.9 in x 3.9 in. Do not exceed this dimension. What happens if the board is larger?

Use a 2-layer board.

In the region of the board with the good layout, keep the decoupling capacitors close to the Vcc pin of the hex and use a continuous return plane under the routing for the traces. This reduces the switching noise.

In the region of the board with the bad layout, do not include a ground plane. Move the decoupling capacitors far away from the Vcc pin of the hex.

Otherwise, the circuits should be identical.

Set up the design constraints as 6 mil signal traces, 20 mil power traces, vias with 13 mil drill hole, 25 mil capture pad.

Complete the part placement and layout.

Add all the important silk screen indicators, especially your name and the brd name.

Before you leave the lab on Wed, be sure to have your design checked off by your TA. If it is not complete, at least have what you have completed reviewed.

During the wk 6 Wed in-class lab, we will do a peer design review of each individual's layout.

Before 9 pm on Wed, Sept 29, you should complete the design, create the three design files and submit them to your TA to place the orders.

To ensure the highest chance of success and getting your fabricated board back on time, complete a design review of your board design files by going thru the submission process on the JLC web site, but DO NOT PLACE THE ORDER.

11.6.1 Grading rubric for the wk 6 lab assignment:

2 points if you submit your design files and the order is accepted

1 point if you submit your design files but there is an error and your board order does not get placed on time.

0 points if your board order is not submitted