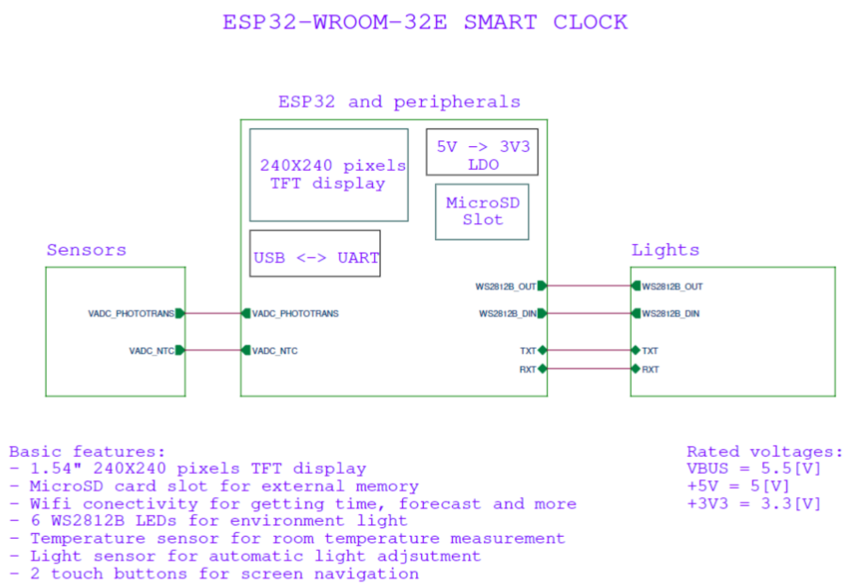
My first remark is that you always want to give some information about your design intended use and operation, I looked through your previous posts and couldn't find some explanation as to what exactly you are designing. This makes the design review a lot harder, like trying to reverse engineering it.

I usually add a general block diagram and some important notes on the first page of the schematic.

Below is an example from one of my designs -

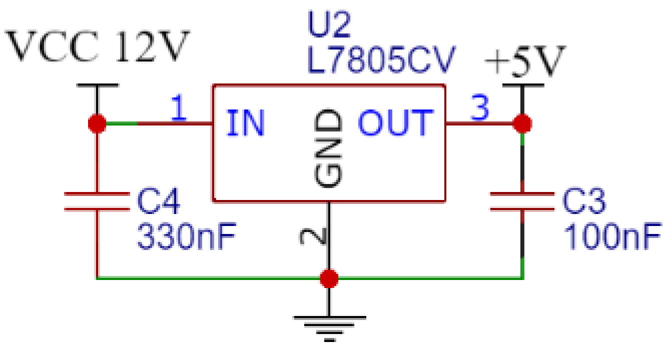


Another thing that pops right out for me is the lack of labels on the nets. There are people who say all of your nets should have an individual name with some basic information, I usually add net names to most of my nets (at lease for all the important nets).

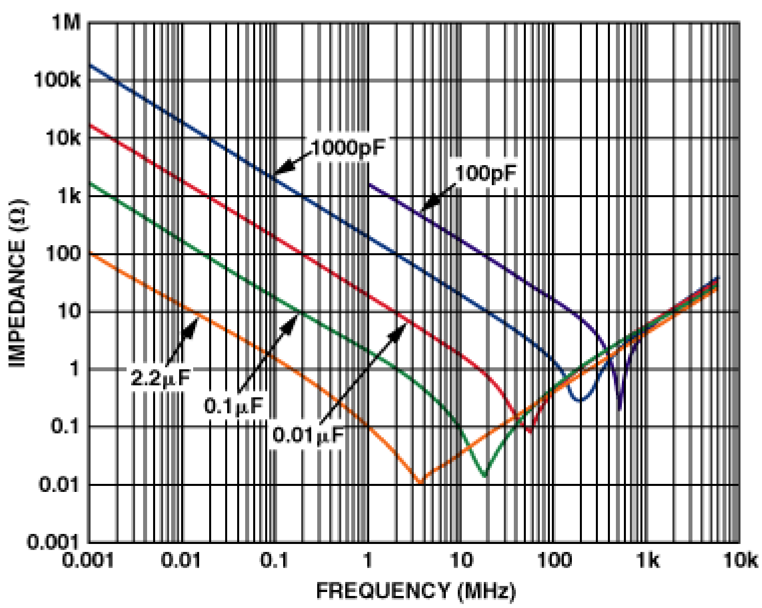
this makes it much easier to review the design and make sure you didn't miss anything, and it's very helpful for the layout.

Take a look at [this schematic](#) from Fedevel for some reference, as the design gets more complicated it gets harder to maintain organization so it's better to start practicing this from the beginning.

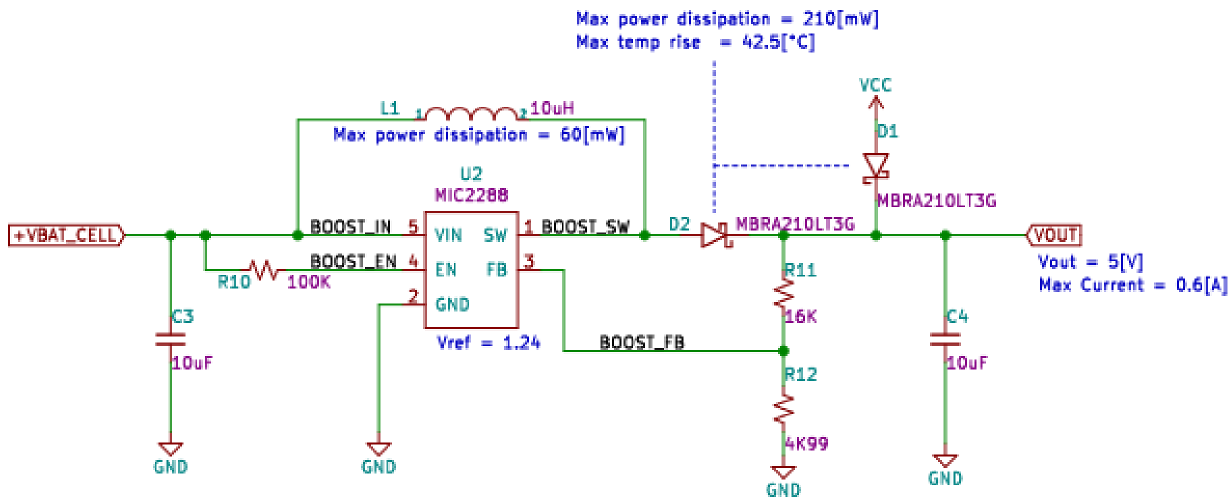
You can learn a lot from reviewing other's schematics (that's one of the reasons I do this right now).



Regarding the voltage regulator, I saw you chose the capacitors according to the datasheet, I would probably go higher with the input bypass capacitor to 1uF, take a look at the picture below just to get the idea of the impedance of the cap as a function of the frequency ([the full article]())



Also, I’m not sure how much current you are going to draw from the regulator but remember to keep power dissipation in mind, it helps if you write your max currents and power dissipation and calculations near relevant components (at least for the main current consuming ICs), for example –



Here I mostly wrote the power dissipation to make sure my components are up to the test, I keep all of my detailed calculations in an Excel file in the project directory, so I can always go back and make sure my calculations were correct (or not :P)

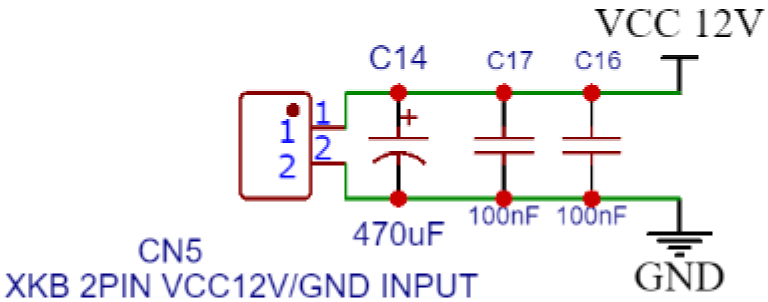
Again, I’m not sure what your current consumption is going to be but even for 0.1[A] the IC is going to get hot which should be a thing to think of (for example making sure you optimize heat dissipation in your layout).

$$V_{IC} = V_{in} - V_{out} = 12 - 5 = 7[V]$$
$$I_{IC} = 0.1[A]$$
$$P_{IC} = V_{IC} \cdot V_{IC} = 7 * 0.1 = 0.7[W]$$

For the TO-220 the thermal resistance from junction to ambient is 50 [°C/W]

$$Temperature\ rise = 50 \cdot 0.7 = 35[°C]$$

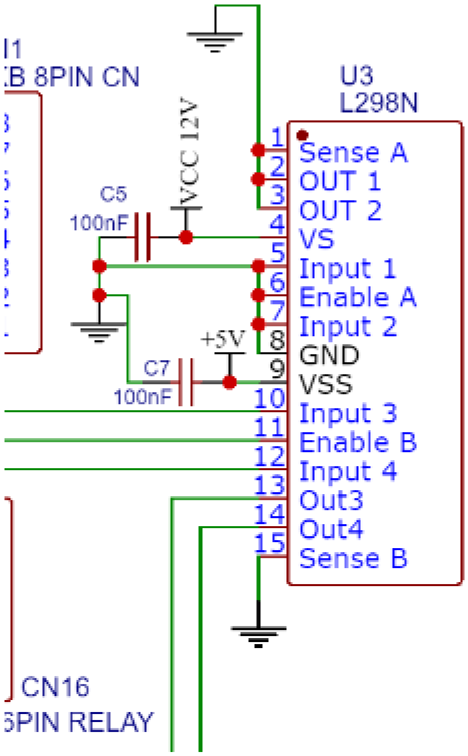
This will be OK, just keep this in mind for later designs.



I saw this connector after I looked at the L7805 so you can ignore my remark about the input capacitance.

As for the input capacitors for the connector, 470uF seems pretty high, not sure exactly why you need such capacitance but if you did your calculations and this is what you got then OK.

As for the 2 100[nF] caps, consider changing one of them to 1[uF] to get better frequency response, and consider using different footprints to allow for more options like 0805 and 0603 or 0603 and 0402 (if that's the case then ignore my remark)



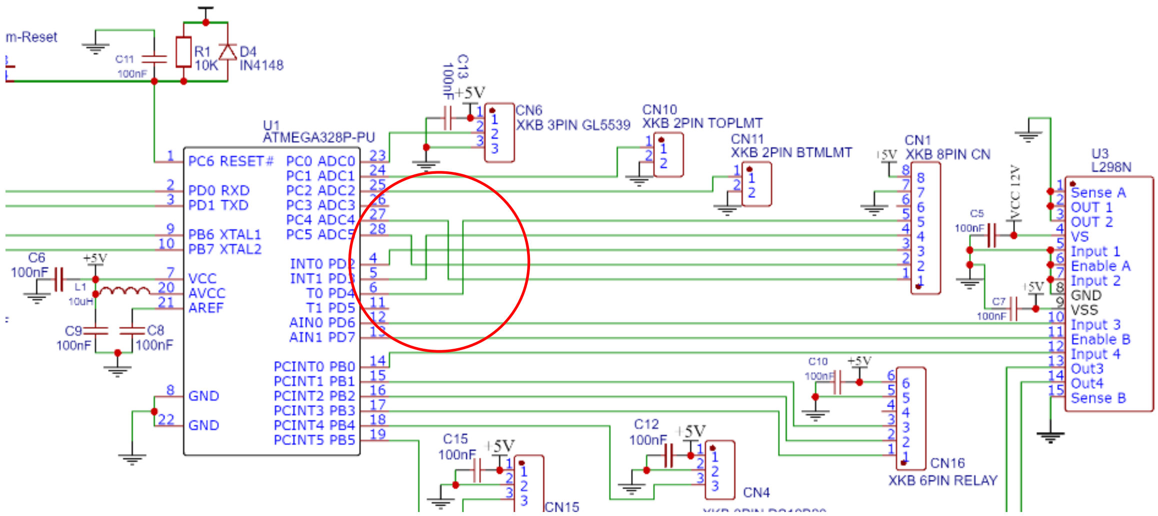
Regarding the motor driver, first of all this component seems pretty old, and the datasheet is a mess.

I always go with the more detailed components, you want all the information for the calculations you'll be needing to do.

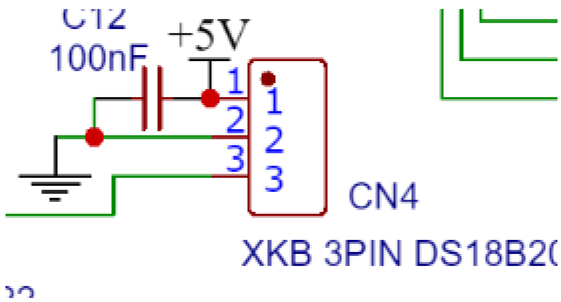
Also, for later projects, do a small reaserch of similar components and choose ICs with detailed explanations and examples.

One thing I can't see in your design is the sense resistors, I haven't used these ICs but it seems like an important component, I skimmed through the D.S and didn't see a detailed explanation about choosing their value but it should be there somewhere.

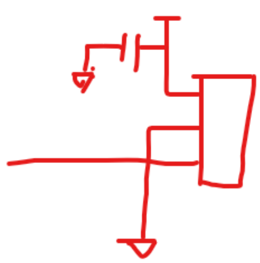
I would also spread the wires a little more to make this more 'readable', remember that you can change the locations of the pins on the symbols to make the wiring less tangled.



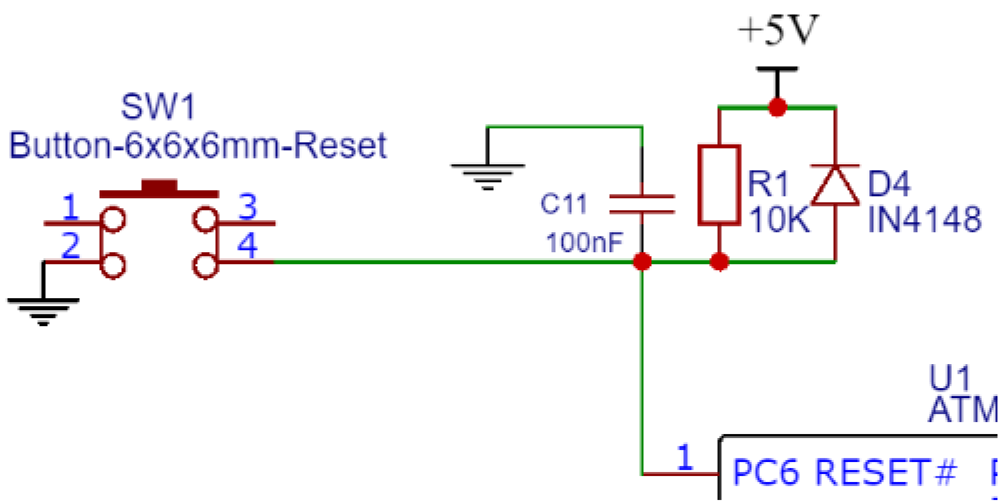
In that same manner I would remove some of these direct connections and use net names to connect the relevant nodes. I would also separate the connectors, that way you can create a different area in the schematic with all the connectors.



For stuff like this, I think this is easier to read -



It’s just personal taste I guess, it’s not really an issue.



As for the reset circuit, again I would move the circuit aside and connect it to the reset pin using net name.

I’m not exactly sure what the diode is doing there, if that’s something needed then keep it otherwise remove it.

I found in this datasheet the following text

PC6/$\overline{\text{RESET}}$

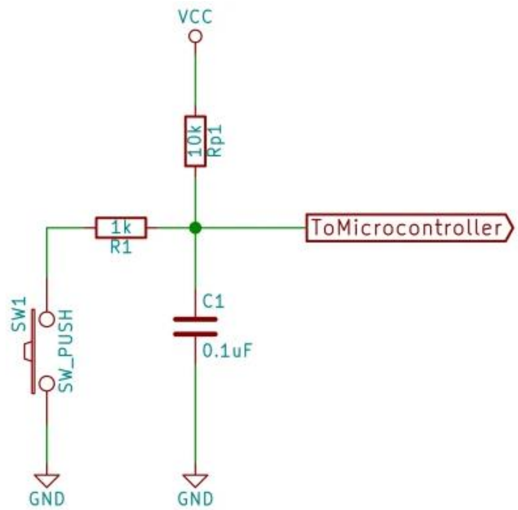
If the RSTDISBL fuse is programmed, PC6 is used as an input pin. If the RSTDISBL fuse is unprogrammed, PC6 is used as a reset input. A low level on this pin for longer than the minimum pulse length will generate a reset, even if the clock is not running. The minimum pulse length is given in Table 28-4 on page 261. Shorter pulses are not guaranteed to generate a reset.

The various special features of port C are elaborated in Section 13.3.2 “Alternate Functions of Port C” on page 68.

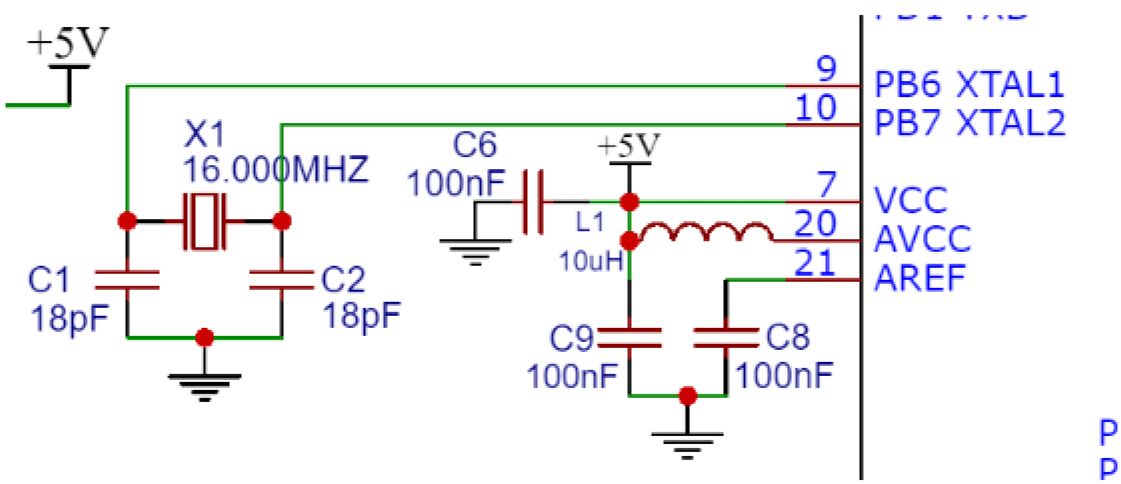
The values you chose for the capacitor and resistor would probably work, otherwise check the datasheet.

You can also add some resistor in series with the button to reduce the peak current and noise over the pin (I know you can fix this in SW later, but I try to prevent such problems beforehand in HW).

Something Like this –

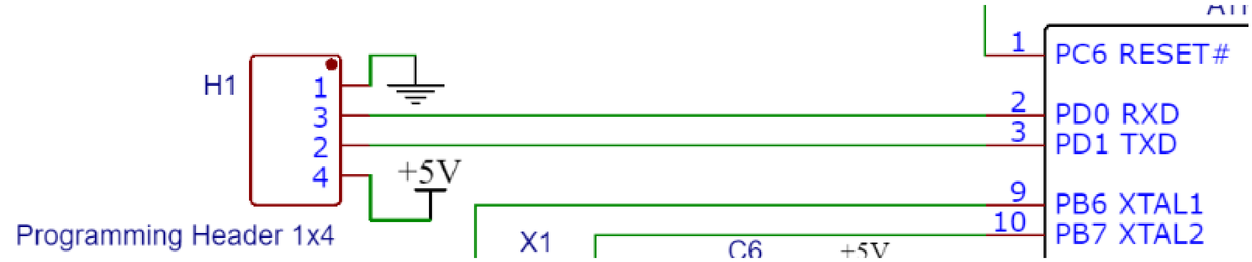


Check out [the full article]() for more information



I didn't check if these values were chosen according to the datasheet or some reference design but do this according to the D.S.

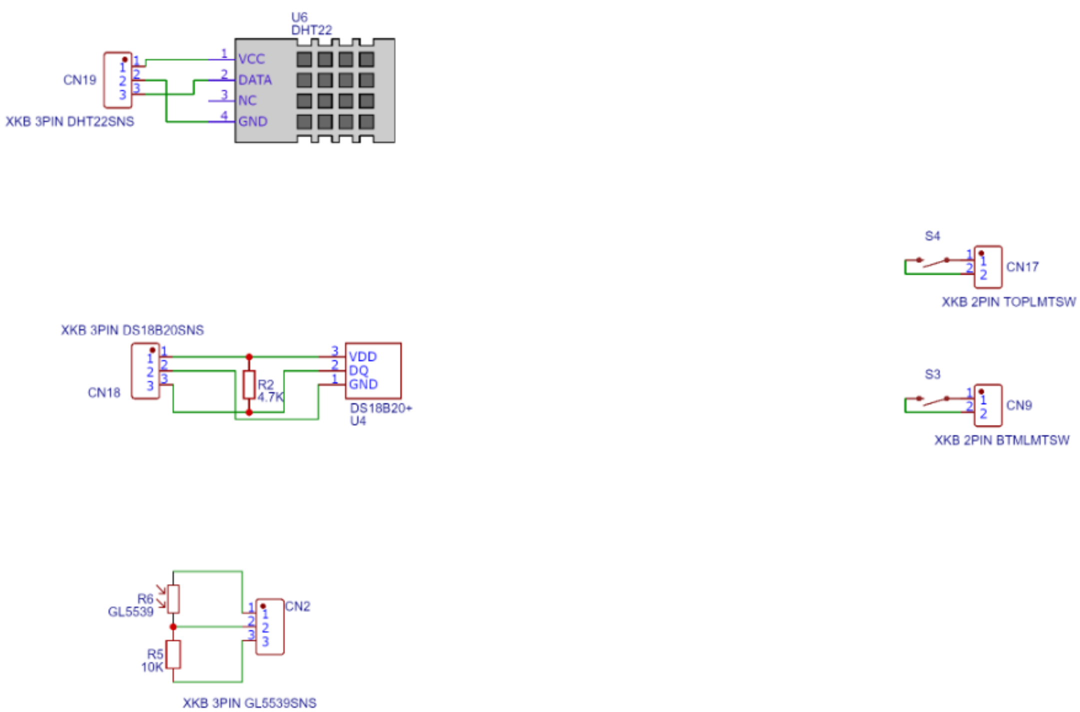
Again, this part of the schematic is a bit cluttered, I would move the components apart and use net names if needed.



Use net names to indicate the direction of the communication, remember that the RX goes to the TX on the other end and vice versa, also keep that in mind for layout and remember to add silkscreen.

I like using net names like 'ATMEGA328_TX' and 'CONN_RX' then connect them with a wire,

Below is an example from my design, you don't need the resistors, I added them for another reason.



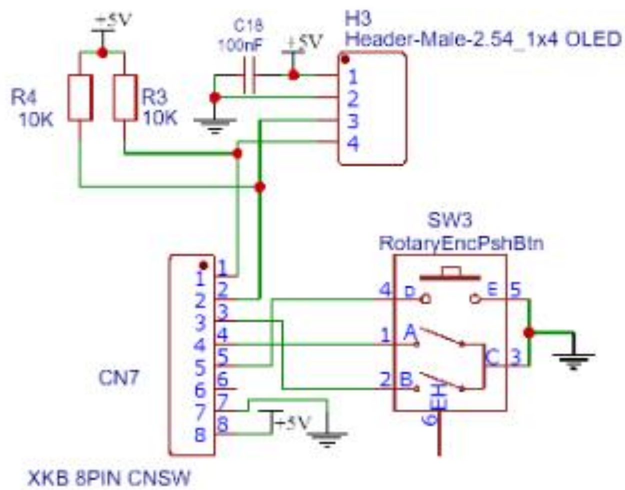
As for the sensors and connectors page, I'm not sure I understand but if all of these are connected separately and not mounted on the board, this should be indicated somewhere.

Again, it's pretty hard to follow, I'm not sure which connector guess where but double check this.

Consider adding 100nF capacitor to the DHT22 (taken from the D.S)

(1) Power and Pins

Power's voltage should be 3.3-6V DC. When power is supplied to sensor, don't send any instruction to the sensor within one second to pass unstable status. One capacitor valued 100nF can be added between VDD and GND for wave filtering.

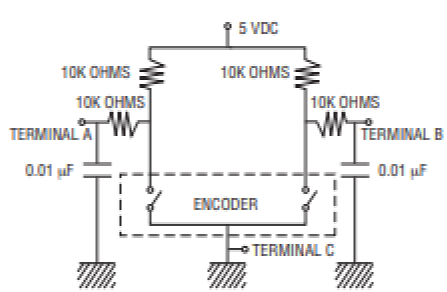


As for the display, again, net names and pin names on the symbols would make it easier to review.

As for the rotary encoder, I'm not sure which one you are using but consider adding a filter circuit to the signals.

For example, the [PEC12R](PEC12R) suggest this circuit –

Suggested Filter Circuit



Check your encoder datasheet and consider adding it just in case it is needed, other wise you can always lose the capacitors and pull up resistors and change the series resistors to 0 ohm.

Over all it seems fine, try to keep the design as neat and organized as possible, it makes it much more readable and helps avoiding stupid mistakes, also use net names as much as possible.

Good luck! Keep working on it and update us! I hope this helps in any way 😊