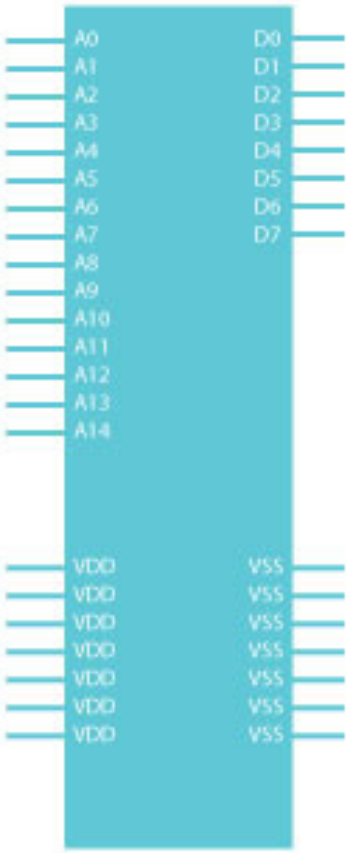
Good Schematics Lead to GOOD LAYOUTS

An electronic schematic describes the electrical connectivity of a piece of equipment or an entire system. It is made up of symbols that represent individual components and contains electrical and mechanical information and their related connectivity, along with other important data. Information contained within the schematic is packaged into a printed circuit board (PCB) where the mechanical footprint is placed onto the board and connectivity information is graphically displayed. The more accurate the information contained in the schematic is and the clearer it is presented, the more it contributes to a robust printed circuit board.

Library Symbols

If we think of the schematic as the building block of the printed circuit board, then an accurate and concise library of parts are the foundation for the schematic and layout. Logical library symbols must accurately represent the electrical data presented in the data sheet as well as link to the mechanical footprint that ultimately is placed on the PCB. Additionally, clear succinct pin names, as communicated from the data sheet, need to be on the symbol. Avoid crowding text but remain mindful that you need to get many parts on a schematic page. When abbreviations need to be used, take care to keep pin names clear.



A good rule of thumb for library parts is to group like pins together, with input pins on the left and output or bidirectional pins on the right. For instance, when looking at a DDR memory part, address pins (A15:A0) can be grouped and placed on the left side of the part, data pins (D7:D0) can be grouped and placed on the right of the symbol, with additional pins grouped according to function.

Power and ground pins can be approached in multiple ways:

1. They can be placed on the symbol with the other signals.

2. Multiple symbols can be created for one part with powers\grounds on their own symbol or symbols. This is more common for large parts that have many power and ground pins.

3. Powers and grounds can be embedded in the part itself, basically hiding the electrical information.

The latter has its shortcomings and is not used in many situations, nor is it recommended. Although it makes for a cleaner schematic, not having the powers and grounds visible means referencing a data sheet to find which pins carry the power or ground when trying to troubleshoot the PCB in the lab.

During creation, keeping the size of pin symbols consistent will lead to a more aesthetically pleasing schematic. For example, two pin resistors, capacitors, and diodes should all take up roughly the same space on the schematic canvas. When building and using a schematic symbol, keep in mind that the symbol is meant to clearly show the electrical connectivity of the part and in no way is the symbol layout meant to reflect the actual mechanical layout of the footprint that will be placed on the printed circuit board.

Schematic Blocks

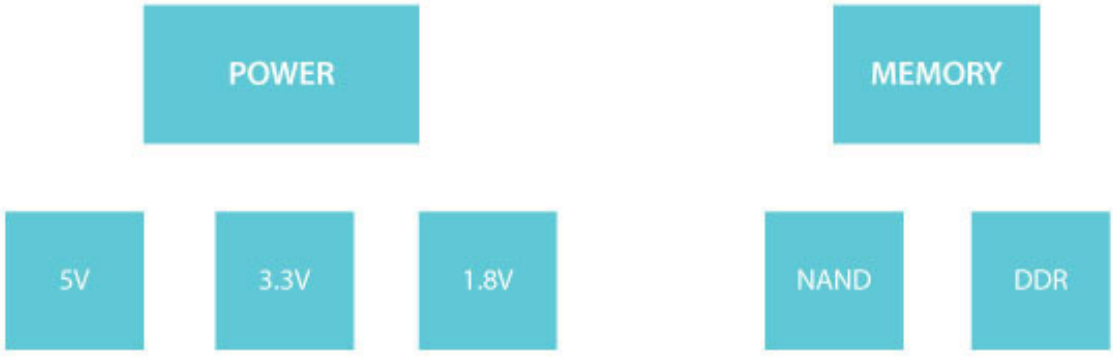
A very effective way to gather your thoughts and get the design committed to a certain form at the beginning is to start with a block diagram. Start with large generic blocks and then break each one into sub blocks. For instance, you may have a power supply block and a memory block.

The power supply block may have 5 volts, 3.3 volts, and 1.8 volts, with each of those blocks then containing the individual symbols necessary to generate those voltages. The memory block may contain a NAND and a DDR block. Each of those would contain the integrated circuit symbols, as well as any bypass capacitors and other associated circuitry. By breaking the design into these hierarchical blocks, which then may be associated to a single or multiple pages of the schematic, allows your design intention to be clearly added to paper.

Schematic

When placing symbols on the schematic, the information should be communicated as clearly as possible. Try to establish a left-to-right, top-to-bottom flow, similar to reading a book. Library symbols should have inputs on the left and outputs on the right, which lends itself to this methodology. Place symbols with plenty of room between them to allow connect lines to be drawn clearly and net information added without overlapping connect lines. Remember that, although you are viewing the schematic on a monitor in color, others may have only a black and white printout. Keep the reference designators near their respective symbols so there is no question about which component each belongs to.

A single symbol may map to several different footprints. When you make the selection to place it on the schematic, you are also selecting the footprint that will be on the PCB. For instance, the same resistor may be available in 0805, 0603, 0402, and 0201 footprint packages as well as radial or axial through hole.



Be consistent in your net naming throughout the schematic and embed as much information into the schematic description of the design as you can, as this will greatly enhance your layout. For instance, adding properties that define a clock signal as a differential pair communicates to the PCB designer information that is critical to the way the signals are routed and connected. Likewise, by defining addresses as a bus or setting minimum trace width properties in the schematic, you convey the information to the layout designer and ensure that requirements are not missed at the printed circuit board level. The goal is to put as much information as possible on the schematic to communicate the design intent while making it easy to read and understand.

The schematic is only as good as the library symbols and the data you include within it. Accurate information at the start is a must. By creating a block diagram of the entire system and breaking those blocks into sub blocks, while remaining clear, concise, and consistent with your net naming conventions, will contribute to a highly functional PCB. Embed as much information as you can into the schematic to ensure it makes it to the layout. Ask colleagues to review your schematics and provide feedback. A good schematic does not happen overnight. It takes practice and will improve over time. ■