

Schematic Best Practices

- Inputs on left side of schematic, outputs on right side of schematic (whenever possible). This makes the schematic easier to read.
- Don't put too much information on any one sheet. Group related items on same sheet of schematic. Don't try to put the entire schematic onto one sheet unless it's a small, simple design.
- For larger designs, make the front sheet an index and/or block diagram (see Intel example).
- Include a title block on each page, usually in the bottom right corner.
- Keep schematics up to date, including all of the features in your actual circuit.

- Use off sheet connectors (ports) for signals that cross page boundaries.
- Use the correct symbol for off sheet connectors (input to pin, output from pin, bi-directional).
- Consider including sheet grid location on off sheet connectors, so that it's easy to find the referenced signal on other sheets.
- Net names can be used to identify connections on a schematic sheet, without requiring a wire to be present on the schematic. For example, each pin that is labeled with the net name 'A15' is physically connected to the common address line A15. This can help make the schematic more readable by removing some of the wiring mess that results when lots of signals are connected on a single sheet.
- Buses can be used for groups of signals. Individual net names should be shown wherever a bus connects to components, so that the exact signal connected to be pin can be determined.

- Add comments to schematics to help explain circuit operation or load options, etc.
- Use solid or dotted line boxes around groups of special circuits or optional circuits to help make circuit operation more clear.
- Write comments in all caps using a sans-serif font (e.g. Arial), so that the comments are easy to read even if the schematic reduced or is printed out on smaller paper.
- Include CAD/layout notes, so that parts can be placed and routed correctly if a PCB is made.

- Consider putting a power/ground table in the schematic or on each sheet to identify which pins on each IC are connected to power or ground, especially if the symbols chosen for the parts do not display the power and ground pins. See the following example:

Reference Designator	+5V	+3.3V	Ground
U1	40		20
U2	2, 30	15	3, 16, 31
U3	14		7

1. Note: The demo version of Orcad limits schematic designs to a small number of components. You can create a separate design for each sheet of your schematic in order to work around this limitation. A full licensed version of Orcad is available in the lab – the full version does not have this limitation.
2. **Note: Save your schematics regularly (every 5-10 minutes)!! The program sometimes crashes.**
3. Note: Starter schematics are available on the course web site.
4. Note: In Orcad, you can adjust the sheet size to any size, using the 'custom' sheet size:
Use an X:Y sheet size ratio of 11.0/8.5 so that your design will print nicely on 8.5"x11.0" paper.
Don't use a sheet larger than 17"x13", since it will be too small when printed on 8.5"x11.0" paper.

OPTIONAL

Consider including the following information for every signal running "off board":

1. Voltage level(s) of the signal.
2. Frequency or frequency range of the signal. DC signals can be marked "DC".
3. Signal limitations, as relevant; e.g., maximum rise time, minimum setup and hold times, etc.
4. Any other special considerations for the signal. For example, if the signal is high-current, this can be noted to help avoid connecting the signal to sensitive components that could be damaged.