

General

The documentation package for ETec474 designs includes a schematic, a parts list, and, if any, engineering change forms. The following is a description of the documentation 'standards' for the schematic and parts list.

Schematics

The following is a set of standards for schematics designed in ETec474. These standards must be met to receive full credit. Schematics may be drawn by hand but normally should be generated using a CAD package. Recommended electronic CAD packages for schematic capture include: LogicWorks, Eagle, Visio, and pSpice. These packages are installed in ET340 (pSpice Professional is on hoh). If you intend to create a PCB, use Eagle.

The final size must be B-Size, 11x17, or, for smaller schematics, A-size, 8.5x11. For hand drawn schematics use a D-Size, 22x34, original and reduce to B-Size to submit. If you are using a CAD package that uses different conventions than those listed, notify the instructor for approval.

1. Your schematic must show the complete circuit for your project, including connectors, sensors, and power sources. If several connectors are used for either multiple PCBs or other devices, an interconnect diagram should be included.
2. All parts must have a reference designator and a value (or part number). The following are the standard reference designators:

B	Battery
C	Capacitors
D or CR	Diodes, Rectifiers
DSP	Displays(LED seven-segment, VF,etc.)
F	Fuse
K	Relay
KP	Keypad
L	Inductors, Coils
LCD	LCD Displays(DSP can also be used for LCDs)
J	Connector, Jack
LED	Single LED (Otherwise DSP)
M	Motor
nc	no connection (or X at end of wire)
OSC	Oscillators
P	Connector, Plug
Q	Transistors
R	Resistors, Potentiometers
RN	Resistor Networks
SP	Speaker
SW	Switch
T	Transformers
TP	Test points
U	Integrated Circuits
VR	Voltage Reference
X	Crystal
XDCR	Transducer
Z	Zener Diode

3. For ICs with multiple devices the devices are shown separately on the schematic with lettered designator suffixes A, B, C etc.(eg. U2A, U2B).
4. Every IC pin must be numbered and included on the schematic. Power and ground pins may be included on a separate power-ground table. The power-ground table must be on the first page of the schematic.
5. Pin numbers are shown on the outside edge of the IC. Pin definitions/names are shown on the inside of the IC where applicable.
6. All critical nodes should be labeled/named. Use the '/' or '_L' labeling convention for active-low signals.

Documentation Guidelines for ETec474

7. DeMorgan symbols and active level 'bubbles' must be used and matched, if practical.
8. A 'Last Designator Used' and 'Designator Not Used' table must be included.
9. If there is more than one ground node, identifiers must be used.
10. All unused devices must be shown on the schematic with connections or 'nc's.
11. Polarized capacitors must have a '+' to identify to positive terminal.
12. All designs created with an electronic CAD package must have its own library, which includes all parts used in the schematic.

Parts Lists

The parts list must be on a separate sheet attached to the schematic. It must include Item Number, Quantity, Description, and Designators List in the format shown. For any variation to the format, see the instructor.

Item	Quantity	Part Description	Designators
1	1	Cap-Mono,X7R,RAD,10%,.22 μ F/50V	C18
3	2	Cap-Mono,X7R,RAD,10%,2200pF/50V	C23, 26
5	16	Diode-Switching,IN914, 8nS, PIV75	CR1-16
7	2	IC-Lin,Audio Amp,MC34119,8DIP	U1,5
9	1	IC-Dig,6809,uP,40DIP	U2