Altium Designer Best Practices

PURPOSE

This document outlines schematic capture and PCB layout best practices and documentation requirements for WWU-EE design projects.

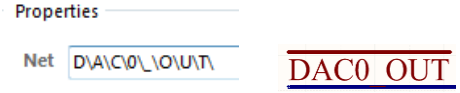
GENERAL

1. Your schematic must show the complete circuit for your project. If there are multiple PCBs, a separate documentation package is required for each assembly.
2. A system block diagram is required for designs that interface with external modules and/or contain multiple PCBs. The block diagram should show module interconnections, power sources, and general system information.

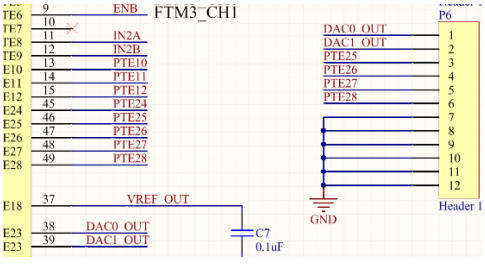
SCHEMATIC DRAWING

1. For each new Altium project, configure the tool to use WWU-EE Templates as described here: **R:\Altium\1-AltiumDesignerInitialConfiguration.pdf**
2. Populate the Title Block.
3. Draw the schematic for readability. A schematic is not just for the engineer to document the design. It may be used by many others down the line – layout engineers, test engineers, contract manufacturers, service technicians, production technicians, etc. These users need to be able to easily read and understand your schematic.
4. Signals should flow in a general left to right, top to bottom direction, when practical.
5. All critical nets should be labeled using a Net Label with a meaningful name. Net labels should be in all capital letters and consist of letters, numbers, and underscores only.

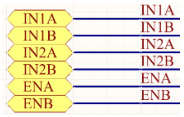
6. Indicate active low signals with an over-bar. To place an over-bar on a net label, add a forward slash '\' after each letter of the net name.



7. Think modular – power supply, microcontroller, signal conditioning, interfacing, etc. If it is practical and improves readability, give each module its own schematic sheet. Don't be afraid to spread things out and leave some white space if it makes the schematic easier to follow. Altium has the ability to create hierarchical schematics for complex designs.
8. To avoid spaghetti schematics, use Net Labels to make implied connections where practical. In the example below, rather than crisscrossing nets to connect PTE25 through PTE28, DAC0_OUT, and DAC1_OUT, use net labels to imply those connections.

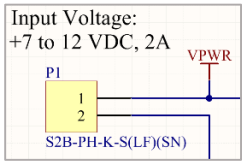


9. If a net needs to span across multiple pages of a schematic, use a net Port  in conjunction with a Net Label. If a net is contained to a single sheet, only a Net Label is required to imply a connection.



10. For power and ground connections use Power Ports and GND Ports.  Positive voltage power ports should have the horizontal bar on top, negative voltage power ports should have the horizontal bar on bottom. Ground Ports should always point down.

11. Use meaningful power port net names. For regulated voltages use more specific net names that imply nominal voltage levels: +5V, -15V, +3V3. For unregulated voltages or power inputs that can span a range, use more generic net names: VPWR, VSRC, etc.
12. For no-connect pins, place a **Non-Specific No ERC** symbol  on each unconnected pin.
13. Add text notes with pertinent information as needed. For example; power supply input voltage range, placement of critical components during PCB layout, test point voltages, etc. These notes should aid future readers of the schematic – layout engineer, test engineer, and service techs.

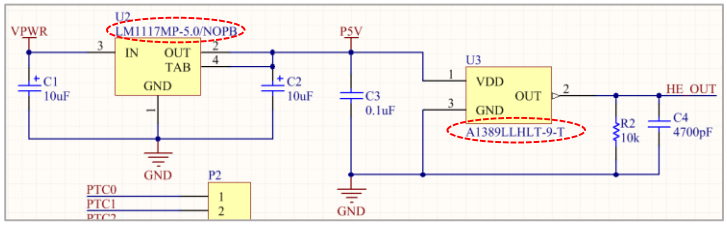


14. All schematic components must have a reference designator following the convention below. The reference designators should be placed in a way that makes it clear what symbol they are referencing.
For components not listed below, reference IEEE standard 315-1975, section 22.4.

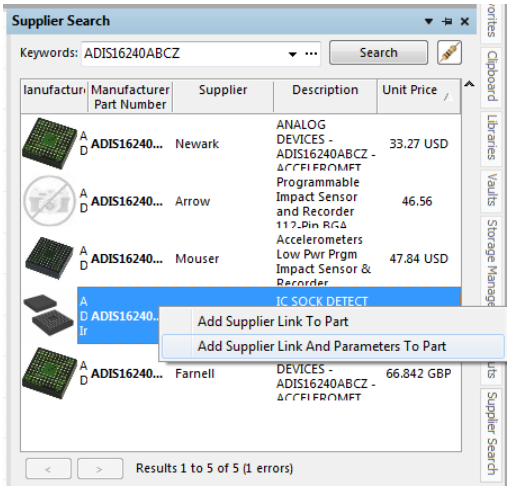
B	Battery
C	Capacitors
D or CR	Diodes, Rectifiers
DSP	Displays(LED seven-segment, VF,etc.)
F	Fuse
K	Relay
KP	Keypad
L	Inductors, Coils
LCD	LCD Displays(DSP can also be used for LCDs)
J	Connector, Jack
LED	Single LED (Otherwise DSP)
M	Motor
nc	no connection (or **X** at end of wire)
OSC	Oscillators
P	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

15. ICs with multiple devices in one package, the devices must be shown separately on the schematic with lettered designator suffixes A, B, C etc. (e.g. U2A, U2B).
16. Generic components like resistors, capacitors, and inductors must also show a value. The value should be placed below the reference designator. Here are value examples:
Resistors: 100, 10k, 10M. **Capacitors**: 10pF, 0.1uF, 1000uF. **Inductors**: 10uH, 10mH, 1000mH
17. Specific components such as op-amps, connectors, microcontrollers, and any special function ICs, must include a part number that is placed at the bottom left corner of the symbol or directly below the reference designator. This is necessary to allow the reader to look up the function of a component when it is not clear based on the

schematic symbol alone.



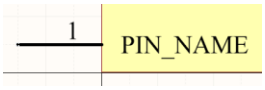
18. When multiple grounds are used, give each ground net a unique name. If multiple grounds need to be joined at a single junction use a Net Tie. The Net Tie symbol is found in the generic library: **WWUEE_SchematicLibrary**
    An explanation of Net Ties: https://www.smtnet.com/library/files/upload/NetTies-and-How-to-Use-Them.pdf
19. Use the **Supplier Search** tool to add supplier and manufacturer data to components as you add them to your schematic. To add data to a component, enter the part number in the search box, find the component from the preferred supplier, right click and select **Add Supplier Link And Parameters To Part**. Click on the target component within your schematic. Data will be automatically added to the target component.
    Adding supplier and manufacturer data to components will make generating a quality Bill of Materials much easier.



20. Each component in your schematic must have the follow parameters, populated with correct values: Manufacturer, Manufacturer Part Number, Description, Package/Case, Supplier, Suppler Unit Price 1
21. Every IC pin must be numbered and included on the schematic.
22. DeMorgan symbols and active level ‘bubbles’ must be used and matched, if practical.
23. A ‘Last Designator Use’ and ‘Designator Not Used’ table must be included.
24. All unused devices must be shown on the schematic with connection or no-connects. Tie unused IC pins to power or ground. For un-used op-amps, tie the positive and negative input terminals to opposite rails of the power supply to avoid erratic oscillatory behavior. Output can remain floating.
25. Polarized capacitors must have a ‘+’ to identify the positive terminal.
26.

SCHEMATIC SYMBOLS

1. Place pins on a grid size of 10. This will provide enough room for standard text sizes between pins and will help avoid schematic wiring issues. For more detailed symbol drawing use a grid size of 1. Press ‘g’ to toggle grid size.
2. Place pin names on the inside edge of the symbol outline and the pin numbers on the outside edge of the symbol outline.



For generic components like passives, turn off visibility of pin number and names.

3. Draw symbols following IEEE 315-1975 guidelines -- R:\Altium\Documents\IEEE315-1975SchematicSymbols.pdf. For symbols not mentioned in this standard, find examples from other reputable sources to replicate.
4. Add/update symbol Properties.
    - Set the Default Designator to have the appropriate prefix. U?, C?, R?, etc.
    - Add the manufacturer general part number to the Default Comment property. The general part number should be the manufacturer's part number excluding designators for variants of the part such as package type, temperature range, etc. This leaves the schematic symbol general enough to use for all variants of the component.
    - Add a complete meaningful description. Reference Digikey description for format.
    - Add a footprint.
    - Add a PCB3D model.
5. Name the schematic symbol using the following conventions:
    - For symbols that are unique to a single component from one manufacturer, use the manufacturer's general part number. The general part number is the part number excluding any designation of part variants.
    - For symbols that are more generic and may be used to reference various versions of a component from various manufacturers use the following format:
      {Device Type} {Critical Spec. 1} {Critical Spec. 2} …
      A general purpose dual op-amp is a good example of this. You may use the same symbol for an LM358, LM833, TL062, or other as long as they have the same pin out, therefore it is advantageous to give it a more generic name that accommodates all cases. It is preferred to give it a name like "Op-Amp Dual General Purpose".

PCB FOOTPRINTS

1. When possible use the **IPC Compliant Footprint Wizard** tool.
2. Each footprint must include a silkscreen reference designator.
3. Polarized components must have an orientation identifier on the silkscreen layer.
4. IC footprint must have a pin 1 identifier on the silkscreen.
5. Assign a PCB3D model. Models can be found at 3dcontentcentral.com.
6. Name the footprint following IPC-7351B (SMD) or IPC-7251 (through-hole) naming conventions. For specialized footprints not covered in these standards, use the following format: *{Mfg. Name}_{Mfg. Footprint Name}.* For connectors, follow a similar format, *{Mfg. Name}_{Mfg. Part Number}*.
   IPC-7251B found here: R:\Altium\Documents\IPC-7351B_7251NamingConvention.pdf

PCB LAYOUT

1. Use WWUEE Template design rules. The design rules are pre-configured for OSHPark capabilities. Itead, SeeedStudio, Advanced Circuits and PCB Pool offer the same capabilities.
2. For general signal routing use a preferred trace width of 8mil. For small pitch devices, use 6mil traces to fan out, then increase trace width where more space is available.
3. For traces carrying higher currents, use a trace width calculator to appropriately size traces. Typical copper weight from a low cost PCB manufacturer is 1oz/ft^2.
   Trace width calculator: http://www.desmith.net/NMdS/Electronics/TraceWidth.html
4. When transitioning layers with a trace, add multiple vias if the trace is carrying higher current.
5. For dense layouts, decrease silkscreen text size to 8mil width and 35mil height. Silkscreen width below 8mil may not print.
6. Critical connection information, such as connector polarity, IO port names, test point labels, etc. should be added to the silkscreen.
7. Run the design rule check to find routing errors.

DOCUMENTATION

1. Use WWUEE OutJob template to generate documentation. Using the OutJob template, you can produce a complete design package including:
    - Documentation
        - Schematics
        - Assembly Drawing
        - PCB 3D View, top and bottom
        - Dimensional drawing
    - Fabrication
        - Gerber files
        - NC Drill files
        - Pick and Place file
        - Assembly BOM
    - Purchase BOM
    - Checks
        - Design Rule Check
        - Differences Report
        - Electrical Rules Check
2. At a minimum, the Assembly Bill of Materials should include the following information.

Index	Quantity	Manufacturer	Manufacturer Part Number	Description	Package / Case	Designator
1	5	Wurth Electronics Inc	742792620	SMD EMI Suppression Ferrite Bead WE-CBF, Z = 100 Ohm	0603 (1608 Metric)	L1, L2, L3, L4, L5
2	1	Allegro MicroSystems, LLC	A1389LLHLT-9-T	IC SENSOR HALL EFFECT 5V SOT23W	SOT-23W	U3
3	1	Abracon	ABS07-32.768KHZ-T	Crystal, 32.768 kHz, 12.5 pF, -40 to 85 degC, SMD Low Profile 3.2 x 1.5 x 0.9 mm, Tape and Reel	2-SMD	XTAL1
4	1	Kingbright	APG1608SEKC/T	LED, SMT, 0603(1608), 0.25mm Tthickness, Super Bright Orange	0603 (1608 Metric)	LED1
5	2	NXP Semiconductors	BAS16	High-speed Switching Diode, 100 V, 4 ns, -65 to 150 degC, 3-Pin SOT23, RoHS, Tape and Reel	TO-236-3, SC-59, SOT-23-3	D2, D3
6	1	NXP Semiconductors	BZX84C13LT1G	DIODE ZENER 13V 225MW SOT23-3	TO-236-3, SC-59, SOT-23-3	D1

If you are using WWU-EE Altium templates, you can additionally generate a separate Purchasing BOM, that includes cost data. This BOM should be in the following format:

Index	Quantity	Manufacturer	Manufacturer Part Number	Description	Package / Case	Designator	Supplier 1	Supplier Unit Price 1
1	4	Kemet	T491C106M025AT	Polarized Capacitor (Axial)	2413 (6032 Metric)	C1, C2, C8, C9	Digi-Key	0.69
2	15	AVX Corporation	06033C104MAT2A	CAP 100nF 25V ±20% 0603 (1608 Metric) Thickness 1mm SMD	0603 (1608 Metric)	C3, C7, C10, C11, C12, C13, C14, C15, C16, C17, C18, C19, C20, C21, C22	Digi-Key	0.28
3	1	Kemet	C0603C472K3RACTU	CAP 4.7nF 25V ±10% 0603 (1608 Metric) Thickness 1mm SMD	0603 (1608 Metric)	C4	Digi-Key	0.1
4	2	Kemet	C0201C120K3GACTU	CAP 12pF 25V ±10% 0603 (1608 Metric) Thickness 1mm SMD	0201 (0603 Metric)	C5, C6	Digi-Key	0.1
5	4	AVX Corporation	TAJD107M016RNJ	CAP, TANT, 100uF, 16V, 20%, 2917	2917 (7343 Metric)	C23, C24, C25, C26	Digi-Key	1.3
6	1	Kemet	C0805C224Z3VACTU	CAP 220nF 25V -20% to +80% 0805 (2012 Metric) Thickness 1.45mm SMD	0805 (2012 Metric)	C27	Digi-Key	0.13