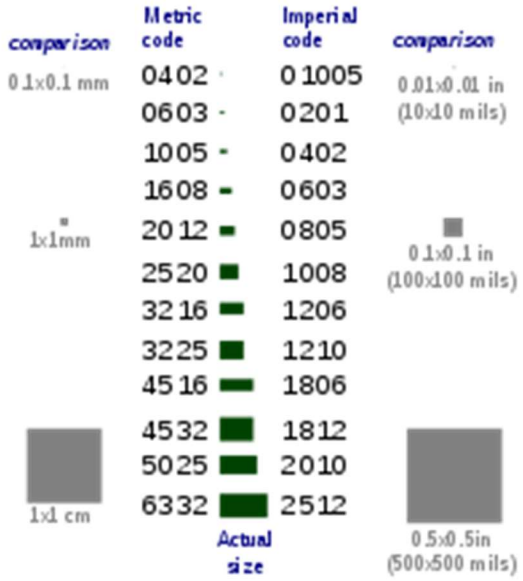
Senior Design Tips – Requirements for Senior Design PCB Fabrications
11/29/2024

1. As of Spring 2025 this document gives the requirements that must be met during the fabrication of PCBs for senior design. These requirements must be met prior to PCB review and approval during senior design 2. This document was created based on design and fabrication issues encountered in the past, with the goal of reducing the risk of not producing working PCBs by the end of senior design 2.

2. In general, when designing your PCB, there is a good chance that your team will have to repair and redesign your PCB (this is the standard engineering design process). Typically, over 90 percents of the boards do not work the first time. There are steps that can be taken to make the repair and redesign process easier (some of these will be discussed below). One of the most important items to consider is that you must do things early so that if your team needs to build a new board, you have the time to do so. Most PCBs take 2-3 iterations. Iteration #1, you may get about 80% there and you find design issues that cannot be fixed, and a new PCB must be built. Depending on the complexity of the PCB design, 3 or more iterations may be required.

3. To make it easier to solder your electronic components onto your PCB, parts smaller than 0603 should not be used. Unless a special justification is given why your group must use parts smaller than 0805, these PCBs will not be approved. Size comparison given below and actual size.

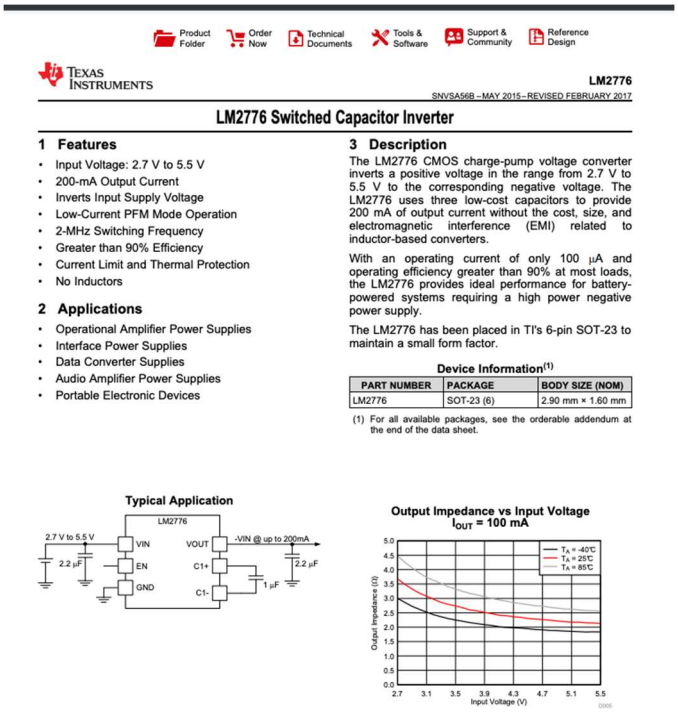


An exemption for parts smaller than 0805 (some regulators must use 0402) will be given for switching regulators as these circuits are very dependent on-board layout. For example, one

TI charge pump circuit needed the output and input capacitors attached directly onto the regulator IC pins.

Package		Approximate dimensions, length × width		Typical resistor power rating (W)
Metric	Imperial			
0201	008004	0.25 mm × 0.125 mm	0.010 in × 0.005 in	
03015	009005	0.3 mm × 0.15 mm	0.012 in × 0.006 in	0.02[34]
0402	01005	0.4 mm × 0.2 mm	0.016 in × 0.008 in	0.031[35]
0603	0201	0.6 mm × 0.3 mm	0.02 in × 0.01 in	0.05[35]
1005	0402	1.0 mm × 0.5 mm	0.04 in × 0.02 in	0.062[36]–0.1[35]
1608	0603	1.6 mm × 0.8 mm	0.06 in × 0.03 in	0.1[35]
2012	0805	2.0 mm × 1.25 mm	0.08 in × 0.05 in	0.125[35]
2520	1008	2.5 mm × 2.0 mm	0.10 in × 0.08 in	
3216	1206	3.2 mm × 1.6 mm	0.125 in × 0.06 in	0.25[35]
3225	1210	3.2 mm × 2.5 mm	0.125 in × 0.10 in	0.5[35]
4516	1806	4.5 mm × 1.6 mm	0.18 in × 0.06 in[37]	
4532	1812	4.5 mm × 3.2 mm	0.18 in × 0.125 in	0.75[35]
4564	1825	4.5 mm × 6.4 mm	0.18 in × 0.25 in	0.75[35]
5025	2010	5.0 mm × 2.5 mm	0.20 in × 0.10 in	0.75[35]
6332	2512	6.3 mm × 3.2 mm	0.25 in × 0.125 in	1[35]
6863	2725	6.9 mm × 6.3 mm	0.27 in × 0.25 in	3
7451	2920	7.4 mm × 5.1 mm	0.29 in × 0.20 in[38]	

From Wikipedia



Product Folder | Order Now | Technical Documents | Tools & Software | Support & Community | Reference Design

TEXAS INSTRUMENTS

LM2776

SNVSA56B – MAY 2015 – REVISED FEBRUARY 2017

LM2776 Switched Capacitor Inverter

1 Features

- Input Voltage: 2.7 V to 5.5 V
- 200-mA Output Current
- Inverts Input Supply Voltage
- Low-Current PFM Mode Operation
- 2-MHz Switching Frequency
- Greater than 90% Efficiency
- Current Limit and Thermal Protection
- No Inductors

2 Applications

- Operational Amplifier Power Supplies
- Interface Power Supplies
- Data Converter Supplies
- Audio Amplifier Power Supplies
- Portable Electronic Devices

3 Description

The LM2776 CMOS charge-pump voltage converter inverts a positive voltage in the range from 2.7 V to 5.5 V to the corresponding negative voltage. The LM2776 uses three low-cost capacitors to provide 200 mA of output current without the cost, size, and electromagnetic interference (EMI) related to inductor-based converters.

With an operating current of only 100 µA and operating efficiency greater than 90% at most loads, the LM2776 provides ideal performance for battery-powered systems requiring a high power negative power supply.

The LM2776 has been placed in TI's 6-pin SOT-23 to maintain a small form factor.

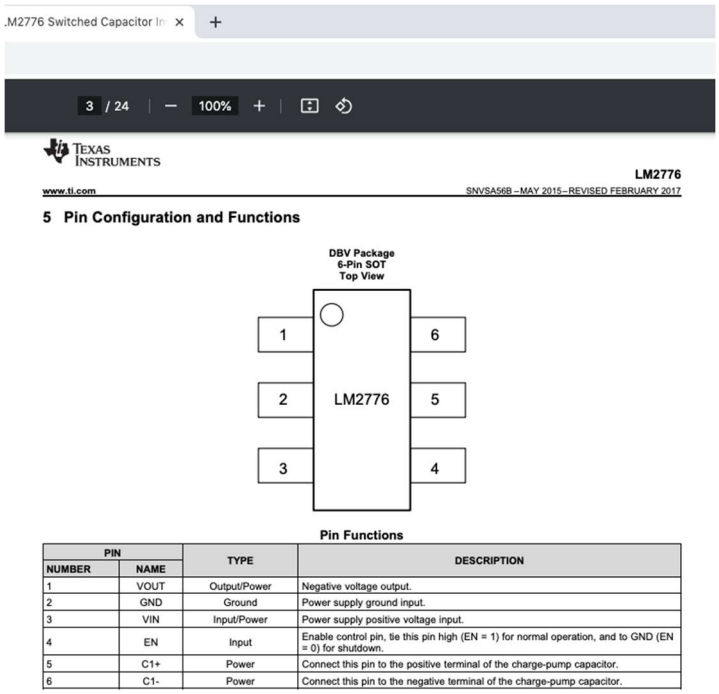
Device Information(1)

PART NUMBER	PACKAGE	BODY SIZE (NOM)
LM2776	SOT-23 (6)	2.90 mm × 1.60 mm

(1) For all available packages, see the orderable addendum at the end of the data sheet.

Typical Application

Output Impedance vs Input Voltage
I_{OUT} = 100 mA



.M2776 Switched Capacitor In ×

3 / 24 | 100%

TEXAS INSTRUMENTS

LM2776

www.ti.com

SNVSA56B – MAY 2015 – REVISED FEBRUARY 2017

5 Pin Configuration and Functions

DBV Package
6-Pin SOT
Top View

Pin Functions

PIN		TYPE	DESCRIPTION
NUMBER	NAME		
1	VOUT	Output/Power	Negative voltage output.
2	GND	Ground	Power supply ground input.
3	VIN	Input/Power	Power supply positive voltage input.
4	EN	Input	Enable control pin, tie this pin high (EN = 1) for normal operation, and to GND (EN = 0) for shutdown.
5	C1+	Power	Connect this pin to the positive terminal of the charge-pump capacitor.
6	C1-	Power	Connect this pin to the negative terminal of the charge-pump capacitor.

For this regulator to work, 0402 capacitors have to be used and placed directly onto pin 1-2 and 2-3.

From TI's datasheet for the LM2776 charge pump regulators.

4. If you must use parts smaller than 0603, be prepared to have these parts assembled by a PCB board manufacturer. Allow your group enough time.

   https://www.pcbway.com/pcb_prototype/trace-width-calculator.html
   https://www.pcbgogo.com/?gad-PCB-US&keyword=pcb%20fast%20prototyping&matchtype=p&device=c&network=g&placement=&gad_source=1&gclid=EAIaIQobChMIxte2utPMgwMV4YpaBR1XjgFxEAAYASAAEgLGA_D_BwE
   http://www.3pcb.com/

   Remember, the issue is not with the initial PCB assembly but any repair that must be performed after. For example (real examples from SD2), if the parts are very small and if your PCB stops functioning 1 day before the final demo and you cannot repair the board at UCF, you do not have enough time to fix the PCB before your final demo. In Fall 2024 SD2, a conservative estimate indicated that at least 10 ESP32 processor ICs needed to be replaced.

5. IC's come in many different types of packages. The link below lists the various type of IC packages. Things to consider beside the IC datasheet is the pin pitch (the distance between the pins), the type of soldering required, or if there are easily exposed pins on the sides. Many of the small surface mount packages have their pins underneath the IC package, making it extremely hard to rework a PCB after assembly. ICs with non-exposed pins require a reflow oven to solder these ICs to the PCB and a special rework process to remove and replace this type of IC. Preferably, pick an IC with the pins on the side and not underneath.

   https://en.wikipedia.org/wiki/List_of_integrated_circuit_packaging_types#Chip_carrier

6. If your team must use these high-density IC's, one method is to design a small daughter board that is used for mounting the IC (assembled by a board house). This board then has a set of header pins (like a breakout board) that are used to attach this daughter board to another PCB. You must also include all the support parts like capacitors, pull-up resistors and crystals. For example, this breakout PCB can be plugged into your main PCB via socket. One of the advantages of doing this, is if this IC fails, it can be easily replaced on your team's main PCB by plugging another one in since it is socketed. Hence, a new main board does not have to be rebuilt. Daughterboards should also be used for regulators and any motor controller or driver PCBs.
   For PCBs that use these high-density parts, PCB approval will only be given for designs that use these daughter boards in their design.

7. For motor PCB boards, diode protection must be used for the regulators and the motor drivers. Noise suppression capacitors should be placed across all brushed DC motors and back diodes should be used across the motor to prevent the high back EMF from destroying the parts.

8. It is highly recommended to use optical couplers between any digital logic (MCUs) and the motor drivers to prevent motor driver failures from destroying your MCU.

9. When ordering a blank PCB, your team must also order the solder stencil if you plan to solder the parts at UCF. Your team can order an unframed stencil for the reflow oven or your team can specify the size of the stencil required (framed stencil for rework).

10. Add test pads on traces as required. For example, if you have a trace that goes between two IC's and both have their pins under the package there is no way to look at the signal on this trace with a scope or a DMM.

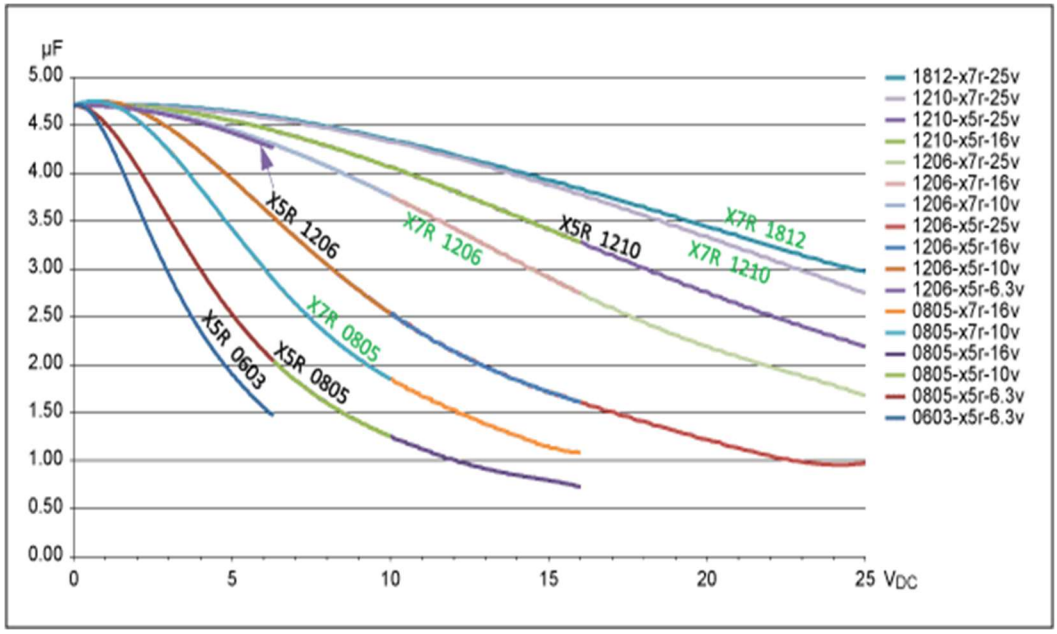
11. All PCBs should have a ground plane. This is especially important for analog circuits as well as digital circuits. In Fall 2024 SD2, an ESP32 board was built with no ground planes. The MCU part of the ESP32 worked but the IO pins did not. Most regulators will not work or start up without a proper ground plane.

12. As for board layout, most IC's datasheet specifications give the recommended PCB layout. The documentation states recommended but the specification datasheet really means REQUIRED.

13. All IC's power supply lines should have the proper bypass capacitors to maintain a clean power for the electronics. The links below discuss capacitors and the use proper use of bypass capacitors.

    Capacitor tutorial: https://www.youtube.com/watch?v=xlvqUts9H9c
    Bypass capacitors: https://www.youtube.com/watch?v=1xicZF9glH0
    Bypass capacitors why we use different sizes in parallel:
    https://www.youtube.com/watch?v=9EaTdc2mr34

14. The proper capacitors must be used as recommended by the datasheet. Most of the times, the capacitor recommended will be a ceramic capacitor. Do not forget that the actual capacitance value will change with operating voltage.



    Vishay suggests that it is due to the actual dielectric constant of the ceramic capacitor significantly changing under applied electrical field strength variations

    https://electronics.stackexchange.com/questions/216367/why-does-capacitance-value-changes-with-applied-voltage.

15. As with capacitors (operating voltage), the correct inductor must be used. The manufacturer of the inductor will give the maximum current allowed for the inductor and at the current the reduction in the inductance. As a safe rule, if I do not know the

maximum current required, I would use the DC current through the inductor and double this value in selecting the inductor current.

16. To make troubleshooting of power supply systems such as regulators much easier, a zero-ohm resistor should be added at the output of each regulator. This zero-ohm resistor can be initially left unpopulated, isolating the regulator circuit from the rest of the electronics making debugging of the regulator circuit easier. If you are not using daughterboards for your regulators, this is a must and required for PCB approval.

17. Regulators and PCB Layout: Some of the new switching regulators are now switching at a frequency of about 1-2 MHz. As such, these regulators are very sensitive to the PCB layout. It is highly recommended (required) that you follow the manufacturers' recommended (actually required) PCB layout given in the datasheet. Also, to make PCB board layout easier, it is recommended that some of the older regulators be used. They are not as efficient, but they use a lower switching frequency making PCB layout easier.

    The newer regulators are very small and are very hard to solder. Take this into consideration as regulator PCBs are one of the most common failures in SD2.

18. All regulator circuits can be tested using an electronic load test equipment. There are several of these electronic loads in the senior design lab. Any team wishing to use one, please contact Dr. Weeks and he will demonstrate its proper use.

19. Besides the datasheet maximum current and voltage output from a regulator, its package thermal resistance along with the thermal resistance of the PCB must be considered. With these small regulator ICs, it is easy to exceed the maximum junction temperature of the regulator. Any team wishing to determine if they meet the thermal specifications for their chosen regulator, please contact Dr. Weeks and he will go through the calculations with the team. Sometimes to meet the thermal specifications, a heatsink must be used.

20. When laying out your PCB, trace widths must be considered. Many times, the default trace is 0.015". To give you a perspective, a 1/64" is equal to 0.0156". I highly recommend that before you layout a PCB, get a ruler and look at the size of 1/64". A trace this small makes PCB rework nearly impossible. A minimum or of 0.033" or larger should be used where possible.

21. Since power supply traces typically draw a lot more current than signal traces, power traces are typically much wider. The links below from the Institute of Printed Circuits (IPC) give a calculator that calculates the trace width for a given current.

    https://www.pcbway.com/pcb_prototype/trace-width-calculator.html
    https://resources.altium.com/p/ipc-2221-calculator-pcb-trace-current-and-heating
    https://www.4pcb.com/trace-width-calculator.html

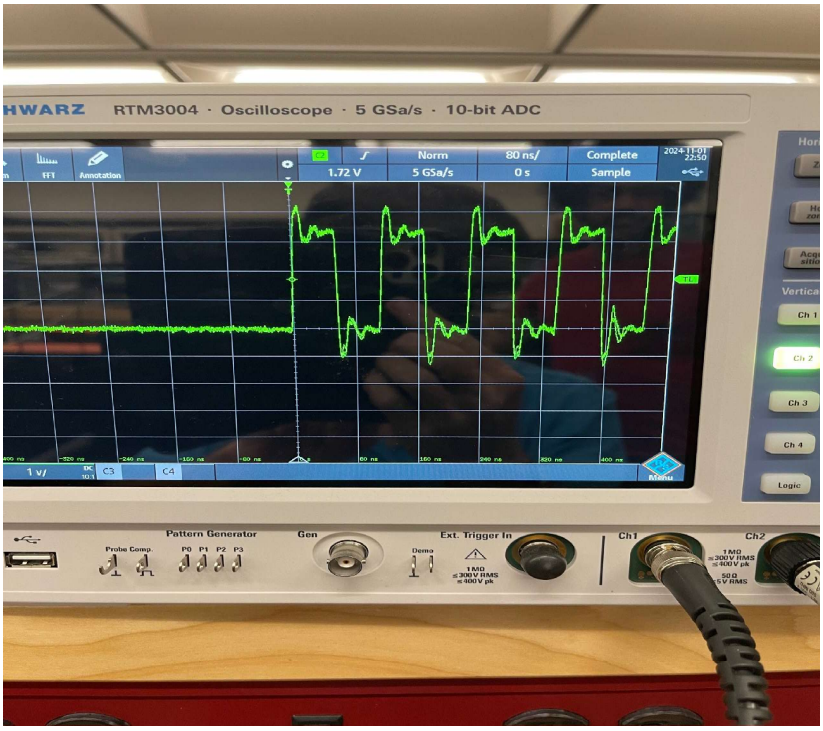
https://www.digikey.com/en/resources/conversion-calculators/conversion-calculator-pcb-trace-width

22. Separation between traces and separation between ground planes: As stated earlier, having 0.015” separations between traces, and between traces and ground is too small making PCB rework nearly impossible. A minimum of 0.033” or larger should be used where possible.

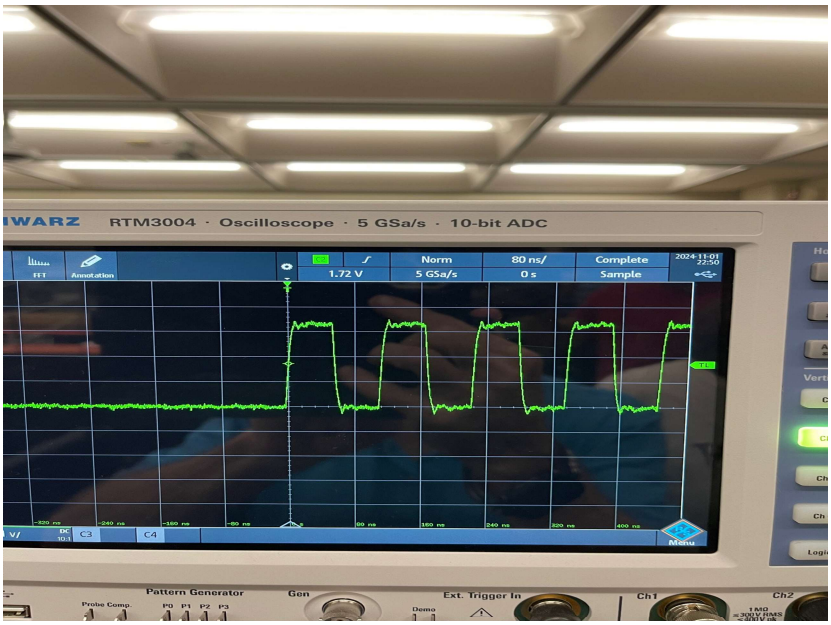
23. Sometimes it is desired to cut a trace when you are unsure of the design. To make this possible, one method is to put a zero-ohm resistor in series with this trace. The other method is to use two pads connected to this trace that are closely separated. If desired, the trace between these two pads can easily be cut at a later time.

24. All digital inputs need to be defined as either a “0” or a “1”. No digital input lines should be left floating. If you are unsure what digital value is needed, you can use low value resistor tied either to Vcc or ground. If you need to change its value, one side of the resistor can be lifted and a small wire can then be tied to either ground or Vcc.

25. Many serial communication protocols require proper impedance termination to function properly.



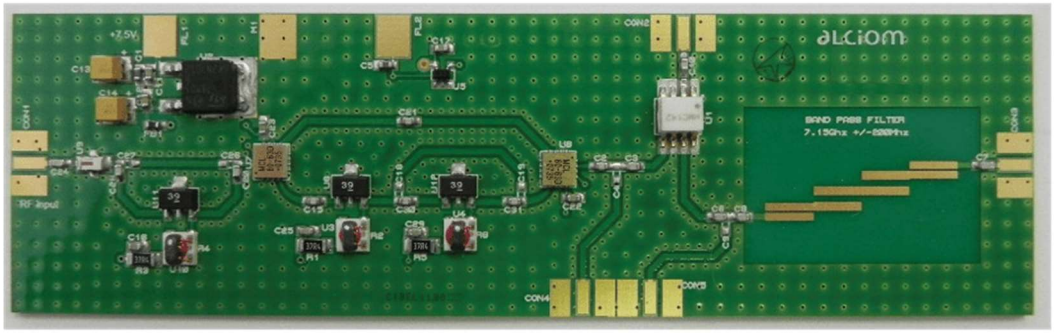
Improper terminated SCLK line.



Properly terminated SCLK line.

Improper terminated digital lines can lead to failure to the digital IC's or even result in software perceived issues where bits in memory or registers are changed (i.e. A = 5 gets changed to A = 4).

26. If your PCB needs to have ground planes on various layers of your PCB, PCB vias are used to tie these ground planes together. The link and the image below give an example of using PCB vias to tie two layers together.



From Circuit Cellar

https://circuitcellar.com/research-design-hub/basics-of-design/lets-understand-ground-stitching-vias/

Grounding and proper wiring are required. Specially, when high current pulses are generated when turning motors on and off. In Fall 2024 SD2, there was radiated EMI emissions resetting / stopping the ESP32 from working.

27. ESP32 VROOM users: The ESP32 needs RS232C communications to program the device. An USB to rs232 converter IC is included on the development boards. When designing your stand-alone ESP32 you must also include one of these ICs on your PCB (for example, CP2102N-A01-GQFN28).

    You should also include zero-ohm resistors on the transmit and receive lines between the MCU and the USB to UART bridge to facilitate debugging.

    https://dl.espressif.com/dl/schematics/esp32_devkitc_v4-sch.pdf

28. ESP32 VROOM users: Additionally, to program the ESP32, you need two switches to be placed on pins IO0 and EN. The schematic in the link below will help you better understand how to implement these two switches.

    https://dl.espressif.com/dl/schematics/esp32_devkitc_v4-sch.pdf

29. We are in the DRY season when electro-static-discharge (ESD) becomes an issue. These discharge pulses can be over 10kV and can last for a few picoseconds destroying electronic components. As such, there are grounding straps in the SD2 lab that should be used.

30. When selecting your MCU, your group must not only consider its cost but ease of programming and ease of hardware design. Arduino base processors make programming much easier when interfacing to sensors. Adafruit and Sparkfun have many libraries available for many different types of sensors. As for hardware development, some MCUs need an external crystal and specialized tools to download software from the IDE to the MCU. Take this into consideration. Also, some MCUs have a long lead time, and many PCB fabrication vendors do not have them in stock. Some PCB vendors won't solder the high density MCUs leaving you to solder them at UCF.

31. **For those groups that involve optical detectors, prior to PCB approval a breadboard of your detector circuit must be completed and tested. This is a joint project between the EE students and the optical students with both expertise contributing.**