

**ME461 Semester Project #2**  
**Learning EagleCAD and Constructing your own F28027 Board**  
**Due October 13<sup>th</sup> 2:00PM**

Make sure to read through this entire document before getting started so you know all the parts of the project and can plan accordingly.

The goal of this project is to teach you how to use the printed circuit board CAD software EagleCAD. I have created quite a number of videos instructing the use of EagleCAD 7.7. You can find all these videos, in addition to the install executable and other needed files, at the class lecture's Box folder under the folder "ME461ClassFiles/EagleCAD." You will be performing all the steps to build your own circuit board short of actually sending your credit card to the fabrication company "Advanced Circuits" [www.4pcb.com](http://www.4pcb.com). NOTE: You will NOT be purchasing your circuit board. I have already fabricated the base circuit board if you are interested in seeing it.

If you search online, you will find that EagleCAD is now at version 9.6 and owned by Autodesk. I am still using version 7.7, *one of the last versions by Cadsoft who was bought by Autodesk*, because the College of Engineering has not purchased the latest license. So I recommend for this class that you perform this project with version 7.7 and then when you get used to EagleCAD you will be able switch to the latest version if you need to outside of this class.

You will first watch the videos I created on how to use EagleCAD 7.7 and any other training you can find online. Note that most of these videos were recorded spring 2020 when, my other class, SE423 was asked to stay at home due to the COVID-19 pandemic. That semester the goal was to instruct the students how to build the circuit board for their own Segbot robot based on a MSP430 microcontroller, which is different from the TMS320F28037 processor I am asking you to use for this project. So the circuits that I build in these videos are somewhat similar to what you will be constructing but definitely not the same. As you watch these videos, feel free to perform the same steps that I do but note that your circuit is different.

After you have finished watching my training videos, you are ready to copy the board schematic given below. This given schematic is the base schematic that each of you need to construct. In addition to this base circuit, you can choose to work on an idea I have on constructing an inexpensive quadrotor design or you can go your own direction and pick three sensors to add to your board. I give more details about the quadrotor design later in this document. If you go the "three sensor" route, the sensors you choose are up to you. The only constraint is that one of those three sensors must be a "surface mount" board mount sensor. I give some websites below that have a large number of board mount sensors. For these three sensors, you will need to create in EagleCAD a "symbol", "package" and "device" for each. Then use these newly created devices in your circuit board and wire them to the TMS320F28027 processor. NO, you cannot find a library online and use it.

**Items to Complete by Due Date. Your job is to pace yourself so you get everything done by the due date.**

1. View all the EagleCAD instructional videos I recorded. As you are viewing the videos you will probably want to build some of the circuit that I build while recording the video to help you learn the different steps of circuit board construction. The circuit that I build in these videos is not the circuit you will be building for this project. You do not need to turn in these “scratch” files that you create while learning EagleCAD.
2. Create the base Schematic given below. Make sure to look at the full schematic and the descriptions of the zoomed in parts of the circuit below.
3. When finished with the base circuit’s schematic, it is probably a good idea to work on the board layout to organize the parts generally in correct positions before adding the quadrotor parts or your three chosen sensors. Your circuit board can be as large as you need, but the idea of the base board is to have it connect to a Raspberry Pi board. Also **very important**, make sure your 20X2 pin female header on your board is orientated such that it connects correctly to the 20X2 male header of a Raspberry Pi. The female header on your board will be soldered on the bottom of the board.
4. With the board somewhat laid out, you are ready to add the quadrotor parts or the three sensors you chose to add to your circuit board and interface with the TMS320F28027 processor. If you have chosen the “three sensor” route, I need to approve the sensors you pick, so send me an email with the sensors you would like to use. I want to make sure you are not picking a sensor that is too easy/hard to attach. Then, as described in the training videos, you will create an EagleCAD symbol, package and device for both of these sensors. If it is a “through hole” device, you will need to indicate the outline of the sensor in your package. You should save these parts in your own EagleCAD part library so you can submit it along with your other files. Then add these three sensors to your board and wire them appropriately to your TMS320F28027 processor. Some of the wires connecting your sensor may also be connected to the 20X2 raspberry pi connector. For example if your sensor uses the I2C serial port to communicate, you will have the SDA and SCL pins connected to your sensor and to the 20X2 connector.
5. In your schematic, run the ERC error check. Take a **screen shot** showing that you have no critical errors. There will be some errors that do not need correcting and are just possible warnings.
6. Now that all your parts are on your board, you need to decide where to place all your parts on the board remembering that they cannot be too close together because wires still need to be routed to connect devices. You will be using 8 MIL (thousandth of an inch) width traces for most signals. For GND, +3.3V and +5V you will be using 20 MIL width traces (except where the pad connected, has a smaller width). Use the “autorouter” to route most traces on your board. For those GND and power pads that are smaller than 20 MILs you will need to hand route first and then run the autorouter. See the training videos again on how to do that. Take a **screen shot** showing your route achieved 100% routing. In addition, when you have a 100% routing, add a GND area fill to act as a ground plane on both the top and bottom layers. Take a **screen shot** of the completed board.

7. Run the DRC Error check. Watch the videos and see EagleCAD Tips.pdf to find all the settings for the Design Rules Check. Take a **screen shot** of the DRC results. You will find that there are some errors that are OK. For example a GND wire's width is small than 20MILs at the surface mount pads that are smaller than 20MILs.
8. Go through the steps to fabricate your circuit board, but DO NOT order your board.
  - a. Generate your Gerber files with the given .cam Job file given at Box.
  - b. Zip all necessary Gerber files and send to [www.4pcb.com](http://www.4pcb.com) "Free PCB File Check". The videos help with this along with EagleCAD Tips.pdf.
  - c. Wait for an email back from 4pcb.com say that you have "No Show Stoppers." Take a **screen shot** of that email indicating no show stoppers.
9. Submit to your Box folder:
  - a. All above requested screen shots.
  - b. Gerber files zip file
  - c. Zip of all your project files including your sensors library file.
  - d. PDF print out or clear screen shot of your final schematic.
  - e. Clear screen shot of your final board layout and routing.
  - f. Clear screen shots of the part's library Device view that you created for the quadrotor design or the three sensor's library Device view.

**Parts Needed for Base circuit (If I forgot something of course add it)**

ME461EagleLibrary.lbr (given in Box folder)

TH\_PUSHBUTTON  
 FT232RL  
 MINI-USB-SHEILD-UX60A-MB-5ST  
 TLV1117-3V3  
 TMS320F28027DAS

rcl.lbr

C-US  
 Find a 1206 size capacitor. Change its value to specify Farads in your schematic

CPOL-US  
 CPOL-USE3.5-8 100uF larger through hole capacitor. Change its value to specify Farads in your schematic. Need only one. For regulator's output.

R-US\_  
 Find a 1206 size resistor. Change its value to specify Ohms in your schematic.

L-US  
 L-USL2012C

con-lstb.lbr

MA03-01      Make sure Pin 1 is a square pad. You may have to modify the associated package.  
 MA03-02      Make sure Pin 1 is a square pad. You may have to modify the associated package.  
 MA07-02      Make sure Pin 1 is a square pad. You may have to modify the associated package.  
 MA20-02      Make sure Pin 1 is a square pad. You may have to modify the associated package.

Pinhead

PINHD-1X2

PINHD-1X2

Make sure Pin 1 is square and Pin 2 is octagon. You may have to change the associated package.

Led

LED

LEDCHIPLED\_1206

**Choose Three Sensors to Add to the Base Circuit, Create a EagleCAD library for the Sensors and Wire the Sensors to the TMS320F28027 Processor on your Board.**

The sensors you choose need to be board mounted sensors in either a surface mount package or through-hole package (this includes a sensor that has already been solder to its own circuit board and then can be connected with header pins.) One of the three sensors MUST be in a surface mount package. No cables connections, at least for the three sensors you need to choose. If you want to add a fourth sensor that has a cable connection, go for it. The three sensors need to be approved by me. In your EagleCAD package for the part, make sure to add an outline of sensor in relation to its pins or pads. Even if the sensor just connects using header pins, you need to go through all the steps of creating the sensor's EagleCAD "package", "symbol" and "device."

Possible sensor websites. You can choose a sensor for a different website also.

[https://www.digikey.com/en/resources/sensors/index?WT.z\\_vanity=SensorSelector](https://www.digikey.com/en/resources/sensors/index?WT.z_vanity=SensorSelector) DigiKey Sensors

<https://www.analog.com/en/products/sensors-mems.html> Analog Devices Sensors

<https://www.pololu.com/category/7/sensors> Pololu Sensors

<https://www.adafruit.com/category/35> Adafruit sensors

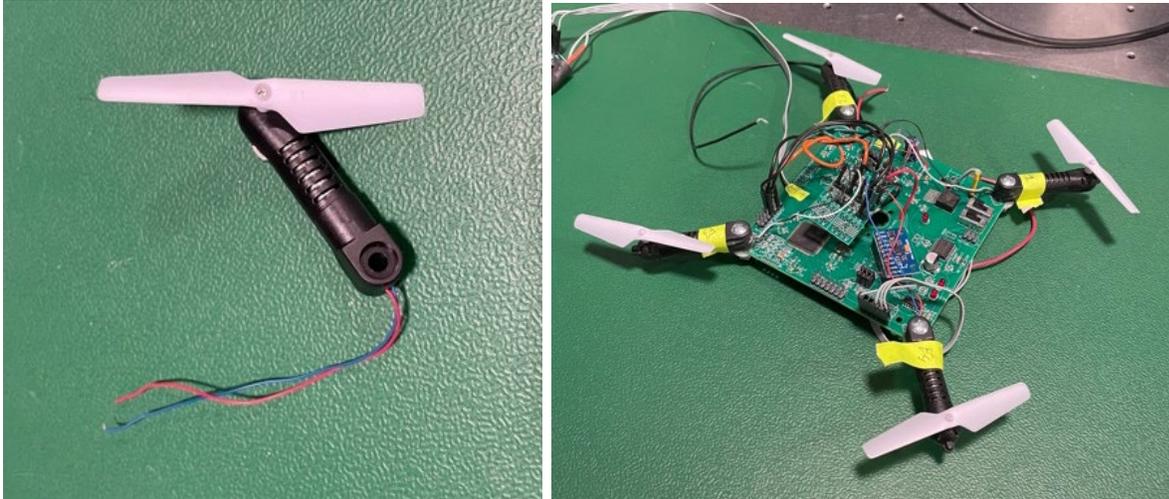
<https://www.sparkfun.com/categories/23> Spark fun sensors

<https://www.robotshop.com/en/sensors.html> Robotshop sensors

<https://solarbotics.com/product-category/parts/sensors/> Solarbotics Sensors

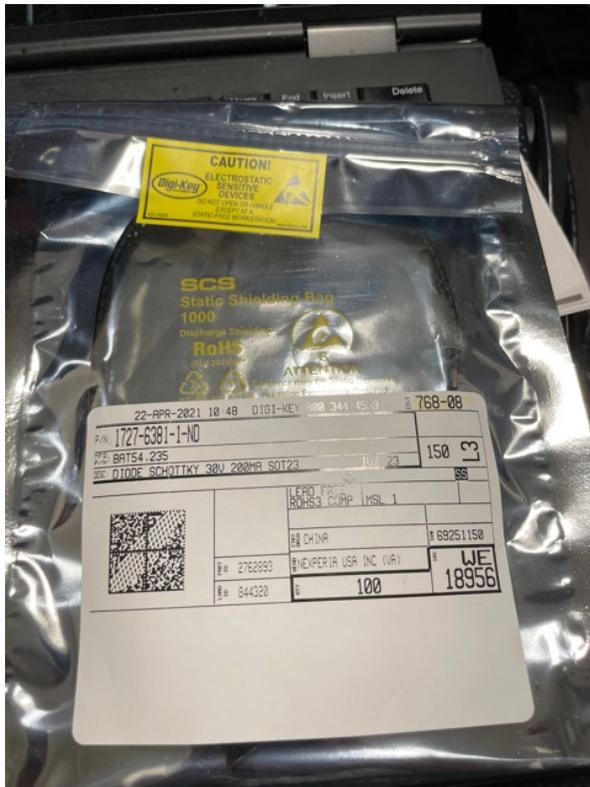
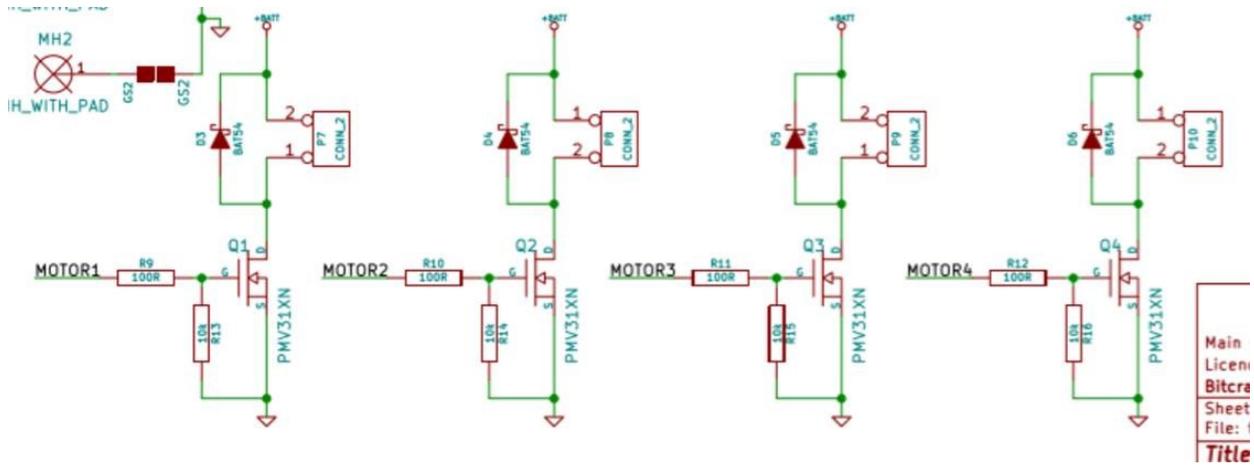
**My Idea for a Quadrotor vehicle using the F28027 base board as a starting point.**

A few summers ago, I found some inexpensive quadrotor spare part motors. I attempted to create a quadrotor experiment that did not necessarily fly, but would be able to stabilize itself when sitting on a point. The advantage of not having the quadrotor fly, and just stabilize on a point, is that you do not have to consider the quadrotor carrying a battery. It would be a nice experiment to understand the stabilization control algorithm of a quadrotor. The experiment worked ok, but the circuit board was a bit too heavy. I am interested in trying out the experiment again using this smaller F28027 base board.

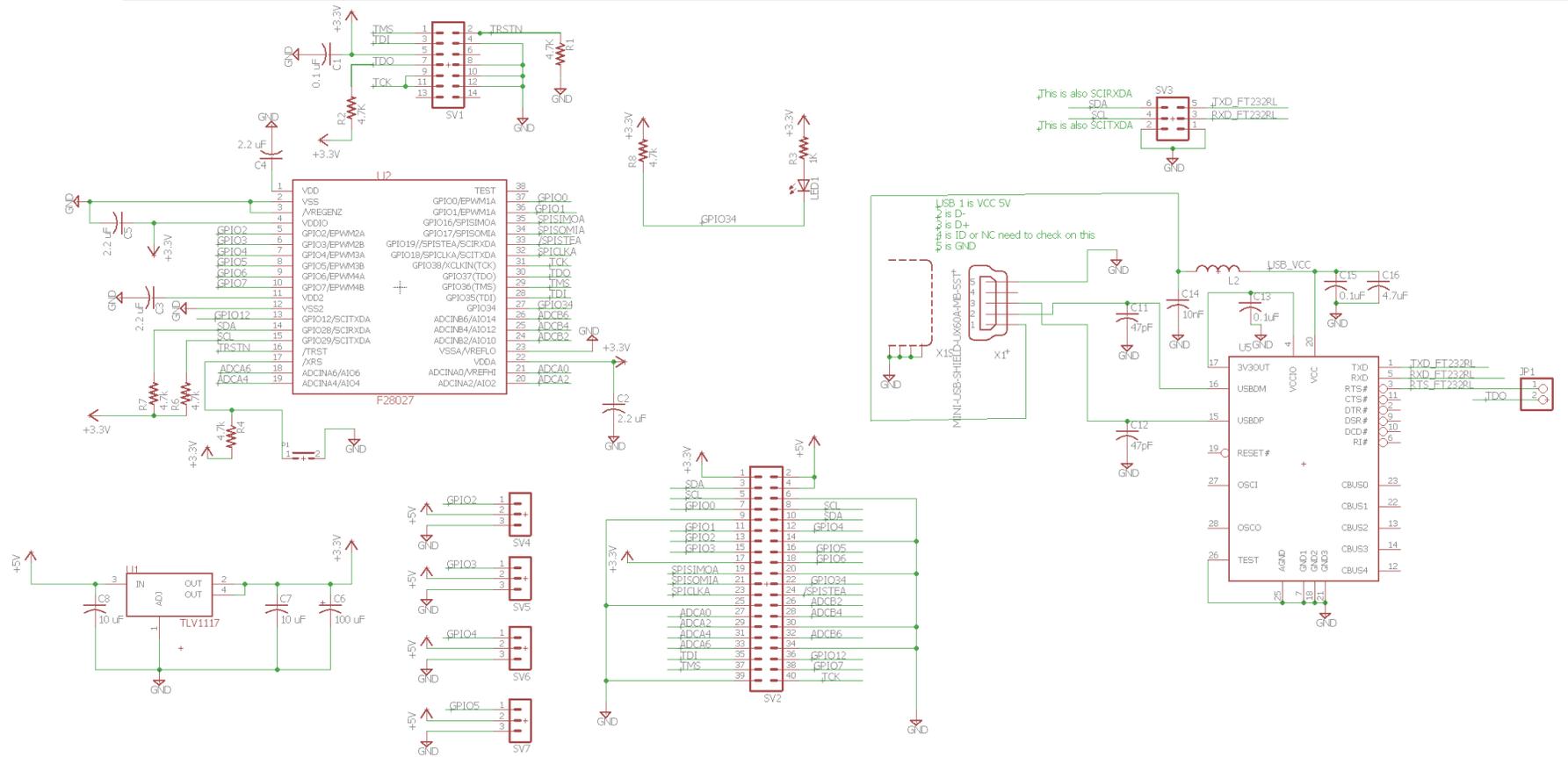


If you decide to go this route, you will add a few items to the base board to give it an IMU sensor along with the circuit to drive four DC motors. You will need to add to the base board:

1. Add a hole at each corner of the board to mount the motor/propeller actuator. Ask me for one of motor/propeller housing when you are doing the design so you can take measurements. Think about ways to hold the motor housing in place on the board in addition to using a machine screw and nut.
2. Each motor/propeller actuator will need a MOSFET to drive current to the motor. The MOSFET will be driven by a PWM signal from the F28027 chip. PWM duty cycle will be proportional to the current driven to the motor. Use the circuit below as a guide. Instead of the PMV31XN part, I instead have a similar part DMN2050L-7. Find the Digi-Key parts in the below pictures. Go to Digi-Key's website to find the data sheet for both the DMN2050L-7 MOSFET and BAT54 fly-back diode. You will need to create surface mount parts for these devices in EagleCAD. In the below schematic, MOTOR1, MOTOR2, MOTOR3, and MOTOR4 are the PWM signals from the F28027 chip. P7, P8, P9 and P10 are the connections to the motor+ and motor- wires of the motor housing. +BATT will be a 4.0V power source. Each MOSFET has a 100ohm and 10Kohm resistor in its circuit.
3. Close to the center of your board, you should add the MPU-9250 IMU board that you also have on your lab "green" board. You will need to create an EagleCAD "through-hole" device for this MPU-9250 board.

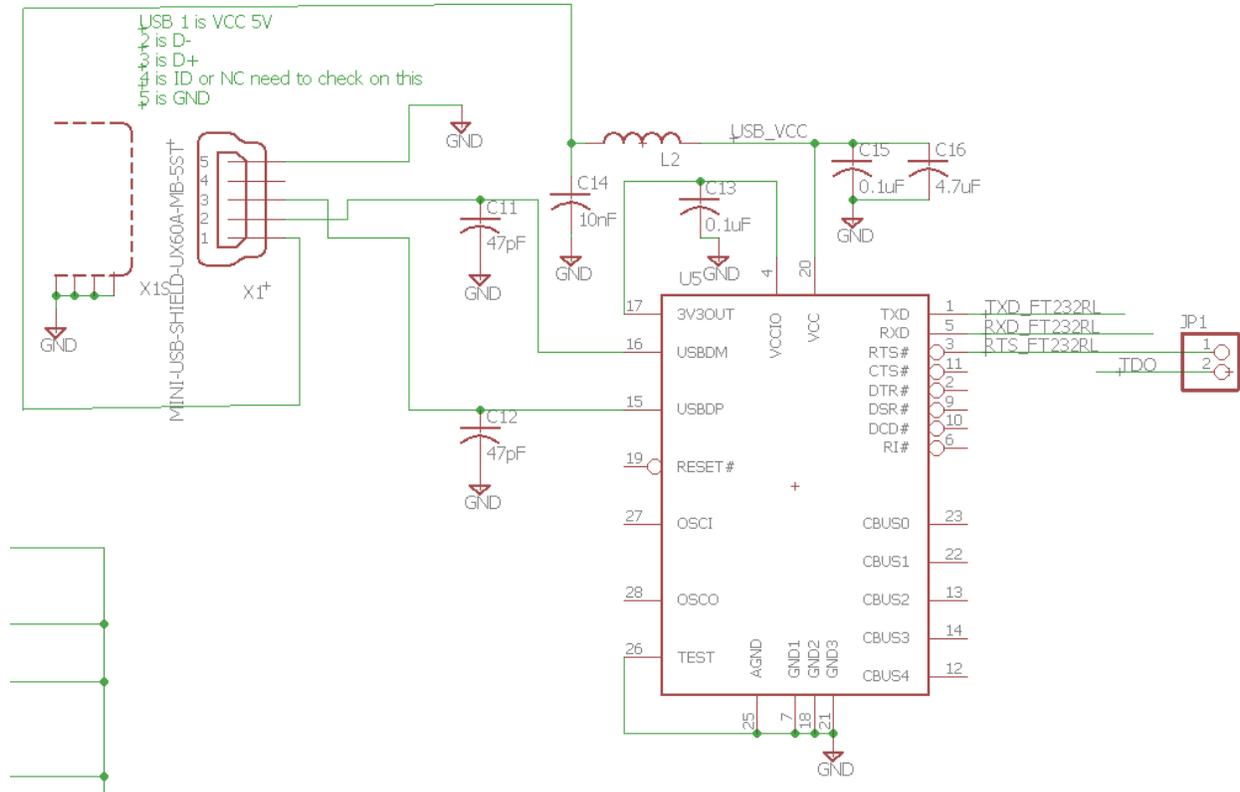


**Base Schematic you need to reproduce and then add your two additional sensors. Zoom in if you cannot see some text.**

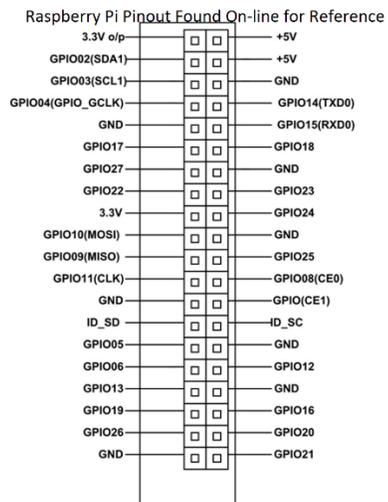
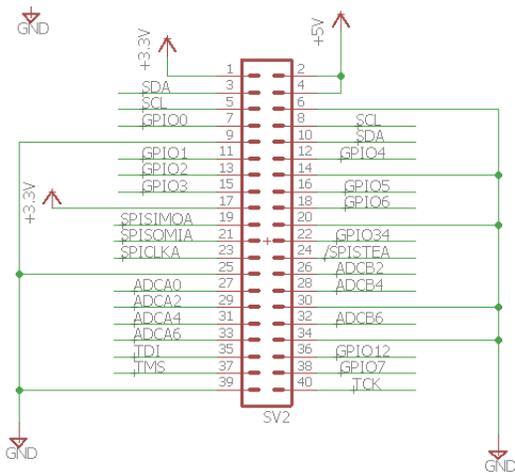
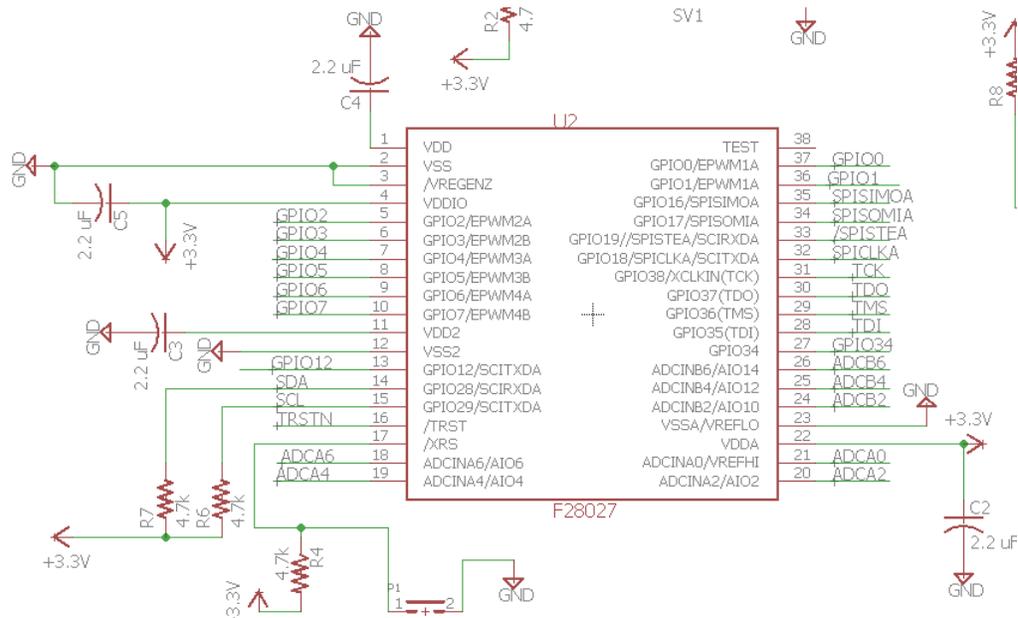


## Breakout of parts of the Base Circuit

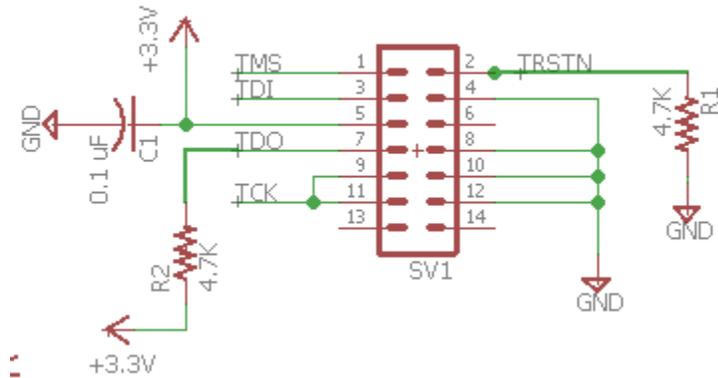
This is the virtual USB COM port circuit that converts the TMS320F28027's UART (SCIA) to USB. I recommend you start with this circuit first. The USB connector and FTDI FT232RL chip are in the ME461Eagle\_Library. The rest of the parts are found in the pinhead and rcl libraries, see above. Do not worry if your chips are labeled differently. For example the FT232RL chip is U5, but your board may have it U1. JP1 is a jumper that can connect the RTS (ready to send) pin of the serial port to a boot pin of the TMS320F28027 processor.



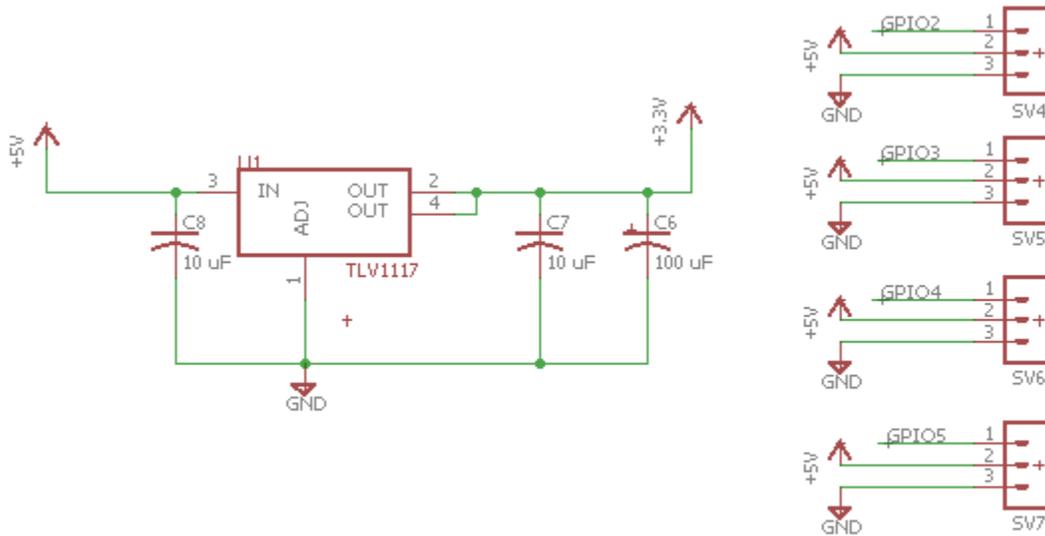
The TMS320F28027 processor. This processor is the lowest speed and lowest memory version of the C2800 processors. It also comes in a package that is “pretty” straightforward to solder and that is the reason I chose it for this board design. When you are laying out this board make sure to keep the four 2.2uF capacitors within at least one centimeter (or so) of the power pin it is connected to. This processor has an internal crystal of 10MHz and that is clocked up by its PLL to 60MHz. It is a slower processor with fewer peripherals and pins but is programmed in the same fashion as the TMS320F28379D processor.



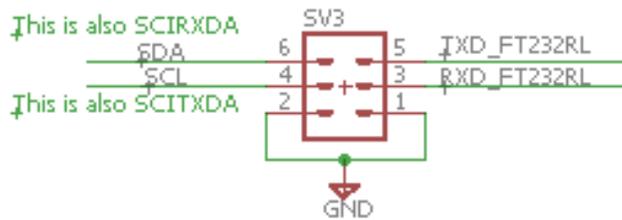
JTAG connector.

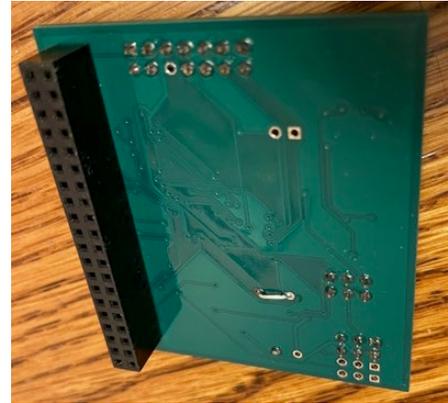


Regulator for 5 volts to 3.3 volts. The 5V would come from the 20X2 connector. We are not going to populate this for lab. This is needed if the board will be a standalone board. Also connectors for 4 RC servos that are wired to TMS320F28027 EPWM pins.



The board's one LED connected to GPIO34, which is a standard choice for C2000 development boards. GPIO34 also determines boot mode on power up. We always want it high on power up so that it why it is pulled up through a 4.7K ohm resistor to 3.3V. The 1K ohm resistor is current limiting the current through the LED to protect both the LED and the GPIO pin.





This are pictures of the base board that I fabricated. Your board does not have to be this exact size. You can make your board different dimensions and position the Raspberry Pi connector and your sensors (or quadrotor parts) however it works for your design.