Progress Report 2/10/19 - 2/16/19
Kevin DeVincentis, Team: A2

This week I worked on creating the schematic for our first PCB revision. I also began working on the layout and routing of the board. For our first revision, the PCB will be 2.5” X 2”. It will have breakouts for the IO pins on the ESP32 microcontroller. We decided on the SPH0645LM4H-B MEMS microphone, available on digikey, for it’s high signal to noise ratio (65db). Our wake up mic will be the Vesper VM1010 as it is a leading microphone marketed as a wake up mic, with the features we need. Since we would like to experiment with the VM1010, we will incorporate breakouts for the ports on the mic.

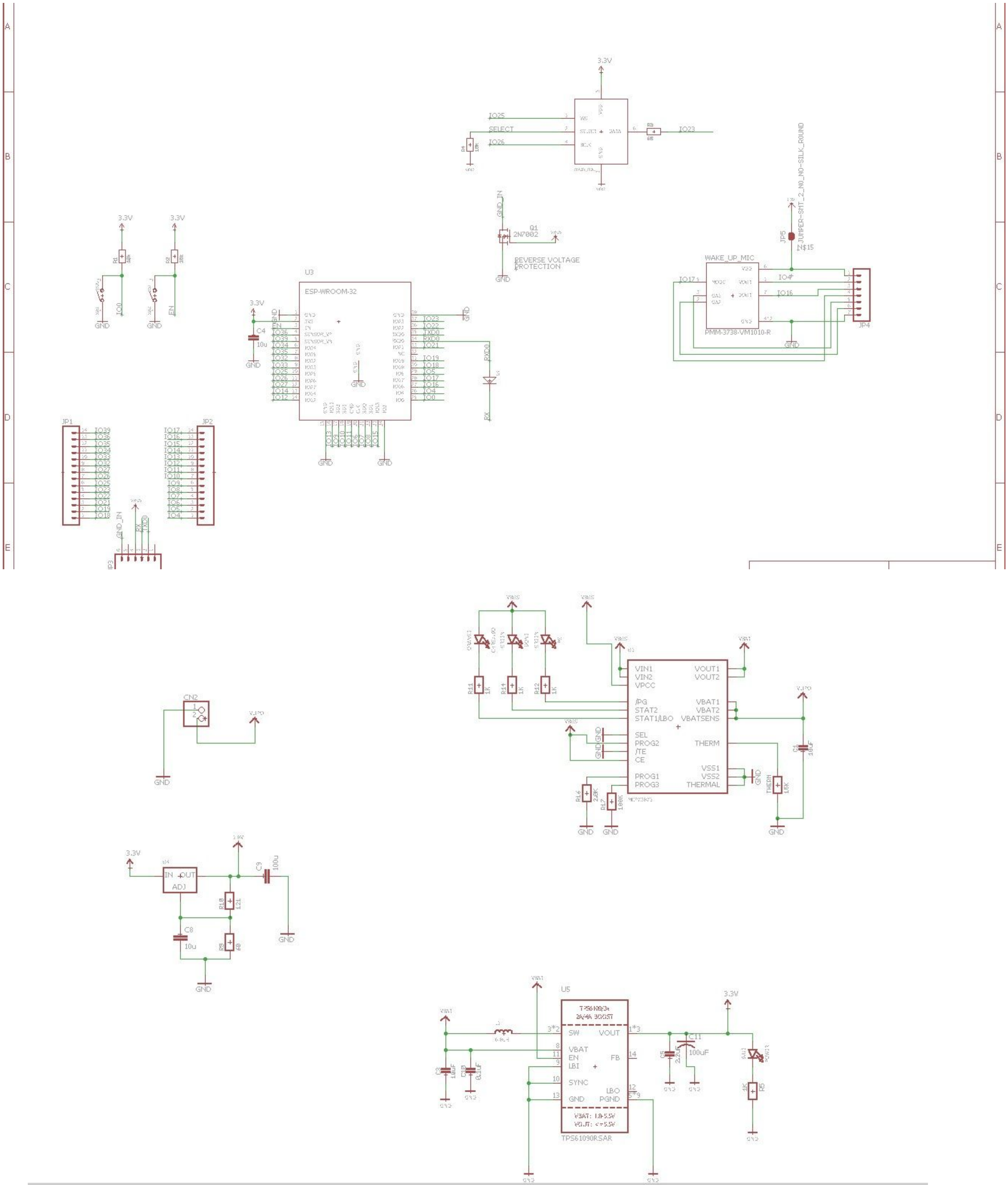
The board will be battery powered, but we also want to charge the battery on the board. Thus the inclusion of a battery charging circuit is necessary. To accomplish this, we are using a battery management IC chip, the Microchip MCP73871, which can power the device via a 5V line while charging the battery. Since the output of the MCP73871 is unregulated, a boost converter, TI TPS61091RSAR, is needed to stabilize the output at 3.3V, which is required by the ESP32 and main mic. We also included a 1.8V linear regulator, LM1117, to power the Vesper wake up mic.

Since our group has no experience in PCB design, we asked a post-doc we know for some tips. He was very helpful and gave us some great pointers about assembly and layout requirements. I was also able to find some reference schematics on adafruit to help me layout the design.

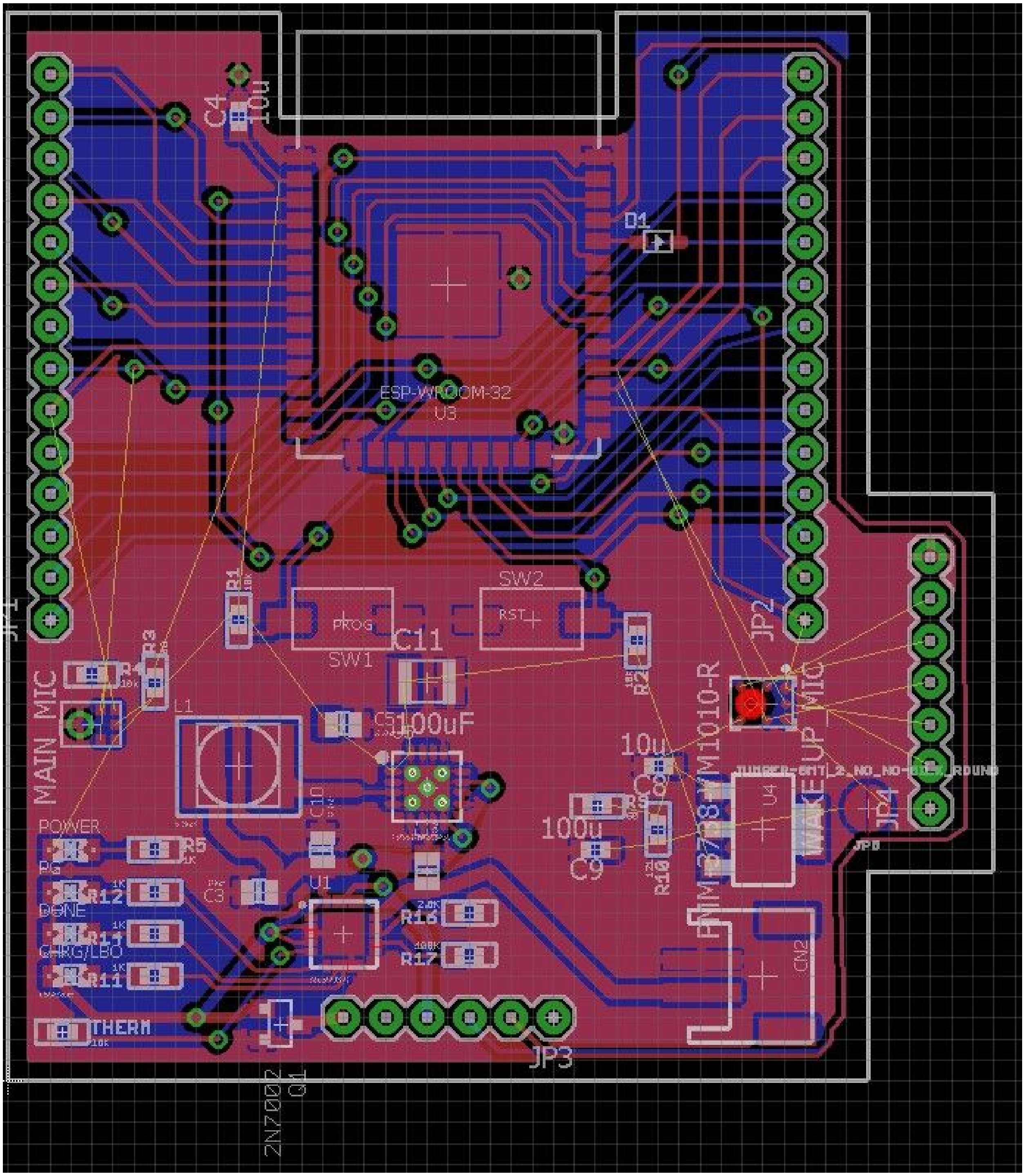
I am behind on the signal processing side of the project. I was not able to collect data using a MEMS microphone since we lacked the hardware required. I decided to wait to collect data with the ADC on the ESP32, which is being bread-boarded. I instead focused on PCB design while Ronit worked on the bringup of the microcontroller.

Next week I plan on completing the schematics and the routing of the PCB. Our goal is to have it completed by Wednesday 2/20/19. I will also begin to collect data with the breakout SPH0645 mic we bought from Adafruit. James has started some of the keystroke detection using his cellphone mic. I will aid him with this and by the end of the week we should have a very accurate keystroke detector for hunt and peck style of typing. I assume further work will have to be done for touch typing, as keystrokes are closer together and usually softer.

Below I have the schematics and layout for our first revision of PCB.



A
B
C
D
E
3.3V
IO25
SELECT
IO26
IO23
GND_IN
Q1
2N7002
REVERSE VOLTAGE
PROTECTION
GND
JUMPER-SMT_2_NO_SILK_ROUND
JP5
WAKE_UP_MIC
IO17
IO4
IO16
PMM-3738-VM1010-R
JP4
U3
ESP-WROOM-32
EN
C4
10u
IO36
IO39
IO34
IO35
IO32
IO33
IO27
IO14
IO12
IO22
TXD0
RXD0
IO21
IO19
IO18
IO5
IO0
IO2
IO15
IO13
RX
IO0
EN
JP1
JP2
JP3
CN2
R11
R14
R12
1K
VIN1
VIN2
VPCC
/PG
STAT2
STAT1/LBO
SEL
PROG2
/TE
CE
PROG1
PROG3
VOUT1
VOUT2
VBAT1
VBAT2
VBATSENS
THERM
VSS1
VSS2
THERMAL
10K
C8
10u
C9
100u
R18
IN
OUT
ADJ
U5
SW
VBAT
EN
LBI
SYNC
GND
VOUT
FB
LBO
PGND
C11
100uF
TPS61090RSAR



ESP-WROOM-32
U3
C4
10u
D1
JP1
JP2
JP3
SW1
SW2
PROG
RST
C11
100uF
R1
R2
R3
R4
MAIN_MIC
WAKE_UP_MIC
L1
C5
C10
U1
U4
C3
C9
100u
10u
R5
R9
R10
R11
R12
R14
R16
R17
POWER
PG
DONE
CHRG/LBO
THERM
CN2
Q1
2N7002