5.2V 5.2V

GND GND

2x2 GPIO header pad grid

TODO: use DC-DC converter for 3.3V rail
Use TI IC TLV62569DBV

Do not input external 3.3V or you'd compete with the regulator

Can use PCB antenna or '-U' clip-on antenna version, same pad footprint

5.2V — C13 100uF OPT — GND

5.2V — C7 100uF — GND

5.2V — C6 1uF — GND

5.2V — 1 IN — 2 GND — 3 EN — GND

3.3V — 5 OUT — C3 100uF — GND

4 BYP — C5 1uF — GND

SPX3819

ESP32-S3 can't handle power supply which can supply at least 500mA or more

Can add electrolyic caps using 5.2V and GND pads, and 3.3V and GND pads

3.3V — GND — C2 22uF — C1 100nF — GND

EN — C14 1uF — GND

IO number is the GPIO pin number in the datasheet so IO15 = GPIO15

Custom SPI pins limits SPI speed to 80MHz (shouldn't be an issue as for example the SX1262 is max 16MHz (min 62.5ns SCK period) (SX1262_datasheet.pdf, Table 8-1: SPI Timing Requirements, t3))
https://docs.espressif.com/projects/esp-idf/en/latest/esp32s3/api-reference/peripherals/spi_master.html#gpio-matrix-and-io-mux
Also Meshtastic uses 4MHz

1	GND	GND	41
2	3V3	GND	40
3	EN	IO1	39 — STATUS
4	IO4	IO2	38 — NOTIFY
5	IO5	TXD0	37 — UART0_TX
6	IO6	RXD0	36 — UART0_RX
7	IO7	IO42	35 — DIO1
8	IO15	IO41	34 — BUSY
9	IO16	IO40	33 — NRST
10	IO17	IO39	32 — MISO
11	IO18	IO38	31 — MOSI
12	IO8	IO37	30
13	IO19	IO36	29
14	IO20	IO35	28
		IO0	27 — BOOT

ESP32-S3-WROOM-1-N4

GPS_EN, UART1_TX, UART1_RX, I2C_SCL, I2C_SDA, USB_D-, USB_D+

Actual SPI pins are:
NSS (CE), MOSI (ID),
SCK (CLK), MISO (IQ)

35,36,37 are NC on ESP32-S3-WROOM-2 so NC for compatibility

45 is strapping pin. Pulled low, but only has an effect on ESP32-S3-WROOM-1 as unlike -2 the eFuses are not set forcing SPI voltage to 1.8V, so it can be used on the -2 but for compatibility it will be NC (to not interfere with its internal pull down) as it controls the voltage on -1 if [permanent] eFuse is not set.

15	IO3	IO47	24
16	IO46	IO48	25
17	IO9	IO45	26
18	IO10		
19	IO11		
20	IO12		
21	IO13		
22	IO14		
23	IO21		

TXEN_OPT, RXEN, BUZZER, IO12, IO13, NSS, SCK

47, 48 are 1.8V on ESP32-S3-WROOM-2 (due to ESP32-S3R8V) so NC for compatibility even though it's 3.3V on -1

When GPIO0 (BOOT) is low, if 46 is high it's an invalid combination so NC it to be safe (3.3.1 Chip Boot Mode Control) also 46 INTERNALLY PULLED DOWN

set eFuse correctly and don't use GPIO3 for controlling mode (3.3.4 JTAG Signal Source Control) (should be fine by default as the 3 eFuses are 0 so GPIO3 is ignored)

Table 8-1. BOOT is internally pulled up but in testing the value is too high, approx. 2.5 MOhms so we need to pull it up some more

STATUS — LED_STATUS1 — R3 470Ω — GND

NOTIFY — LED_NOTIFY1 — R4 470Ω — GND

Calculated that this resistor value results in <7.5mA flowing through each LED

Likely used for Serial

3.3V — CN_UART0 JST 1mm

UART0_TX — R10 — UART0_RX

470Ω

A 499ohm resistor is inside the wroom module (for U0TXD) for the 80MHz harmonic filtering but can also be used to protect the IO pins? so only need on RX? Is this even necessary?

GND

CN_UART1 JST 1mm — 3.3V_UART1

UART uses same VCC GND pinout as I2C so if you plug I2C device in serial port there plugged-in device is not damaged

Likely used for GPS

UART1_TX

UART1_RX

GND

Add 470 Ohm series resistors for protection?

3.3V — R19 100kΩ — Q2 AO3415A — Q3 AO3416A — GPS_EN — R18 100kΩ — GND

3.3V_UART1

SOT-23-3P or SOT-23-3L, ensure you have the correct footprint

Pinout should match

Header pad for each extra pin, NOTIFY, and STATUS

H_EXT_GPIO1

STATUS	1	1
NOTIFY	2	2
IO4	3	3
IO5	4	4
IO6	5	5
IO7	6	6
GPS_EN	7	7
BUZZER	8	8
IO13	9	9
IO12	10	10
3V3		3V3
GND		GND

GND

CN_I2C_1 JST 1mm — 3.3V — I2C_SDA — I2C_SCL — GND

I2C_SDA — R11 1kΩ — 3.3V

I2C_SCL — R12 1kΩ — 3.3V

I2C 1k pullup resistor value is an educated estimate

CN_I2C_2 JST 1mm — 3.3V — I2C_SDA — I2C_SCL — GND

3.3V — VDD — RESET — GND — MAX809T

EN — R1 10kΩ — PS_RESET1 — C10 100nF — GND

MAX809T used to ensure the MCU is always turned on / reset when power is connected/reconnected

3.3V — BOOT — PS_BOOT1 — R2 10kΩ — C11 100nF — GND

Leave enough space under buttons to comfortably press with fingers from below in the orientation the board is held

ANT — RF_OUT1 — RP-SMA/SMA — GND

RP-SMA is probably better

5.2V — C12 100uF — C9 22uF — C8 100nF — GND

Best to put respective decoupling capacitors right next to power pads

	GND	GND	
	VCC	DIO1	DIO1
	VCC	BUSY	BUSY
DIO2	DIO2	NRST	NRST
TXEN	TXEN	MISO	MISO
RXEN	RXEN	MOSI	MOSI
	GND	SCK	SCK
	GND	NSS	NSS
	GND	GND	
	GND	ANT	ANT
	GND	GND	

E22-900M30S

3.3V — R9 100kΩ — NSS

ANT

Remove ANT solder pad and add a probhited region around it if using stamp hole instead of SMA to connect antenna

TODO?: Add ESD protection for USB?

TYPEC-304YSSW-ACP16 USB1

5.2V

A1B12	GND	GND	13
A4B9	VBUS	GND	14
A5	CC1		
A6	DP1		
A7	DN1		
A8	SBU1		
B1A12	GND		
B4A9	VBUS		
B5	CC2		
B6	DP2		
B7	DN2	GND	15
B8	SBU2	GND	16

CC1 — USB_D+ — USB_D-

CC2 — USB_D+ — USB_D-

GND

CC1 — R13 5.1kΩ — GND

CC2 — R14 5.1kΩ — GND

DIO2 — R7 0Ω — TXEN

TXEN_OPT — R8 0Ω DNM

Choose one or the other!
See ALL available options in variant.h

BUZZER1 MLT-5020 — Flyback diode (don't burn transistor!) — D1 1N5817WS

3.3V

Q1 S8050 — R15 470Ω — BUZZER

R16 10kΩ — GND

Test resistor values and update schematic