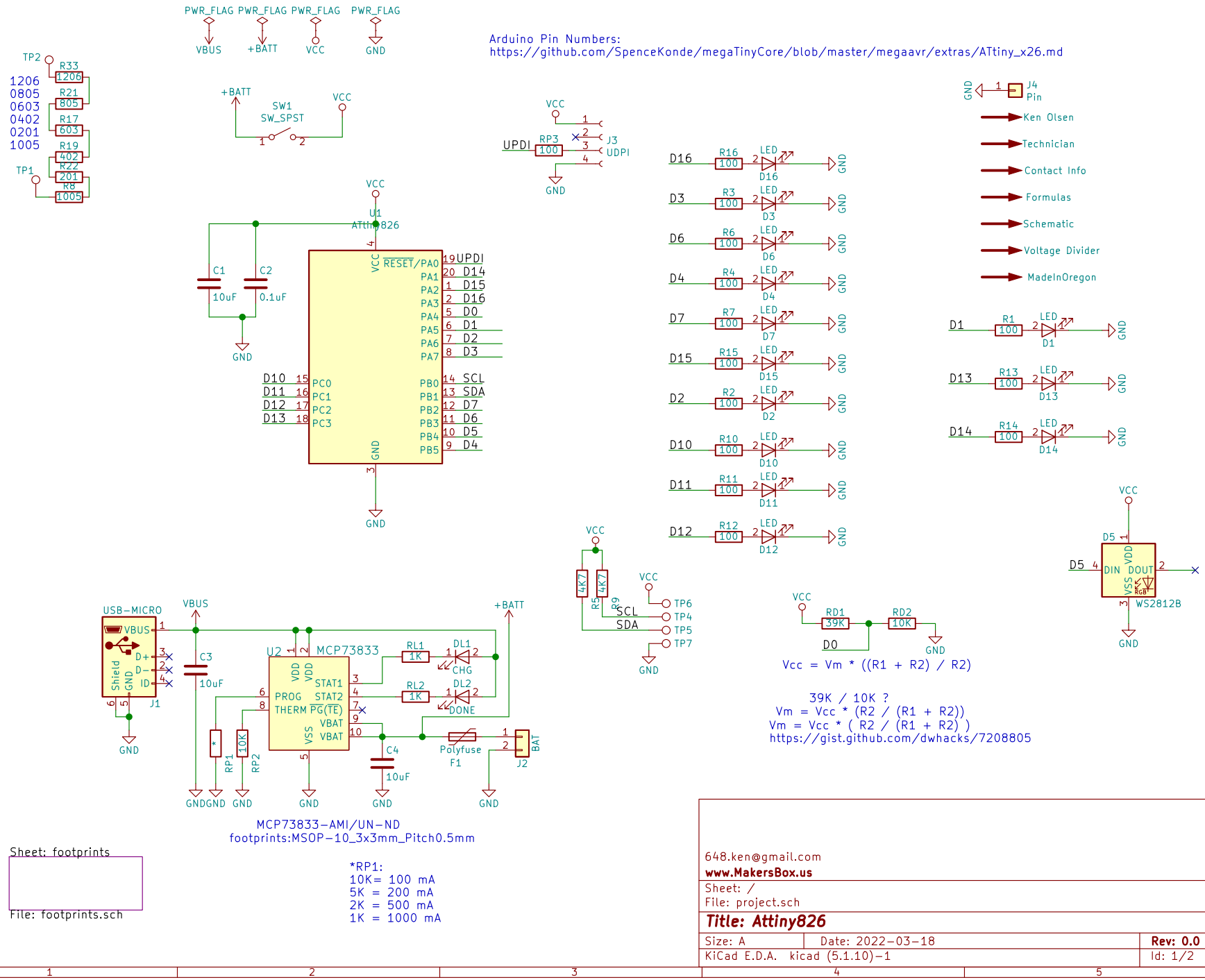
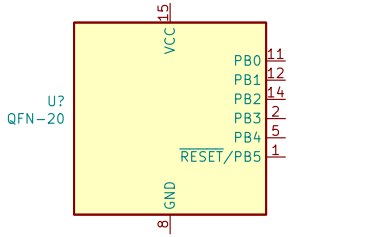
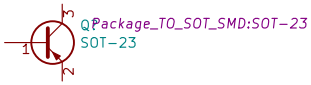
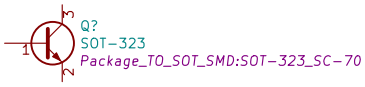
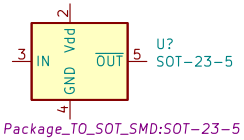

PWR_FLAG PWR_FLAG PWR_FLAG PWR_FLAG
VBUS +BATT VCC GND
Arduino Pin Numbers:
https://github.com/SpenceKonde/megaTinyCore/blob/master/megaavr/extras/ATtiny_x26.md
TP2
TP1
R33 1206
R21 805
R17 603
R19 402
R22 201
R8 1005
1206
0805
0603
0402
0201
1005
+BATT
SW1
SW_SPST
VCC
U1
ATtiny826
C1 10uF
C2 0.1uF
GND
VCC
RESET/PA0 19 UPDI
PA1 20 D14
PA2 1 D15
PA3 2 D16
PA4 5 D0
PA5 6 D1
PA6 7 D2
PA7 8 D3
PB0 14 SCL
PB1 13 SDA
PB2 12 D7
PB3 11 D6
PB4 10 D5
PB5 9 D4
D10 15 PC0
D11 16 PC1
D12 17 PC2
D13 18 PC3
GND
VCC
UPDI
RP3 100
J3
UDPI
GND
D16 R16 100 LED D16 GND
D3 R3 100 LED D3 GND
D6 R6 100 LED D6 GND
D4 R4 100 LED D4 GND
D7 R7 100 LED D7 GND
D15 R15 100 LED D15 GND
D2 R2 100 LED D2 GND
D10 R10 100 LED D10 GND
D11 R11 100 LED D11 GND
D12 R12 100 LED D12 GND
GND
J4
Pin
Ken Olsen
Technician
Contact Info
Formulas
Schematic
Voltage Divider
MadeInOregon
D1 R1 100 LED D1 GND
D13 R13 100 LED D13 GND
D14 R14 100 LED D14 GND
VCC
D5
DIN DOUT
VDD VSS
RGB
WS2812B
GND
VCC
4K7 R5
4K7 R9
SCL
SDA
TP6
TP4
TP5
TP7
GND
VCC
RD1 39K
RD2 10K
D0
GND
Vcc = Vm * ((R1 + R2) / R2)
39K / 10K ?
Vm = Vcc * (R2 / (R1 + R2))
Vm = Vcc * (R2 / (R1 + R2))
https://gist.github.com/dwhacks/7208805
USB-MICRO
VBUS
D+
D-
ID
Shield
GND
J1
VBUS
C3 10uF
U2
MCP73833
VDD VDD
PROG
THERM
STAT1
STAT2
PG(TE)
VBAT VBAT
VSS
RL1 1K
RL2 1K
DL1 CHG
DL2 DONE
+BATT
C4 10uF
Polyfuse
F1
J2
BAT
RP1 *
RP2 10K
GND GND GND GND GND GND
MCP73833-AMI/UN-ND
footprints:MSOP-10_3x3mm_Pitch0.5mm
*RP1:
10K= 100 mA
5K = 200 mA
2K = 500 mA
1K = 1000 mA
Sheet: footprints
File: footprints.sch
648.ken@gmail.com
www.MakersBox.us
Sheet: /
File: project.sch
Title: Attiny826
Size: A
Date: 2022-03-18
Rev: 0.0
KiCad E.D.A. kicad (5.1.10)-1
Id: 1/2



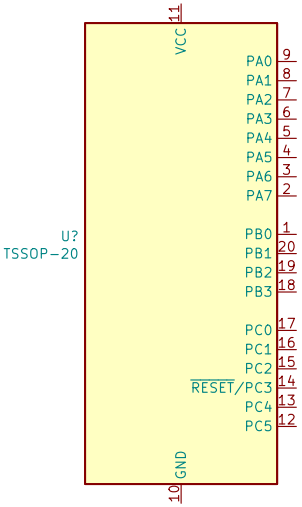
Package_DFN_QFN:QFN-20-1EP_4x4mm_P0.5mm_EP2.6x2.6mm



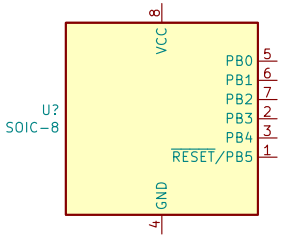




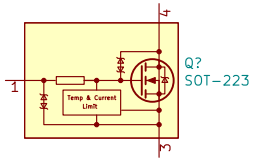
Package_TO_SOT_SMD:SOT-23-5



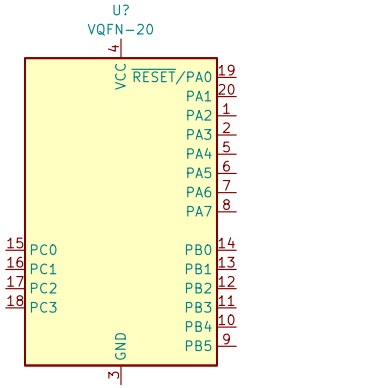
Package_SO:TSSOP-20_4.4x6.5mm_P0.65mm



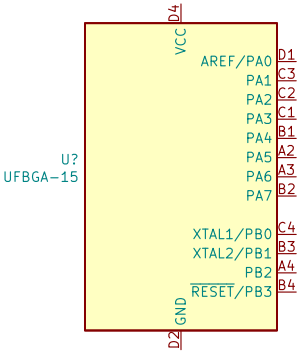
Package_SO:SOIC-8_3.9x4.9mm_P1.27mm



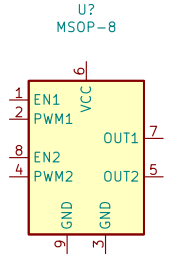
Package_TO_SOT_SMD:SOT-223



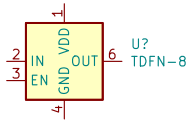
Package_DFN_QFN:VQFN-20-1EP_3x3mm_P0.4mm_EP1.7x1.7mm



Package_BGA:UFBGA-15_3.0x3.0mm_Layout4x4_P0.65mm



Package_SO:MSOP-8-1EP_3x3mm_P0.65mm_EP1.95x2.15mm



Package_DFN_QFN:TDFN-8-1EP_3x2mm_P0.5mm_EP1.3x1.4mm

Sheet: /footprints/ File: footprints.sch		
Title:		
Size: A4	Date:	**Rev:**
KiCad E.D.A. kicad (5.1.10)-1		Id: 2/2