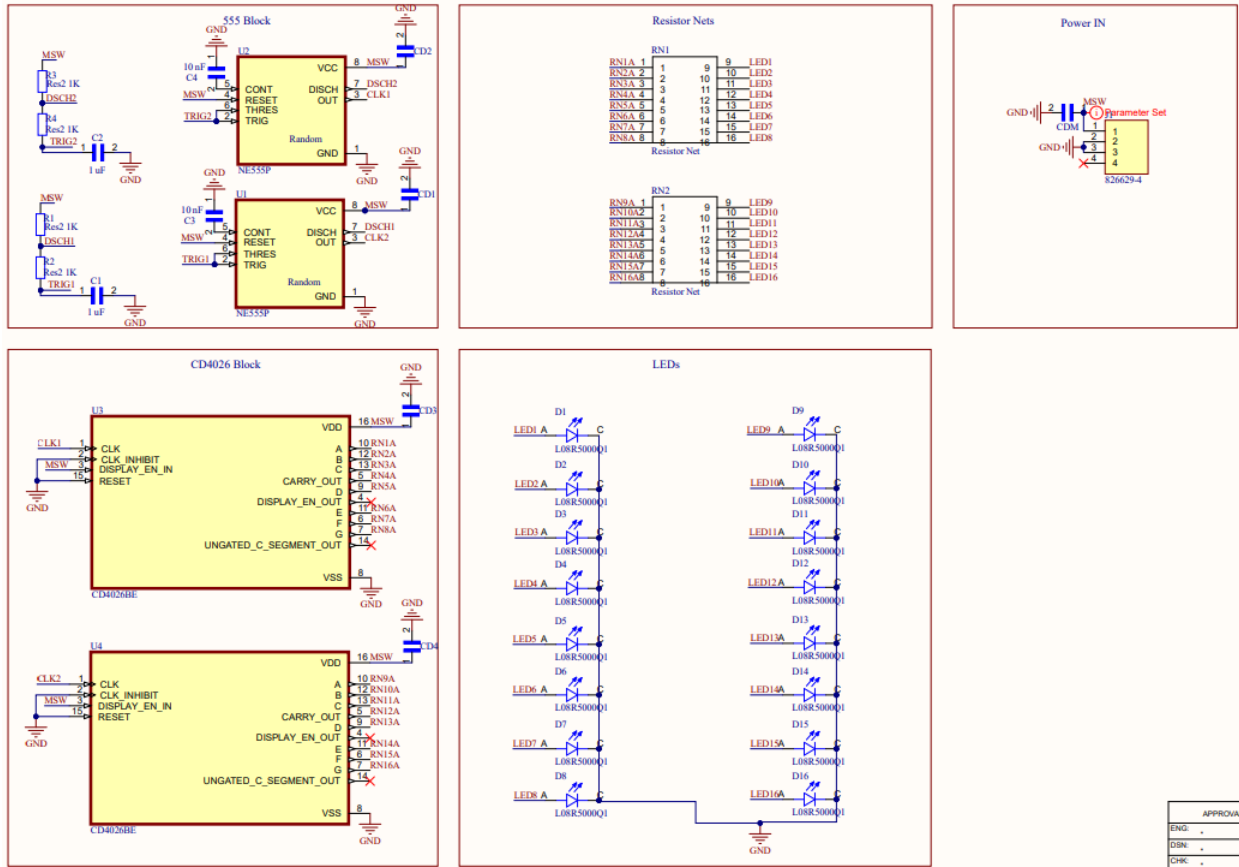# WOPR for IBM 5150/60

Schematic and PCB details

Circuit Schematic:

The schematic again uses hierarchical design although all the components fit on a single page. Different modules (555, CD4026 etc.) occupies individual blocks with interconnections done through net names to avoid debugging confusion.

The two 555 timers will provide two different clock signals to the two **CD4026s**, which will generate two different counting speed. The 555 resistor values show in the schematic is a placeholder, actual resistor values will be changed. The CD4026s will then directly drive the 16 LEDs (8 each) through two DIP resistor nets to create a random blinker effect. The resistor value is 500 ohms, this is to limit the voltage and current going through the LEDs to **20mA @ 2Volts** (output voltage from the CD4026s will be at approximately 11-12 volts).

The board power will be provided by the IBM 5160's 12V rail. A 4-pin header is included for this purpose. A cable will be constructed with 4pos Molex-D plug on one end and Dupont headers on the other. The master switch will be in series with the 12Volts line and mounted externally on the enclosure's front panel to turn the unit on/off.

LEDs are orange LEDs rated for **200 millicandelas**. It'll be neither too bright nor too dim.

PCB Design
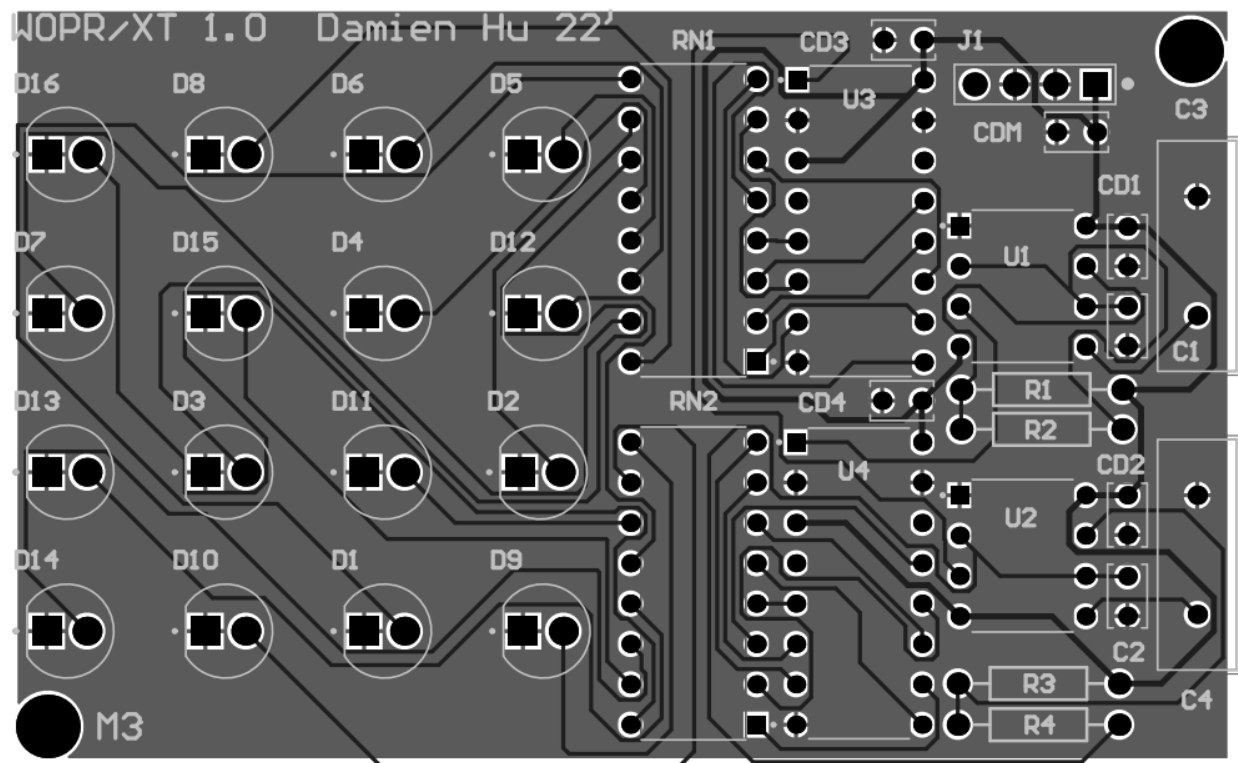
The PCB size is 78.9mm x 50.1mm (3.106 x 1.972 inches). The PCB can be divided approximately down the centerline into two parts – the LED quadrant and logic quadrant.

The LEDs are separated exact 1 cm from each other. Different pad shapes are included to distinguish positive from negative as the LEDs will be mounted on the bottom side of the PCB.

Both the CD4026 and resistor nets are packaged in DIP-16 to minimize clutter. All the ICs are locally decoupled with a net decoupler near the input of J1.

The grey area on the PCB is the GND plane. It occupies the entire back side of the board with all traces running on the top layer. Main power traces are thickened to 15 mils, should be more than sufficient for the rated power consumption of the completed PCB.

Two M3 mounting holes are provided if the 3D printed enclosure design passes QC and becomes ready for a laser cut acrylic version.

Design Rule Check

*Rules are only checked against JLCPCB's manufacturing constraints.

Altium Designer

Design Rule Verification Report

Date:	9/5/2022		
Time:	2:14:31 PM	**Warnings:**	0
Elapsed Time:	00:00:01	**Rule Violations:**	0
Filename:	C:\Users\hudai\iCloudDrive\Fall 22\ECE 1895\WOPR-for-IBM-PC\Altium\WOPR PC\PCB.PcbDoc		

Summary

Warnings	Count
Total	**0**

No rule violations are found.

Altium generated Bill of Materials

Comment	Description	Designator	Footprint	LibRef	Quantity
FG18X7R1E105KRT00	COMMERCIAL GRADE, HALOGEN FREE DIPPED RADIAL / BULK	C1, C2	CAPRB250W50L400T250H550	FG18X7R1E105KRT00	2
CK45-E3DD103ZYGNA	Cap Ceramic Single 0.01uF 2000V E -20% to 80% (14.5 X 5mm) Radial Disc 7.5mm 105Â°C Bag	C3, C4	CAPRB750W60L1450T500H1850	CK45-E3DD103ZYGNA	2
FG18X7R1H473KNT06	COMMERCIAL GRADE, HALOGEN FREE DIPPED RADIAL / AMMO PACK	CD1, CD2, CD3, CD4, CDM	CAPRB250W50L400T250H550	FG18X7R1H473KNT06	5
L08R5000Q1	LED, Orange, Through Hole, T-1 3/4 (5mm), 30 mA, 2.1 V, 635 nm	D1, D2, D3, D4, D5, D6, D7, D8, D9, D10, D11, D12, D13, D14, D15, D16	LEDRD254W57D500H1070	L08R5000Q1	16
826629-4	Conn Unshrouded Header HDR 4 POS 2.54mm Solder ST Thru-Hole Automotive Carton	J1	TE_826629-4	826629-4	1
Res2	Resistor	R1, R2, R3, R4	AXIAL-0.4	Res2	4
Resistor Net	Resistor net, 510Ohm, 2%, 2.25W. 16Pin DIP Thru Hole. 8 Resistors.	RN1, RN2	DIP794W45P254L1969H508Q16	4116R-1-511LF	2
NE555P		U1, U2	DIP794W45P254L959H508Q8	NE555P	2
CD4026BE	CMOS Decade Counter/Divider with Decoded 7-Segment Display Outputs and Display Enable 16-PDIP -55 to 125	U3, U4	DIP794W45P254L1969H508Q16	CD4026BE	2

Note: As there are no available footprints for the DIP-16 resistor net, I created a custom symbol and coupled it with a generic DIP-16 footprint. The library file can be found in the Altium folder of the repository.

GitHub: https://github.com/Damien130/WOPR-for-IBM-PC.git