Embedded Systems Project Report

Adam Cotter - 20079973

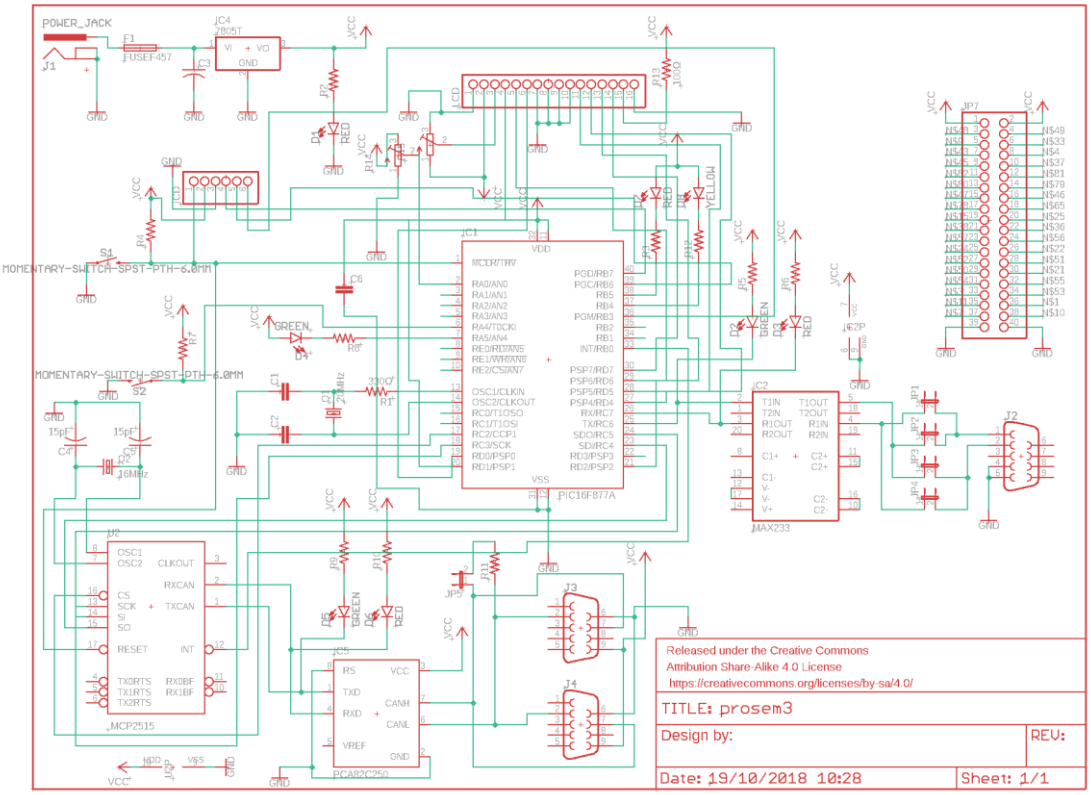
Bachelor of Science in The Internet of Things
Semester 3
Waterford Institute of Technology

1) Introduction:

This report will cover the process undertook to design and manufacture a printed circuit board using EAGLE to design the schematic and board layout, having an external manufacturer create the base boards and install the components ourselves.

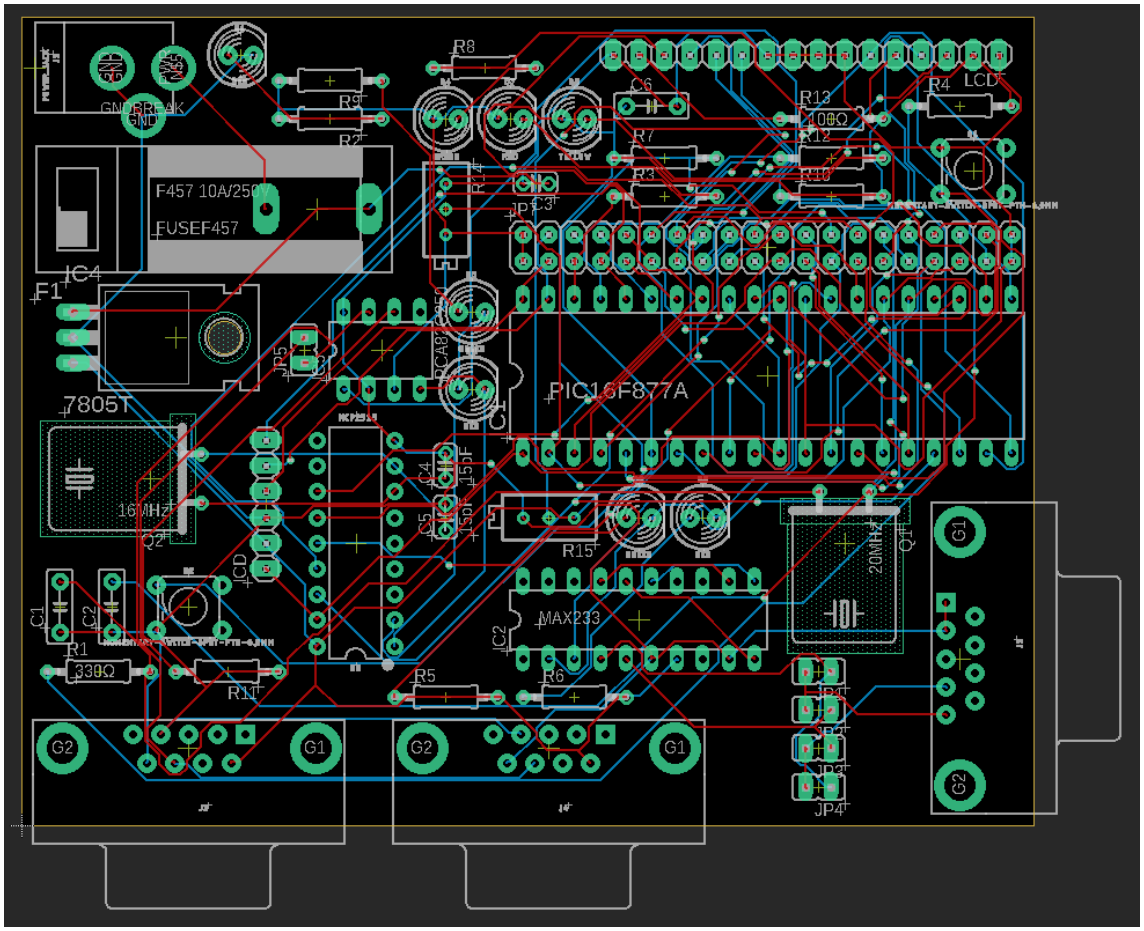
2) PCB design and manufacturing:

The schematic was first designed using EAGLE. We were given a list of components and circuits needed to create an operational board and were left to create the design ourselves.



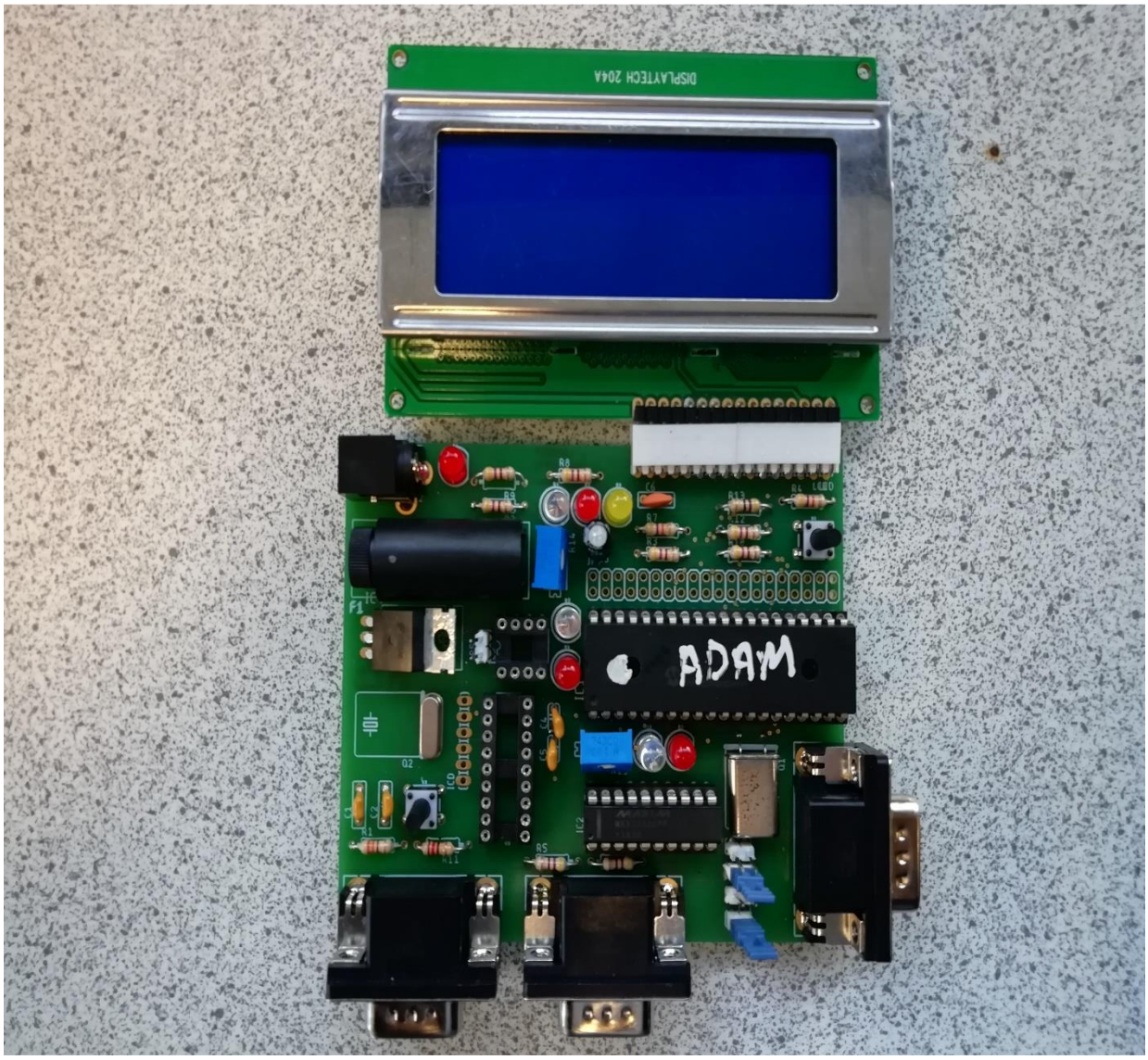
(Above: Schematic Design)

After creating the schematic, a connected board layout was created in EAGLE. The components were placed in an appropriate manner. The board was then routed to connect each pin to their respective connections. The board file was then exported into the relevant layer files and sent to the manufacturer for creation.



(Above: Board Layout)

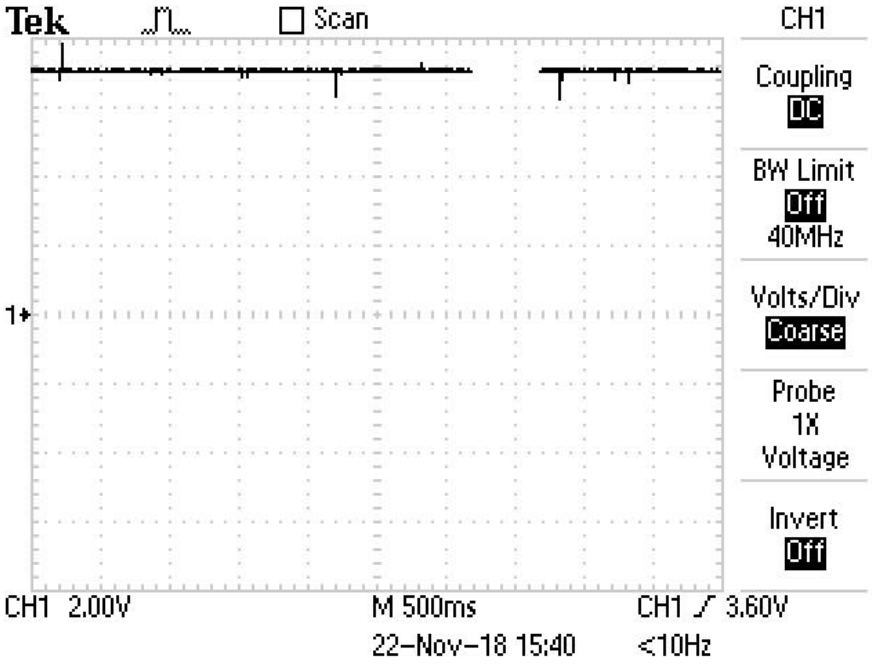
After the base boards were received we soldered each component to the board manually and attached the processors. We then installed the custom software on to the boards and checked if the inputs worked.



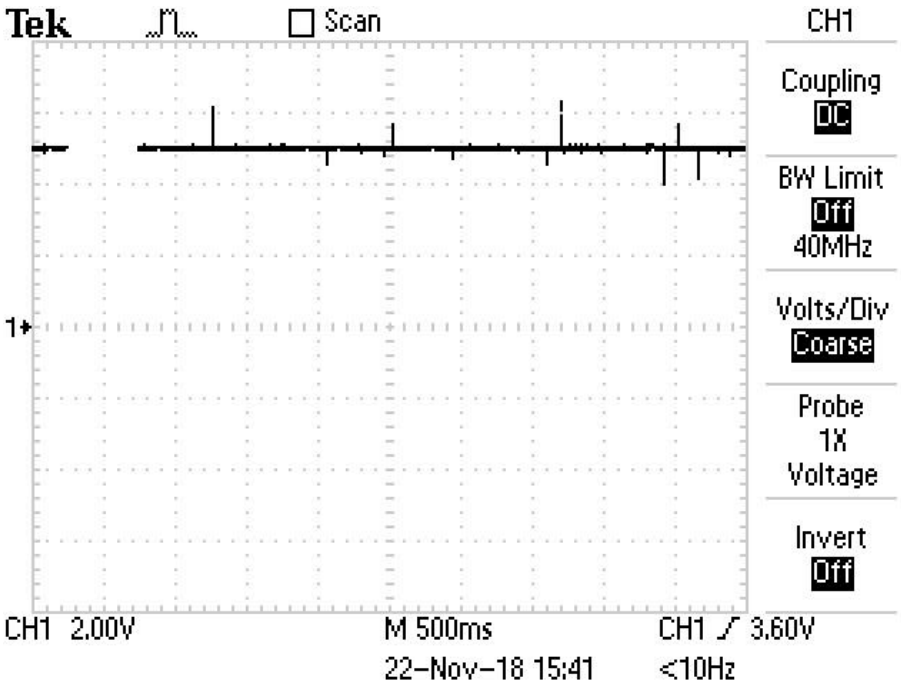
(Above: Finished Board)

3) Hardware Testing:

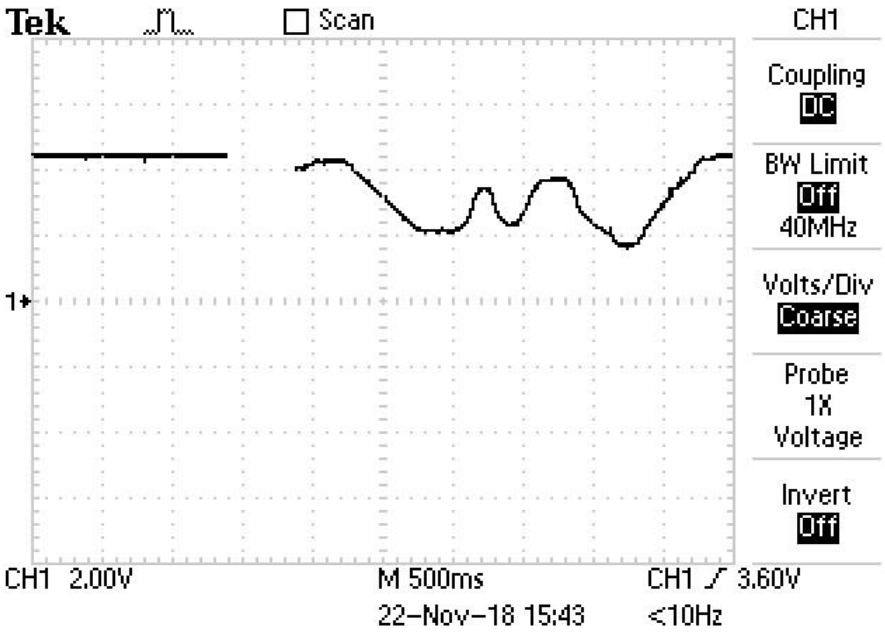
When the board was finished, it's connections were tested to check for faults. Below are screenshots from the oscilloscope used to test the board.



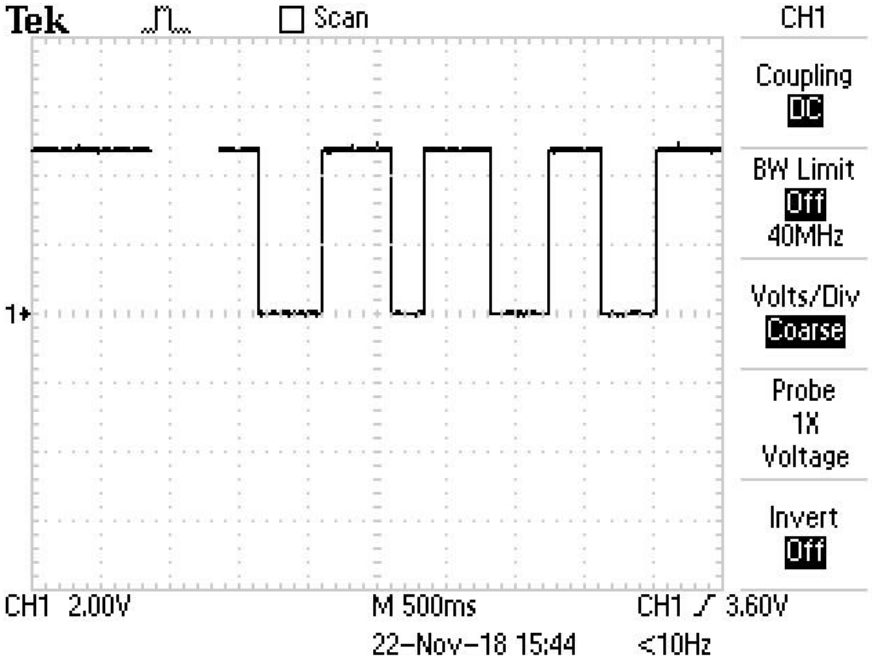
7V Input



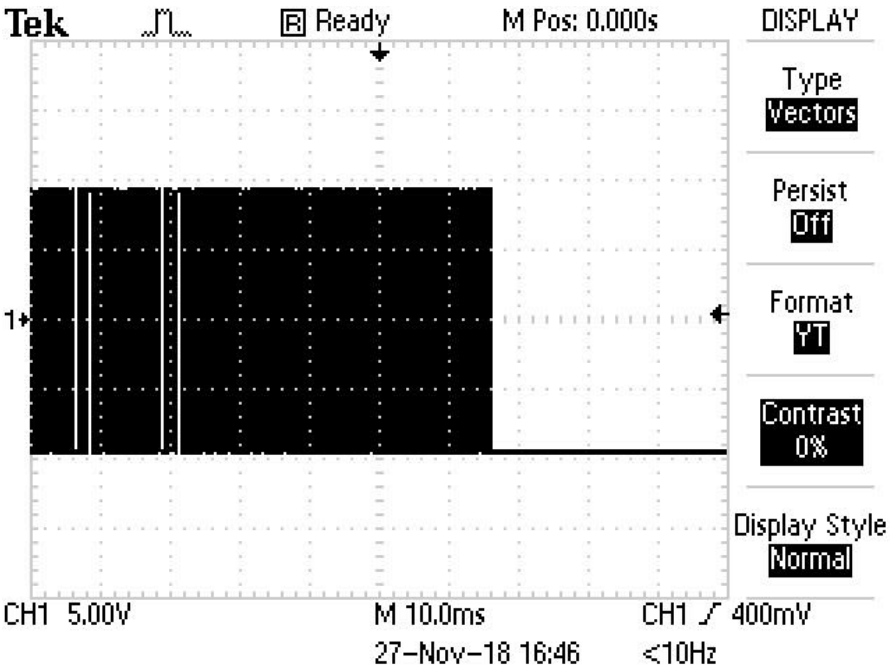
5V Input



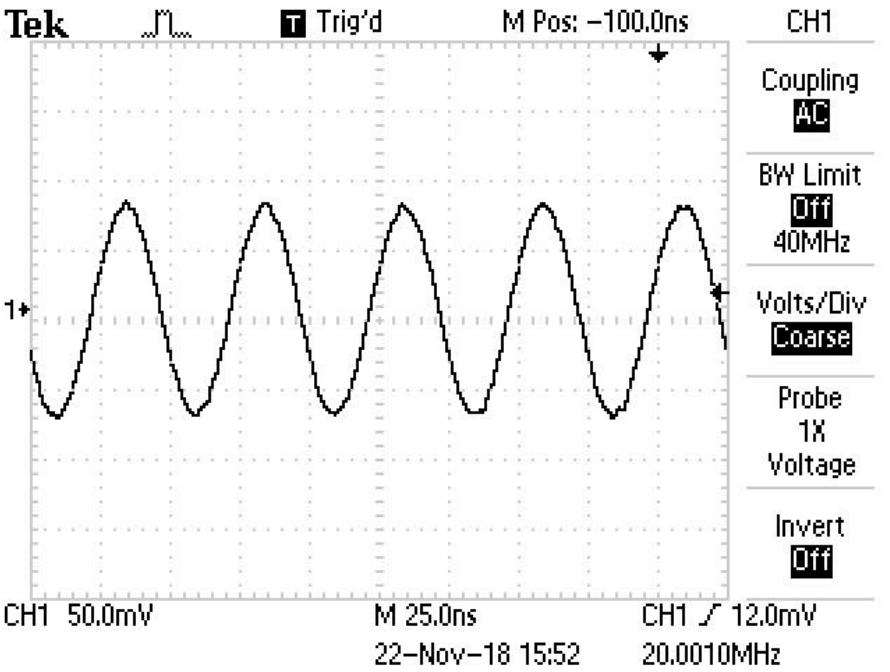
Analogue Input



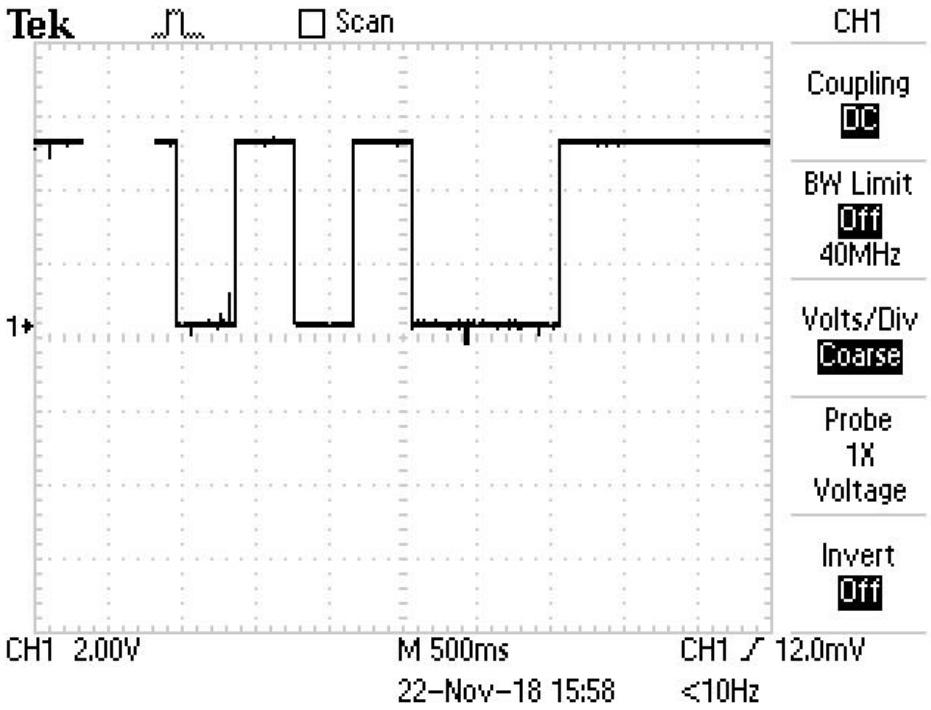
Digital Input



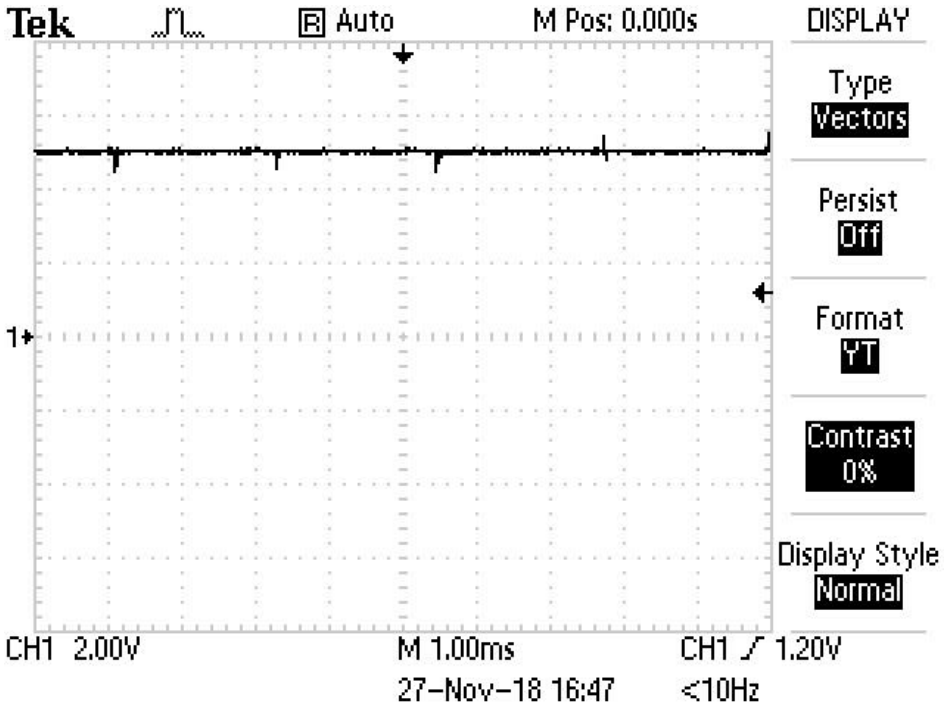
LCD Pin 4



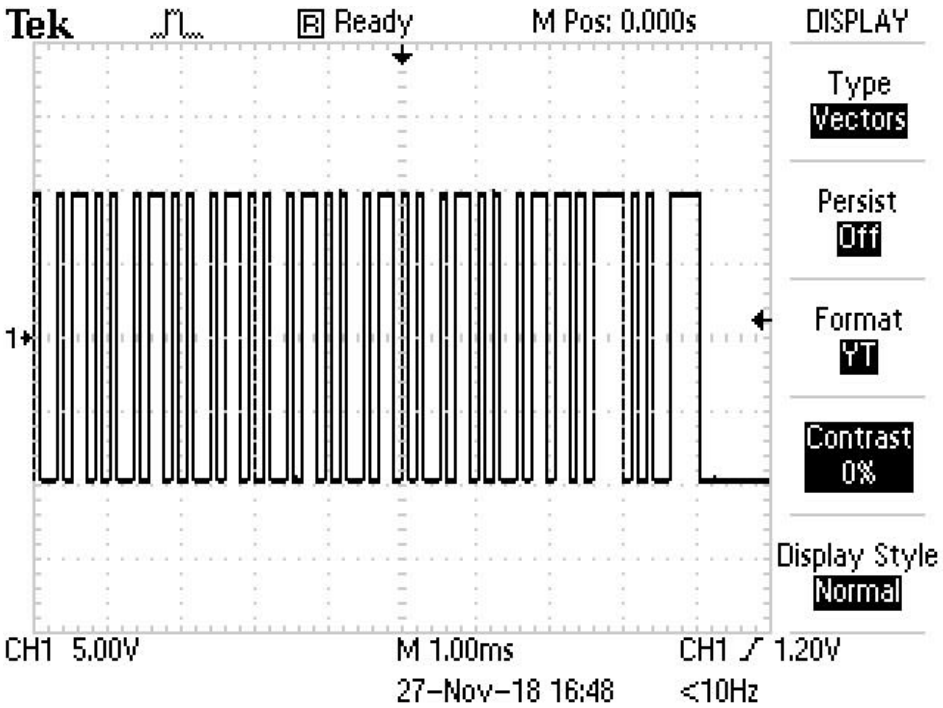
Clock



LCD Pin 11



Data Transmit


Tek
Ready
M Pos: 0.000s
DISPLAY
Type
Vectors
Persist
Off
Format
YT
Contrast
0%
Display Style
Normal
1
CH1 5.00V
M 1.00ms
CH1 ⁄ 1.20V
27-Nov-18 16:48
<10Hz

RS232