I would really be grateful if you start to build the Lightning Capture Device, that you go to the Photrio thread and say hi. Also please post photos of your completed project.
Please refer to the Photrio thread for further build help
Lightning Capture Device - Cheap - Simple & it works (photrio.com)

GitHub repository where all documentation & code can be found. billbill100 (github.com)

Please refer to the Photrio thread for further build help

ESP32 Hardware Build V1.1 30/07/2024

Building the Lightning Capture should be relatively easy. Below are some hints & advise on the build.

It can be built as one of two versions.
a) Fully featured with Oled display and adjustable values
b) minimal design, without Oled or controls.

Ensure the correct firmware for your chosen version is used.

Refer to the ESP32 Wiring Guide schematic documents, for connections. Note. Schematics show a bare board, rather than it sitting in the breakout board.

The breakout board has 38 terminals, the ESP board only has 32. Mount the ESP32 so the spare 3 sockets on either side are to the rear. Do not use the legends printed on the breakout board as they will not be correct for the 32 pin ESP32 board.

All of the boards can be purchased from eBay, Amazon or AliExpress, which is by far the cheapest. Details of the parts required are detailed in the Parts List document

All of the modules are pre-built and do not require component soldering, however, the header pins on some modules, including the main ESP32 board require the header pins to be soldered.

Using DuPont jumper wires, the boards can be connected together. You will need one pack of 20cm and one pack of 10cm Dupont male-to-female wires.

Additionally, there are more Red and Black wires used, so one pack of each, 20cm Red & Black wires can be purchased to keep the colour coding correct

There are many 3.3V (Red)& GND (Black) power connections. These will not all fit into the breakout board screw-terminals. It will be necessary to connect them in groups with a choc-block or similar, then to the screw terminal.

When fully built & tested, consider using a little hot-glue on the connectors, to stop them working loose.

Ensure you pay particular attention to the polarity of the connections. The boards are clearly marked.

The OLED is marked Vcc and is connected to 3.3V. (Red wire) and GND, (Black wire). Note some displays have the polarity reversed. Check very carefully the legends printed on the board.

The Light Sensor board is marked VCC (3.3V) and GND. DO (digital Out) is the white wire.

The Traffic Light board is marked with the colours and also Vcc and Gnd.

The Rotary encoder is marked. Showing Vcc and 0V at either end of the row of pins, with the others clearly marked.

All of the parts (other than buttons with rear nut) were hot-glued into the project box. This works well for the OLED. Wooden standoffs were cut to raise the breakout board above the buttons. Similarly, a wooden standoff was used for the opto-isolator. PCB standoff pillars can be used instead. Part description is in the parts document.

The photos are of the prototype and show a slightly different opto-isolator board. This required a resister to be added, to make it work at 3.3 volts. The opto-isolator board specified in the parts document also has to be modified by changing a resister, but being through-hole mounted, it is easier on this particular board.

The prototype has been tested with the Nikon Z8 and works perfectly using a standard 3.5mm to Nikon shutter release cable.
Refer to the camera connection guide for details of Nikon and other cameras.