This file: readme_for_laser_power_meter.pdf

Transconductance amplifier (refer to the **schematic**):
The schematic for the photodiode (**PIN** for P-type intrinsic N-type) amp is in the box in the lower left. The op-amp via the feedback resistors R2 & R3 will hold the inverting & non-inverting inputs at the same voltage (floating ground in this case). With zero volts across the photodiode its current output will be linear over many orders of magnitude (i.e. from a few tenths of a milliwatt on the high end to a few nanowatts on the low end). In this configuration the photodiode acts like a current source. The more light that hits it the more current flows from the cathode to the anode.

The op-amp is set up as a transconductance amplifier (converting the photodiode current to voltage at the op-amp output). R2 & R3 control the gain (i.e. the volts/amp at the op-amp output). A commercial power meter calibrated for the laser wavelength is used to adjust R3 such that the output voltage of the op-amp, in volts or millivolts, is the same as the power meter in watts or milliwatts. Note: The commercial power meter sensor is placed directly in front of the sample to be illuminated. A glass slide is used to pick off a few percent of the beam just before the sample and then it goes through a neutral density filter to nock the intensity down enough to stay in the linear range. In this case the max current through the gain resistors on the PIN is about 0.1mA.

Note: The **connectors for the PIN and the DVM are reversed**. The connector on the PIN should be rotated 180 degrees to allow for a straight through cable. Since it's hard to re-solder that connector I made a cable that has the connector on one end different from the other.

A brief explanation of how to **make a photodiode amp** using op-amps is posted at:
https://www.physics.unlv.edu/~bill/PHYS483/LED_PIN.pdf
For more info on making **amplifiers and filtering** using op-amps see Op-Amp basics at:
https://www.physics.unlv.edu/~bill/PHYS483/

Scaling the output:
In our case the max power of the laser, after going through all the optics, was 110mW. Since I was using a 4.5 digit **DVM** (digital volt meter) with a +/-2V range as the display, and can set which decimal points come on, I upped the gain so it would output 1.1V when the power meter read 110mW. The larger voltage on the output means any noise picked up between the PIN and the DVM will be a smaller percentage of the signal. Note: The DVM and low pass filters will knock down any noise picked up so that only low frequency noise would get through. Low frequency noise is much less likely to be picked up than high frequency noise. Note: A voltmeter was used to display the power readings until I finished assembling the DVM circuit. If one doesn't need to read low laser power with high resolution one could use a standard voltmeter as the display (but it would read a factor of 10 higher because of the gain of 10). The gain can always be changed so the decimal point is in the correct location (gain = 1, i.e. output would be 0.180V instead of 1.80V). I posted two schematics and board layouts (one using the decimal points 2, 3, & 4 for the different gain ranges, the other using decimal points 1, 2, & 3). The later is correct for our setup with the gain of 10 on the PIN.

Filtering the power supply:
Power for the op-amp is supplied from a 6V **switching supply** (tiny wall wart, like a phone charger). Switching supplies switch pulses of current on and off rapidly to keep the output at the expected voltage. That creates high frequency noise which we want to minimize. Where the 6V comes into the PIN amp there are two **decoupling capacitors** (C6 & C7) to help smooth the power supply ripples. Think of the caps as buckets of water that a hose squirts into on one end in pulses and water flows out the other end. There will still be ripples in the bucket and on the output but much less than the pulses coming in from the hose. Next comes two **ferrite beads** (tiny, high frequency inductors). **Inductors** make it hard for the current to change fast and will make the current ripples smaller. Then comes two Schottky diodes. A Schottky diode is a high speed, low forward voltage drop, one way valve for current. This rectifies the

ripple to smooth the voltage. I guess I only needed the bottom diode for the false ground (see Single supply operation below for explanation). Not sure why I put one on each rail. Next comes four more decoupling capacitors of different sizes (C1-C4). At this point the voltage is quite stable. It also helps that the circuit only draws a few milliamps. You may ask, why put a tiny capacitor (say 100 or 1000 times smaller) in parallel with a much larger capacitor. Here the water bucket analogy fails (adding a thimble to the side of a large bucket doesn't help). The reason is that smaller capacitors usually have lower inductance. Remember that inductance limits how fast the current can change. In this case it limits how fast the capacitor can charge when a ripple comes in. By putting smaller caps in parallel with larger caps the smaller ones can respond quickly to smooth high frequency noise while the larger caps will respond slower but can supply more current to smooth lower frequency noise.

For more info on **decoupling caps** see page two of "Things you should know about resistors, capacitors, and inductors" at:
https://www.physics.unlv.edu/~bill/PHYS483/RLC_info.pdf

Picking an op-amp:
The op-amp used is a **MAX44241AUK+T.** It was chosen for three reasons (size, low offset voltage, and single supply operation).

Size: It's large enough that I can solder it by hand but small enough that it won't take much space on the circuit board. Recall that the entire PIN amp has to fit inside a 1" circle (the lens tube).

Offset Voltage: It has very low input offset voltage and drift with temperature changes. Typically the input offset is 1uV at room temperature and this 1uV will only change by about 1nV/C as the temperature changes. Once you adjust the gain of the op-amp it should read the right voltage regardless of small temperature changes all the way down to tiny voltages. Note: This type of amplifier is called an auto-zero amplifier. It alternates between holding the output in place while disconnecting the inputs, then it shorts the inputs while measuring what the output would be, adjusting things to bring the output back to zero, and then reconnecting the input and outputs. As long as the signal you're measuring isn't near the sample frequency this works well. In our case the the output has a low pass filter that prevents the output from changing fast.

Single supply operation: The op-amp is what's called a single supply rail to rail output op-amp (SSRR). This means that the output can go almost to the power supply rails (In this case the power supply is 0V and 6V and the output can go from about 0.1V to 5.8V). The input shouldn't go within 1.5V of the 6V power supply rail (<4.5V) to maintain the extremely high common mode rejection ratio (**CMMR**). CMRR is how good the op-amp is at amplifying the difference between the input while not amplifying the common voltage on the inputs.

R1 & D2 provide a false ground about 0.3V above the 6V return. If there is no light hitting the photodiode the output should be zero but he op-amp output can't go below 0.1V. This would make it look like some light is still hitting the photodiode. This is where the false ground comes in. R1 is across the power supply and will draw about 0.6mA continuously. This current flows through D2. D2 has a forward voltage drop of about 0.3V at this current. By using the high side of the diode as the floating ground the op-amp is able to go down to 0.1V (0.2V below the false ground). This allows the output voltage to go all the way to 0V output when no light hits the photodiode. Note: There is no **dark current** to generate a voltage when no light is hitting the photodiode because there is no voltage drop across the diode.

For more info on how to **read an op-amp data sheet** see:
https://www.physics.unlv.edu/~bill/PHYS483/op_amp_datasheet.pdf

Display with X10 & X100 options:
The schematic for this part is everything other than the box in the lower left. Since the PIN output will be

linear over many orders of magnitude I added a X10 and X100 gain settings (X1 for <2V or <200mW, X10 for <200mV or <20mw, and X100 for <20mV or <2mW). Note the DVM has a positive voltage range of 0-1.9999V.

The power supply is filtered similarly to the description for the PIN circuit but with the addition of a 5V regulator to power the DVM. I used the same schottky diode to rase the ground pin of the 5V regulator about 0.3V above ground. This time there is 25-35mA going through the diode due to the larger current draw for the LED display on the DPM. The op-amps (a quad version of the one used with the photodiode amp but in a SOIC package) has the negative rail tied to the 6V return (same as the photodiode amp). Again, this allows the op-amp output to go slightly below the false ground so we don't need to worry about single supply issues not going all the way to zero. Note: Most voltage regulators need more headroom (i.e. an input voltage of 1.5-2V above the output voltage) to work properly. This voltage regulator is called a low dropout regulator because the voltage drop between the input and output can be quite low. With a 35mA output the dropout voltage is about 0.12V.

Four op-amps are used to buffer or amplify the output voltage from the photodiode amp and the floating ground. Everything is buffered so there is very little current draw across the wires. Each amplifier has a leakage/bias current of around 1nA. 1nA*0.5 ohms (the approximate resistance of each wire on the 1m ribbon cable between the PIN and DVM) = 0.5nV drop per wire due to the op-amp (negligible). R1 was added as a pull down so the DVM would get pulled to the false ground when the PIN was disconnected. This draws about 4uA with a 2V output from the PIN. 4uA * 0.5 ohms = 2uV drop on both the float ground and the output voltage. A 4uV difference out of 20mV (the most sensitive setting) would be off by 4 counts out of 20,000 or 0.02% error. On the less sensitive scale it would be less than one count of error and not show up on the display. R1 could be increased to 1M or 2M and it should remove virtually all the voltage drop across the wires.

Calibrating the X10 and X100 gain stages:
I used a stable voltage reference with low pass filters and voltage dividers to simulate the input from the PIN. I used a 5.5 digit benchtop DVM to accurately measure the input voltage going to the DVM. I put in a voltage just below the maximum for for each gain setting (say 180-190mV for the 200mV scale). I would then adjust the gain pot, R3 or R7, until it was exactly a gain of 10 or 100. Note: With a gain of one, the 4.5 digit voltmeter was accurate to the last digit without adjustment. The 100X was harder because I ordered the wrong size resistors and pots. The schematic and board layouts match how I should have done it. Values for R6 & R7 are different as noted on the schematic. Because of the larger value pot I wasn't able exactly get a gain of 100 (the resistance seemed to change in tiny steps instead of smoothly). The closest was always off by a few counts (about 0.01%).

100X addition of R5:
When first testing I had oscillations on all outputs (1X, 10X, and 100X - the higher the gain the worse the oscillation). I thought R5 would solve the problem and installed it. At the same time I noticed I had my input voltage for testing connected to the wrong ground. Connecting it correctly removed the oscillations on all stages. I left R5 on the board but if you build this try it with R5 = 0 ohms (or a low value resistor). It may work fine. If not, use a 100 ohm resistor.

Rotary switch:
The switch is a DP3T (double pole three throw, basically two switches moving together with three positions each). This allows the voltage to be switched between the three gain settings and the decimal point to be switched to the correct position for each gain setting.