Sapflow Measurement Heater Control Module

Specification and Design Document

Dave Hawthorne

Coweeta Hydrologic Laboratory

# Background

Sapflow rates in forest trees are determined by measuring temperature differences in the sapwood. This method requires probes be inserted into the trunk. There are several variations on the design but they all involve a temperature sensing function, either thermocouple or thermistor based, and a heater element. This heater may be running continuously or pulsed.

The power requirements for the heater can be a significant component of the experiment’s power budget. For the pulsed method around 10J of heat is required per measurement. For the continuous method each heater draws around 0.25W.

The heater element is constructed of resistor wire and is energized by applying a voltage to it.

Coweeta has around 100 trees being monitored like this. Some experiments have mains power running them, some rely on solar power and some require routine battery replacement.

Current practice at Coweeta is to simply switch the battery voltage across the heaters using a solid state relay. These cost around $70 each. There is no regulation of the voltage. For the continuous method we either use a commercial two channel DC to DC converter ($500) or four channel linear regulators. These were manufactured in-house over a decade ago and many have failed. The BOM cost for these devices is around $80 per channel, much of this due to the high-power integrated circuit the design relies on.

The continuous heat probes may only require 2V across the elements. Currently the practice is to “discard” 10V of the battery supplied 12V to achieve this. This means that for every watt usefully applied to the experiment, five or more watts are wasted heating up the equipment cabinet.

There is a better way of powering the heater probes. Fundamentally, rather than connecting the devices in parallel, we connect strings of them in series. This way those devices share a common current path rather than a common voltage. For example if we work with the example of 10ohm probes that require 2V

# Features

Voltage range: 6V to battery voltage (max 15V)

Current range: 0A to 20V

Power dissipation: 10W

4/4

We assume that the heater circuit is electrically isolated from the tree.

# Operation

Refer to the schematic for this section. The relatively simple circuit is logically divided into a number of sub-circuits, each described below.

## Reference Block

The **ref** node is a constant voltage that is adjustable – via RV1 – from 0V to 5V. The R1/R2 voltage divider keep the reference terminal of U1 at half the voltage of it’s cathode – or, as feedback ensures – the cathode is twice the reference. The TL431 tries to keep the reference at 2.5V, hence the cathode remains at 5V. RV1 divides this and C2 keeps that voltage steady. Should an installation require a digitally controlled voltage then all components of this sub-system can be removed except C2. A pulse width modulated signal can be fed in on J3.

## Load Voltage Measurement Block

This block outputs a signal equal to the voltage across the load divided by three. This signal is referenced to ground. The scaling is determined by the ratio of R9 to R7 and R10 to R8.

## Comparison Block

This operational amplifier module outputs the difference between the reference voltage and the scaled load voltage signal, amplified by 100 times. The feedback loop encompassing the Power, Load Voltage Measurement, Comparison and Gate Drive blocks strive to keep this difference as small as possible by driving the MOSFET harder when the load voltage is too low, and less hard for the reverse. The gain is throttled back to 100 rather than being the opamp’s default high gain too aid with stability. The gain is determine by the R4/R5 ratio.

## Gate Drive Block

It was found in simulation that feeding the output of the comparison stage directly to the MOSFET’s gate could lead to instability under some load conditions. This block introduces a little bit of feedback to dull any ringing following on-off transitions.

## Power Block

The MOSFET (Q2) is the heart of the circuit. Its gate terminal is fed whatever voltage is required to drive the MOSFET such that the load voltage meets the target value. If the target value exceeds the battery voltage then the FET is turned fully on – around 10mΩ resistance so that whatever voltage we can squeeze from the battery is applied across the load. The FET must disipate all the waste heat from the circuit. This is the product of the total load current and the difference between battery voltage and load voltage. The diodes in this block provide a modicum of lightning protection; refer to that section below.

## Indicator Block

When the circuit is enabled the light comes on. D2 and R13 also serve to ensure we always have at least tiny load in series with the MOSFET so that the “off” load voltage is small even for no load conditions. It just avoids confustion.

## Enable Block

This controls power to the reference and opamp modules. When the **enable** node is pulled high (above 1V) then Q3 turns on. Current flows through R17 and R16 pulling the **direct** node down from the battery voltage. This, in turn, switches Q4 on, pulling the **VCC** node up to **BATT+**. There are other ways of turning off the MOSFET feedback loop, such as pulling the **gate** node to ground, however the current draw of the voltage reference circuitry when active is around 1.5mA (18mW) and it was felt this was a tad high particularly if energy for the system had to be lugged to its forest location in the form of lead acid batteries.

## Lightning Protection

We appear to have a high attrition rate on the existing regulator designs, both the DynaMax switch mode regulators and the in-house built linear regulators have failed over the years. Corrosion does not seem to be a factor. One possible cause is transient voltages induced into the systems by nearby lightning strikes. The cables running to the regulators from the trees being measured act like antennas and - potentially – kilowatts of power can be dumped into those circuits in the space of a millisecond. Neither regulator possessed any circuitry to withstand such a strike. We can add around $1 worth of components to provide rudimentary resilience. This doesn’t provide any guarantee – a direct strike on a tree being probed would likely vaporize the contents of the logger box.

I’ll pick an arbitrary level of protection based on economics and board space. My understanding is that we are limited to use of transient voltage suppression (TVS) diodes to limit impact. Our system operates with low voltage and low impedance. Gas discharge tube (GDT) - or gas arrestors – act when the voltage across they exceeds a threshold (minimum design value is over 50V) whereby they arc dropping the voltage to ten volts or lower until there is no current source.

Once lightning causes the GDT to crossbar like this the battery current would maintain the condition until it burned out something wiring or connector or tracks on the PCB. So we would need to include a fuse on the battery line which would blow to protect the circuit from the protection. This could be a self reseting poly-fuse or a expendable fuse. The advantage of the latter is that we’d know when there had been a significant event. GDTs don’t act immediately, usually TVS diodes or varistors are included behind the GDT, separated by some impedance. Given we aim to minimize such impedance then a common-mode choke could sit between the GDT and the TVS diode to provide common-mode impedance without added (much) resistance to the loop.

Let’s aim for a system that can handle a 60A spike. The MOSFET can handle instantaneous current of 300A. We can fit a 50A rated TVS diode without too much pain on the board and it will cost around $0.60

The TVS diode (D4) runs from the battery line to ground. It will be a Littelfuse 1.5KE18A or equivalent ???. This is a unidirectional device that will prevent +BATT from rising more than 24V above or 1V below ground – or die trying. It can absorb a peak 1500W of power in a typical lightning strike pulse (a 10/1000us waveform). This corresponds to 60A across 19V. It will handle 200A when preventing the negative going pulse. D3 is across the load terminals – from LOAD to +BATT. It prevents LOAD exceeding +BATT by more than 1V, something that might also happen each time the current into an inductive load is switched off. These two diodes thus prevent LOAD exceeding 25V. When it is turned off the MOSFET (Q2) acts as a diode, if the voltage on its drain (the LOAD node) drops around 1V below that on its source (GND) it conducts. If the FET is turned on during the induced voltage then it will already be contacting. I will include pads for a third diode (D1) running from source to drain - should it be found that extra energy absorption is desirable. D4 is a big item 12mm long x 6mm diameter (a DO-201 package). D3 and D1, needing to handle less energy can be smaller DO-15 packages.

As with the traces between the connectors J1, J2 and Q2, we need to also keep the traces to the protection diodes as short as possible.

The MOSFET is the component most exposed to any induced current but we need to consider safety of other devices too. For this reason, we don’t want any possible induced current path other than around Q2. This precludes use of a ground plane that connects all four mounting holes together.

Should it be found advantageous, a GDT could be added to the installation, terminals connected to BATT+, GND and the load output. A common mode choke on the load lines between GDT and PCB could help shield the TVS diode from some of the spike prior to the gas arrestor crossbarring. A fuse would be needed on the battery line to the voltage regulator board.

Physical Notes

Screw terminals can be 5mm or 5.08mm (0.2”) pitch (The through-holes are oversize)

Mounting

We use 4-40 nuts and bolts for attaching the PCB and MOSFET to the mount plate

The plate is 3” by 3/16” 6061 aluminum flat bar, cut to 1.7”, 3.2” or 4.7” lengths for 1, 2 or 3 unit assemblies.

The design distance between PCB and mounting plate is 1/4”

For a 3/16” thick mounting plate and 1/16” thick PCB we need 3/4” bolts

# Construction

We can either place the MOSFET on the positive voltage side of the heater element or the negative side. If it sits on the positive side then, when not energized, the elements and all the wiring connected to them would sit at ground potential.

- All devices’ heat sink tabs are at ground potential, so the heat sink can be grounded too.

- all components are common (and therefore cheap and readily available)

- single sided PCB with through-hole components: easy to manufacture

nk tabs are at ground potential, so the heat sink can be grounded too.

- All components are commonly available from multiple manufactures (and therefore cheap and readily available from multiple sources for many years.)

We will use a custom designed printed circuit board (PCB), which we will load with components in-house. We would use through-hole (leaded) components rather that their cheaper and smaller surface mount equivalents to reduce the required skill level needed for loading and soldering. The bill of materials cost (BOM cost) will still be around $10 per channel.

Each PCB would be for a single channel. If multiple channels are required in an installation then boards can share a heat sink and power and ground lines can be soldered together – principally to reduce the number of wires running to the screw terminals. There would not be enough saving in BOM costs or power requirements to justify allowing for sharing common circuitry between channels.

This can be single sided (copper tracks on one face only). We would use a PCB prototyping service who can provide a week turn around time. They would be provided the design in electronic format. The economics of PCB manufacture is such that we could get 20 boards fabricated for around $40 (Double sided boards cost about the same). The simplicity of the design reduces the risk that the board will have a design flaw. The alternative of using a prototyping board (with a grid of unconnected pads) would substantially increase the time taken to load board with components. It also greatly increases the likelihood of mistakes during this process.

- single sided PCB with through-hole components: easy to manufacture

Housing

The control units will be housed in fiberglass equipment enclosures along with the data-loggers multiplexers and other circuitry. This boxes provide protection from the elements but their contents can still be exposed to humidity and insect/mice infestation. There is also the danger of exposure during servicing or due to enclosure being breached by bears and vandals. A bare circuit board would be potentially damaged in these scenarios. Exposed copper on the boards - such as component legs – is at risk of short circuiting with other items in the enclosure.

I would suggest that each board be “potted” with silicone, lacquer or epoxy, leaving just the screw terminals, the potentiometer screw and the MOSFET exposed. This is the technique used by Campbell Scientific with their equipment.

To facilitate ganging boards together to provide multiple channels (and using a common heat-sink). We might use a length of rectangular conduit to provide some mechanical protection to the electronics. This technique would contribute a few pennies to the BOM cost of each channel.

Potted the board removes the need for housing to be air-tight – a difficult proposition given the exposed terminals and transistor. The downside of potting is that it complicates diagnosis and repair of faulty units. Given the low cost, and simple design of these items we might choose to replace - rather than repair - them.

# 