Mittuniversitetet
MID SWEDEN UNIVERSITY

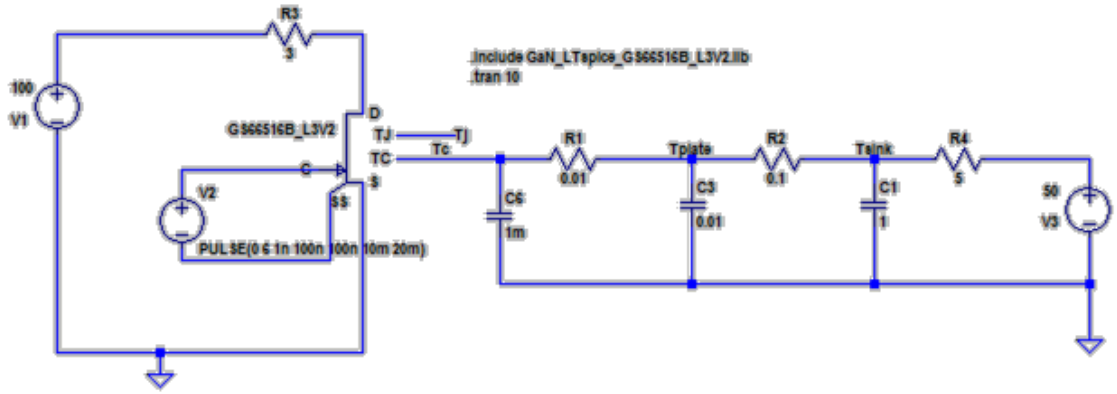
Assignment – III

LTspice is an easy to used but yet very powerful spice simulator mainly used for simulation or electronic circuits. The underlying equations used in electrical circuits have the same form as many other physical system such as magnetics, thermal, mechanical. This gives an opportunity to utilize the simulator to perform more complicated tasks such as coupled physical simulations in a very efficient way that cannot be achieved with full FEM simulations. In this assignment three different systems should be modelled. First electrical coupling from a switched DC regulator to a sensitive analog signal will be investigated and then two electro thermal systems.

TASK – I, Transient heating of a MOSFET under overcurrent conditions.

For accurate modelling of the heating of semiconductor devices both temperature dependent transistor model should be used but also a realistic thermal model off the transistor and thermal design.

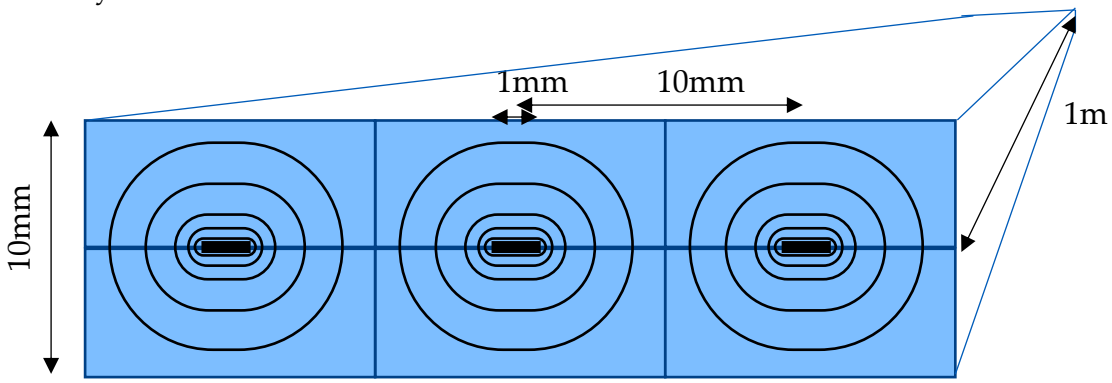
The task is to simulate the self heating of a transistor that is used in an inveter circuit turned on at the nominal grid frequency 50Hz. Set up a simulation using the transistor GS66516B (GaN systems) to switch at 50% duty cycle at 50Hz. The transistor should drive a load of 3 Ohm from 100V source. The transistor is connected to a heatsink with the characteristic as in the figure. The heat sink model is based in the a thin isolation material and a following heat sink.



- Determine how fast the circuit will take before over heating.
- What are the important factors in the thermal design.
- Calculate the dominant thermal time constant of the design.
- Discuss overall methods to ensure fail safe operation of an inverter like this.

TASK – II, Resistive heating application

In heating applications one challenge is that the resistance of an electrical wire is not constant but change with temperature during the heating. Additionally the thermal properties of the surrounding media will influence the behavior. The heating system is constructed by etched aluminum on a foil. The foil windings is arranged in three 1m sections connected in parallel. The sections is 10um thick and 1mm wide and separation between two tracks is 10mm. The two glass plates are each 5mm thick and the heater is driven by a 12V source.



- Preform all calculations in either Excel or Matlab so it easily can be changed for other parameters or mistakes in calculations.
- Calculate the resistance of the heating wire
- Design the system in LTspice with a voltage source and resistor.
- Set up a behavioral current source to emulate the power dissipation in the aluminum wire.
- Design a lumped thermal model
    - Calculate thermal resistance and Capacitances connected as a C-R-C-R-C… network for different distance from the aluminum wire.
    - The different segments with the following distance from the wire; 10um, 100um, 300um, 1mm, 2mm (segment thickness 10, 90, 200, 700, 1000um).
- Perform a thermal simulation assuming a constant reistance.
- Implement the temperature dependent resistance for the heating wire.
- Hand in a report showing voltages, currents, power and temperatures as function of time both with constant resistor and the temperature dependent resistor.

Material properties		
Thermal Conductivity	0,96	W/m/K
Heat capacity Glass	670	J/kg/K
Heat capacity Aluminum	897	J/kg/K
Density Glass	2600	kg/m3
Density Aluminum	2712	kg/m3
Convection heat transfer coefficient	20	W/m2/K
Conductivity of Aluminum	2,65e-8	Ohm m
Temperature coefficient of resistance	3.8e-3	1/K