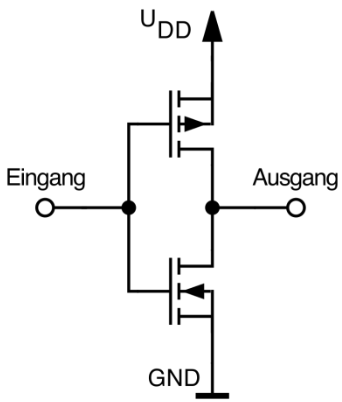
ngspice, development update and electrothermal simulation

Holger Vogt
University Duisburg-Essen
Duisburg, Germany

ngspice – what is it ?

Circuit simulator that numerically solves equations describing (electronic) circuits made of passive and active devices for (time varying) currents and voltages

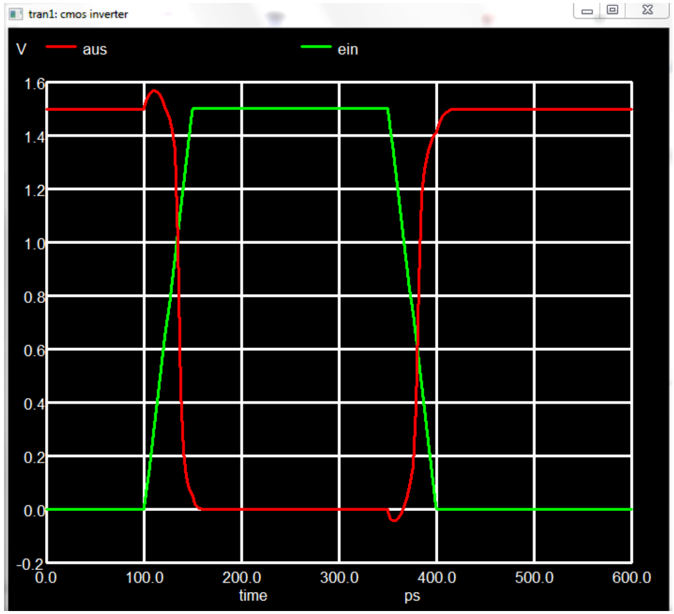
Open source successor of venerable spice3f5 from Berkeley



the circuit

```
CMOS inverter

.include ./bsim4soi/nmos4p0.mod
.include ./bsim4soi/pmos4p0.mod
.option TEMP=27C

Vpower VD 0 1.5
Vgnd VS 0 0

Vgate Ein VS PULSE(0 1.5 100p 50p 50p 200p 500p)

MN0 Aus Ein VS VS N1 W=10u L=0.18u
MP0 Aus Ein VD VS P1 W=20u L=0.18u

.tran 3p 600ps

.control
  run
  plot Ein Aus
.endc

.END
```

the input



the output

The two major application areas

PCB design support

Circuits are made with a mix of ICs and discrete components

Requirements:

Comfortable user interface (offered by third parties)

PSPICE and LTSPICE model compatibility

KiCad, Eagle, PartSim, CoolCAD, PSIM, WeSpice …

IC design support

Circuits are made of (MOS) transistors and (parasitic) passive components

Requirements:

BSIM 3, 4, (BULK) models etc.

Large circuit capability, speed

HSPICE PDK compatibility

gEDA, Yosys, efabless, Isotel, XSCHEM, PySpice …

Towards ngspice-32 (03/2020)

Improved graphics

Text orientation, fonts, color, linewidth selections

Monte Carlo

Improved statistical IC analysis

UNICODE support

utf-8 and wide-char for strings:

Text in plots, node names, file and directory names

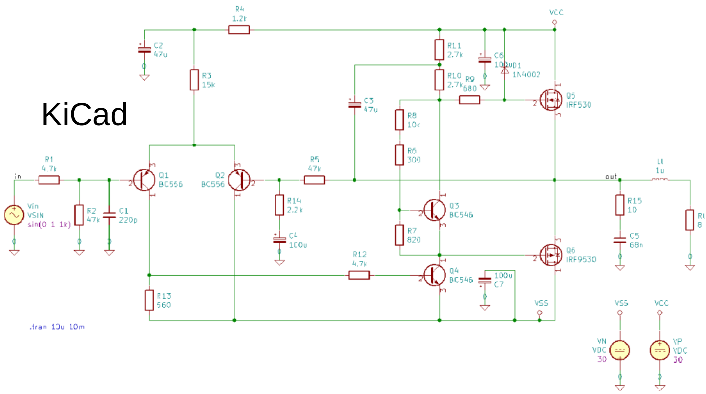
Revised VDMOS model

efficient model for power MOS transistors, now includes self heating (electrothermal model)

In addition: major code cleanup, enhanced robustness against buggy input, improved error handling, improved convergence

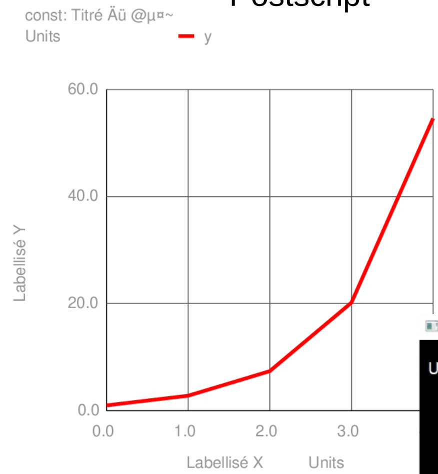
Some impressions

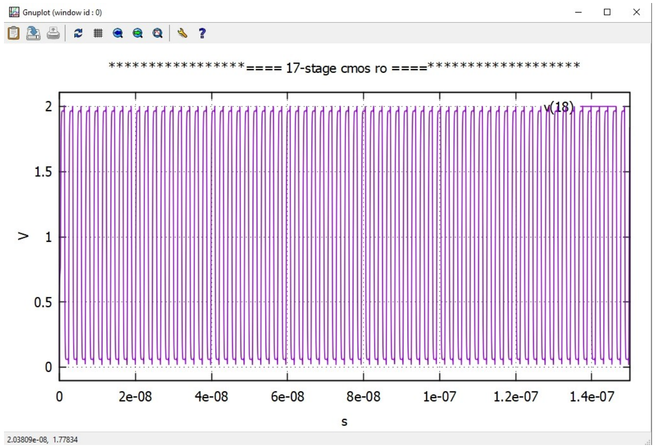
KiCad



Inputs

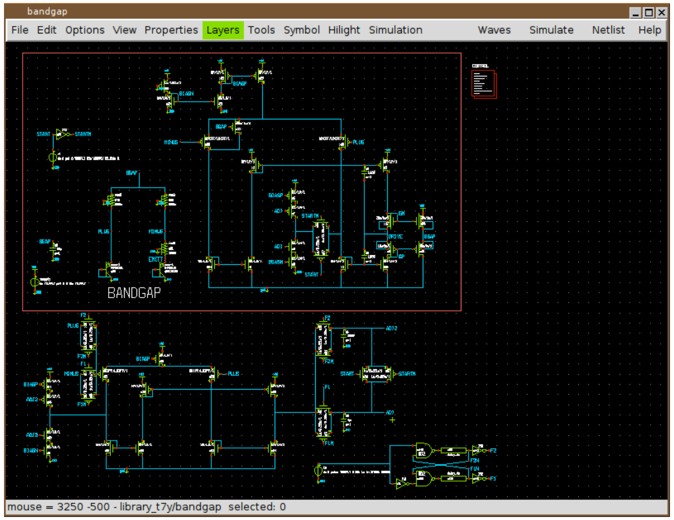
Postscript



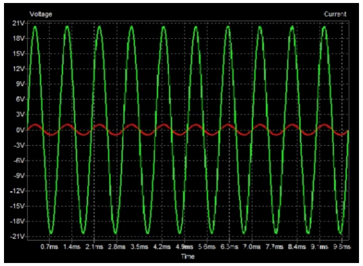


gnuplot

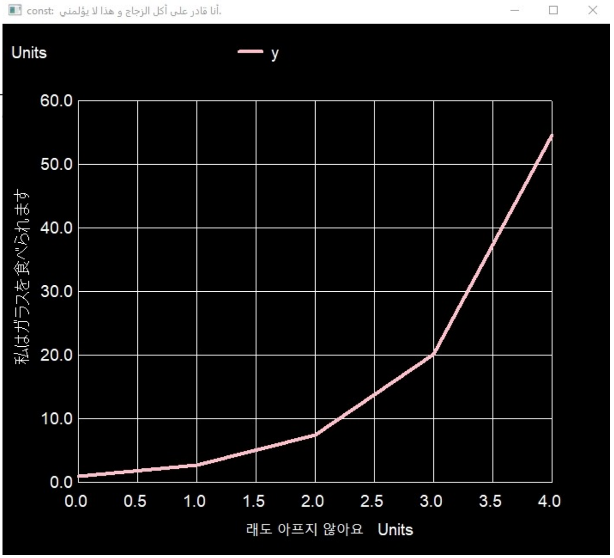
Outputs



XSchem

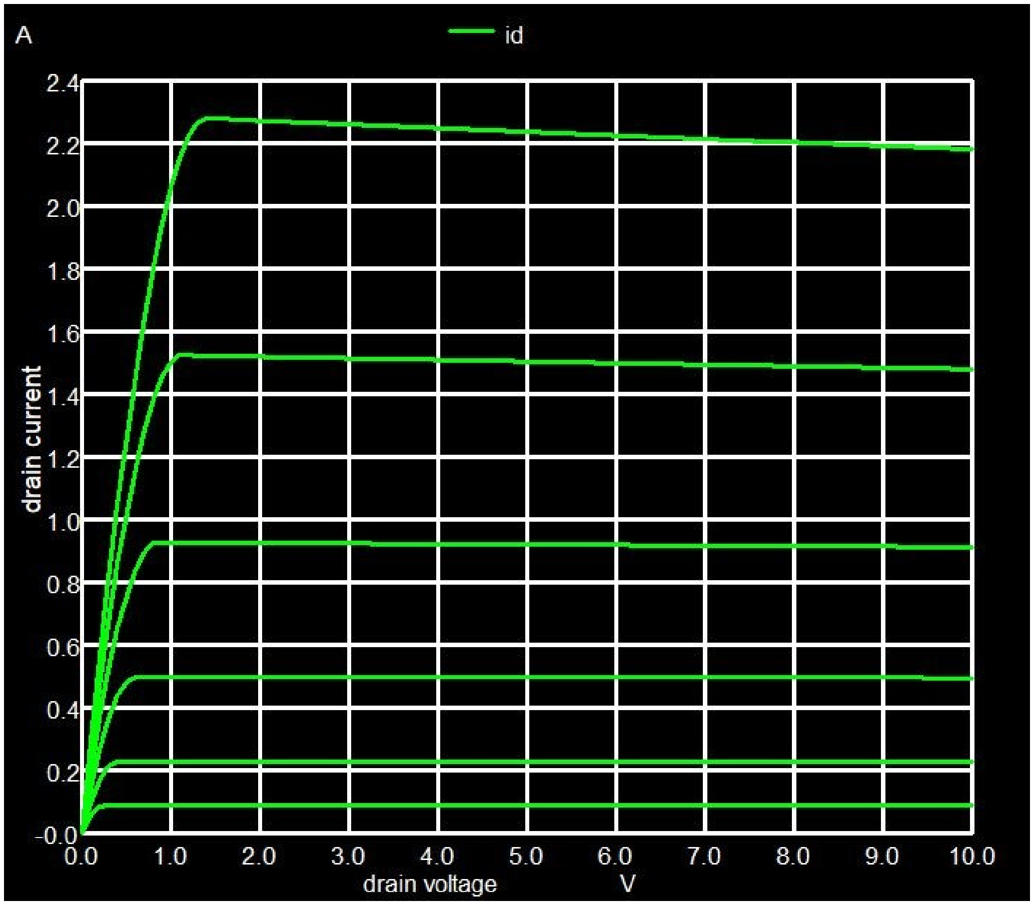


KiCad



UNICODE axis labels

Temperature influence on devices



Power MOS

Drain voltage is rising, current is decreasing?

=> Temperature effect!

Electric power => heat => temperature

Mobility is decreasing with temperature.

=> How to model this effect?

Electro-thermal modelling

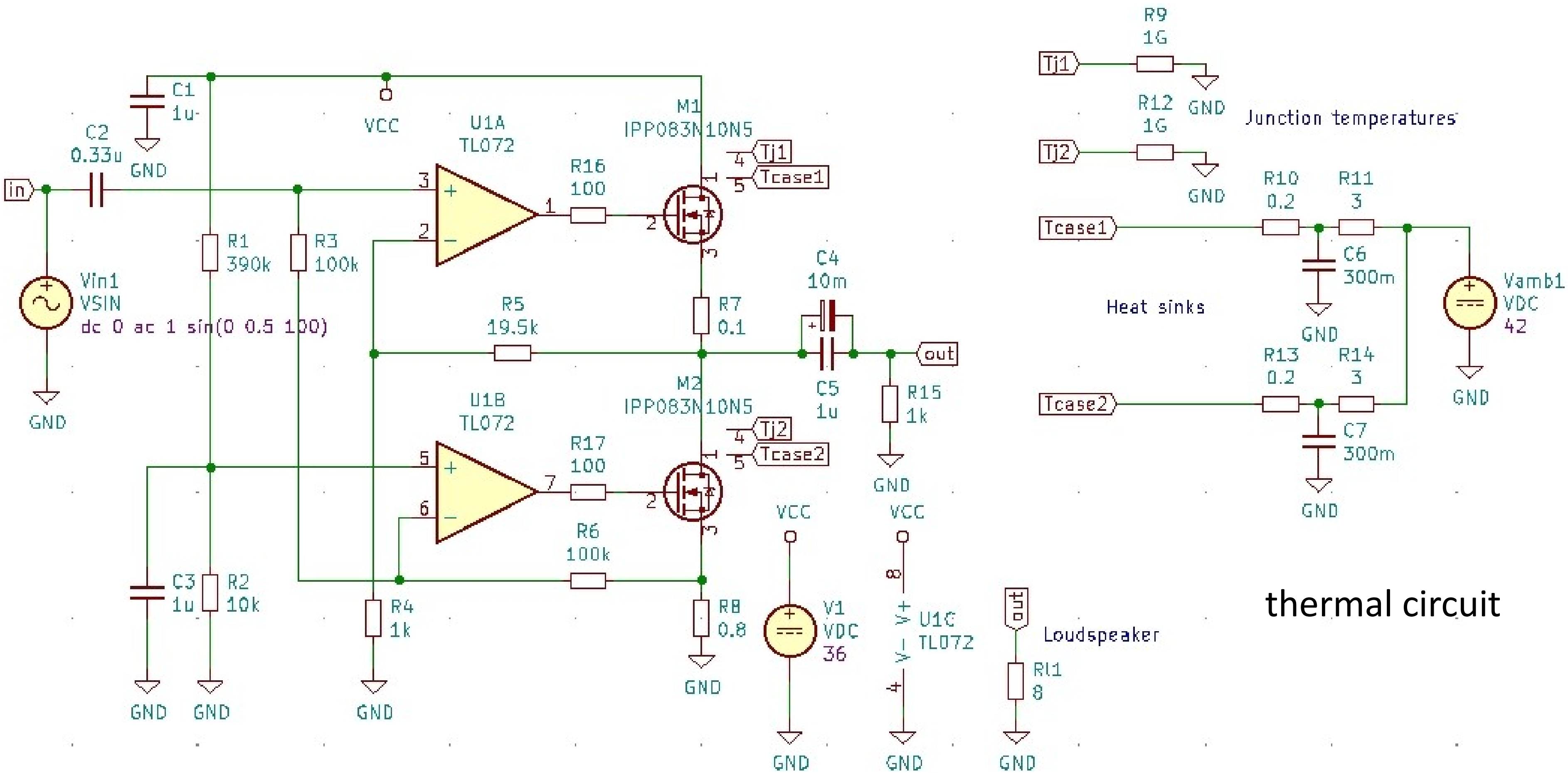
Make use of the equivalence of electronic and thermal properties (circuits).

Translate thermal to electrical circuits, run both circuit parts in ngspice. Electrical power dissipation generates heat, restricted heat flow rises temperature, temperature changes power disspation.

	Thermal term	Electrical term
Capacitance	*c* [J/K]	*C* [A/V]
Conductance	*g* [W/K]	*1/R* [1/Ω]
Temeprature/Voltage	*T(t)* [K]	*V(t)* [V]
Heat/Current	*Q* [W]	*I(t)* [A]
Time constant	*c/g* [s]	*RC* [s]

Example: Amp with power MOS

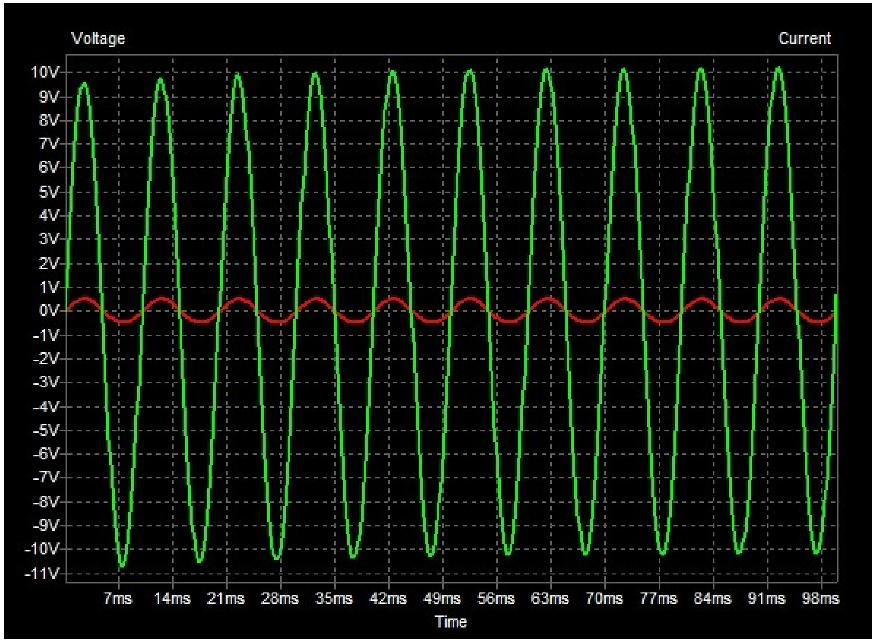
http://www.tubecad.com/2009/08/blog0168.htm



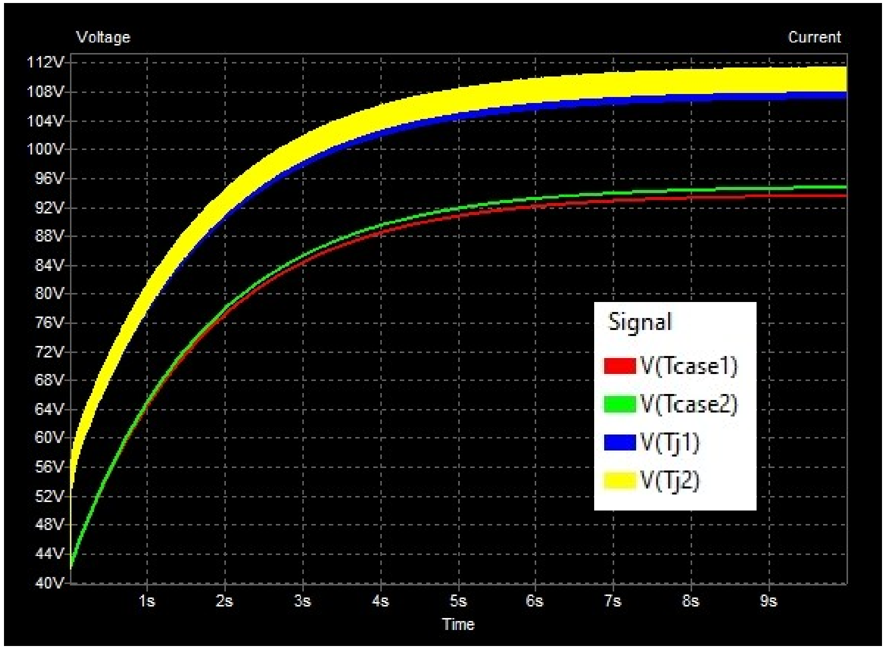
electric circuit

Amplifier electro-thermal simulation results

Integrated signal and temperature simulation



Amplifier input and output voltages.
Simulated for 100 ms at 42 °C (similar curve results at 108 °C)



Transistor case and transistor junction temperatures.
Simulated for 10 s with Tj rising from 42 to 108 °C

Modelling of thermal runaway

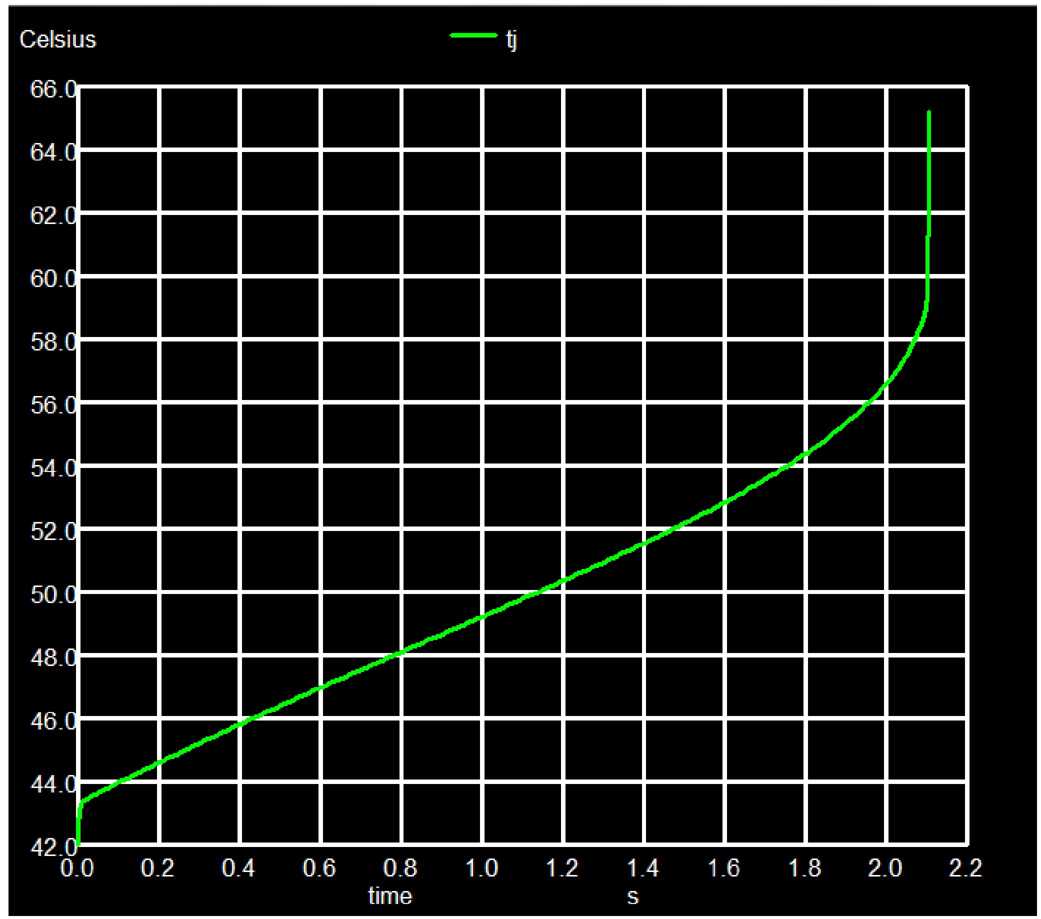
Resistor with negative temperature coefficient

1 Ω, 1 V, -0.03 Ω/K

After some time the temperature rises beyond bounds

=> thermal runaway

device destruction



Summary

Ngspice-32 will be available in March 2020 with several new features:

- Many bug fixes and improved error messages
- Electro-thermal simulation with VDMOS Power MOS model
- UNICODE compatibility
- Enhanced Monte Carlo simulation capability

And even then: Still a lot to do for ngspice-33…