Ph 623 - 02 April

- HW 9 (more digital practice) due next Thursday 1pm
- Digital lab writeup -and- Circuit Simulation Prelab* due next Wed. 3 pm

  * Note we are reversing the order of Circuit Simulation and DAC/ADC labs from Syllabus
- Exam II next Tuesday, 07 April covers through digital

Today

1. Circuit Simulation

Simulation Programs

Spice (Core of Multisim)

= Public domain program developed at UC Berkeley in early 1970's.

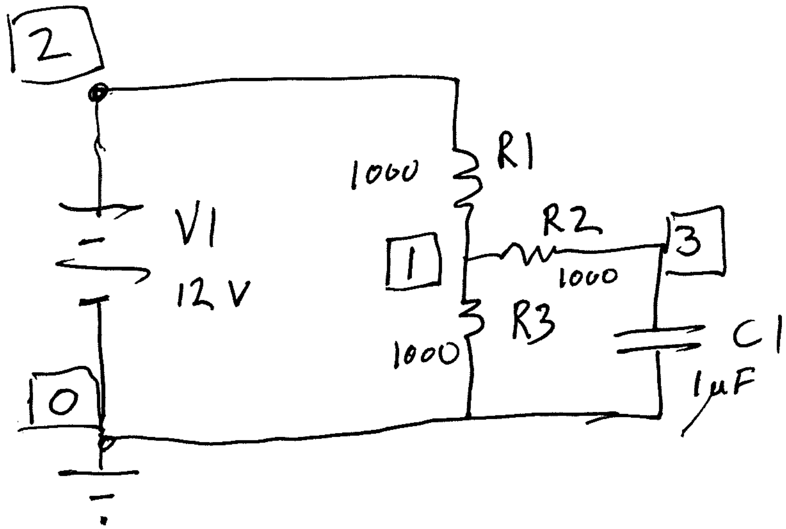
Spice I : 1973 - Fortran
Spice II : 1975 - "
Spice III : 1989 - C
Last Berkeley Spice - 1993

Various commercial versions since:
XSPICE PSPICE ISPICE
Have added support for digital simulation

Many programs - all based on a few of these SPICE "engines".

How it works

Schematic capture is outside of Spice. Spice always works from a netlist = list of components and nodes they connect.



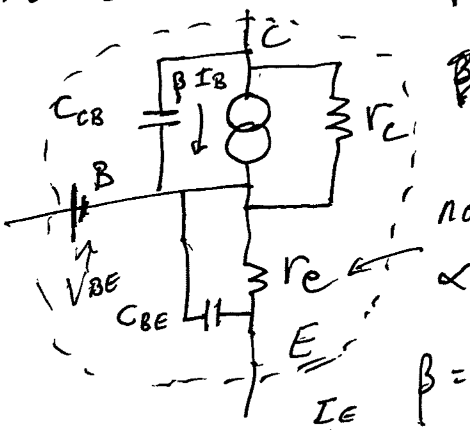
simple - Notepad

File Edit Format View Help

```
** Design1 **
*
* NI Multisim to SPICE Netlist Export
* Generated by: teacher
* Thu, Apr 02, 2020 00:31:01
*

*## Multisim Component R3 ##*
rR3 1 0 1000 vresR3
.model vresR3 r(  )

*## Multisim Component R2 ##*
rR2 1 3 1000 vresR2
.model vresR2 r(  )

*## Multisim Component C1 ##*
cC1 3 0 1e-006

*## Multisim Component R1 ##*
rR1 2 1 1000 vresR1
.model vresR1 r(  )

*## Multisim Component V1 ##*
vV1 2 0 dc 12 ac 0 0
+           distof1 0 0
+           distof2 0 0
```

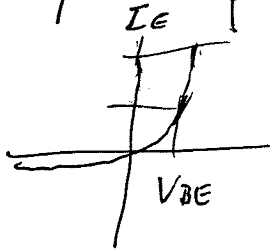
Each component has a <u>model</u>

- simple for ideal resistor
- harder for capacitor $(V = \int I \, dt)$
- Transistor = approximation



nonlinear: $\propto \frac{1}{I_E}$

$\beta = \beta(I_c)$



4 nodes $\rightarrow$ 4x4 Matrix - each element is model of components between them.

<u>Analysis</u>

<u>A.C. analysis</u> = small signal
linearize models
Solve analytically

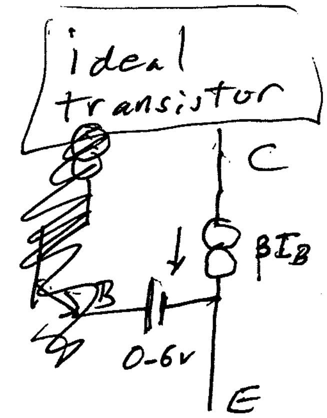
But 1^{st} need Q-point so know where to linearize around.

D.C. sweep - solve by nonlinear least squares with no input - get Q point.

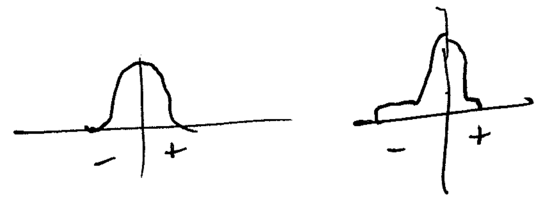
<u>Transient Analysis</u>: non-linear least squares solution at each time step.

"Wrapper" = User interface

1) Schematic Capture → Net list + GRAPHIC OUTPUT
2) Component (model) Library
   - different levels of sophistication for models



3) ~~Noise~~ Tolerance evaluation
   - Worst case
   - Monte Carlo

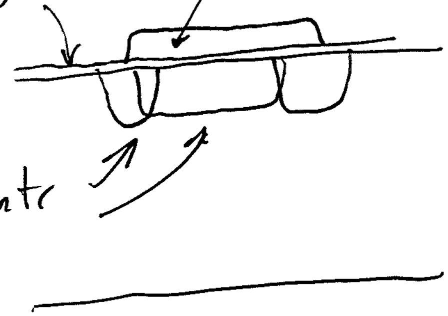


4) Temperature Effects (Tempco's in model)

5) P.C. Board layout
   - Temperture distribution
6) Strays → circuit → simulation
   - Even more important for integrated circuits ~~&~~
     a) Bigger Effects
     b) \$\$\$\$ to test

Add-ons:

Physics package to model transistor characteristics given



- dimensions
- background dopants
- implant energies
- diffusion time and temperatures
- electron + hole drift and ballistic characteristics
- Tolerance distribution on all of above

Approximate Model of component → SPICE simulation of entire circuit.

Mentor Graphics has package with <u>all</u> of these Things
(maintainance contract ~$50K/yr)

Digital Simulation

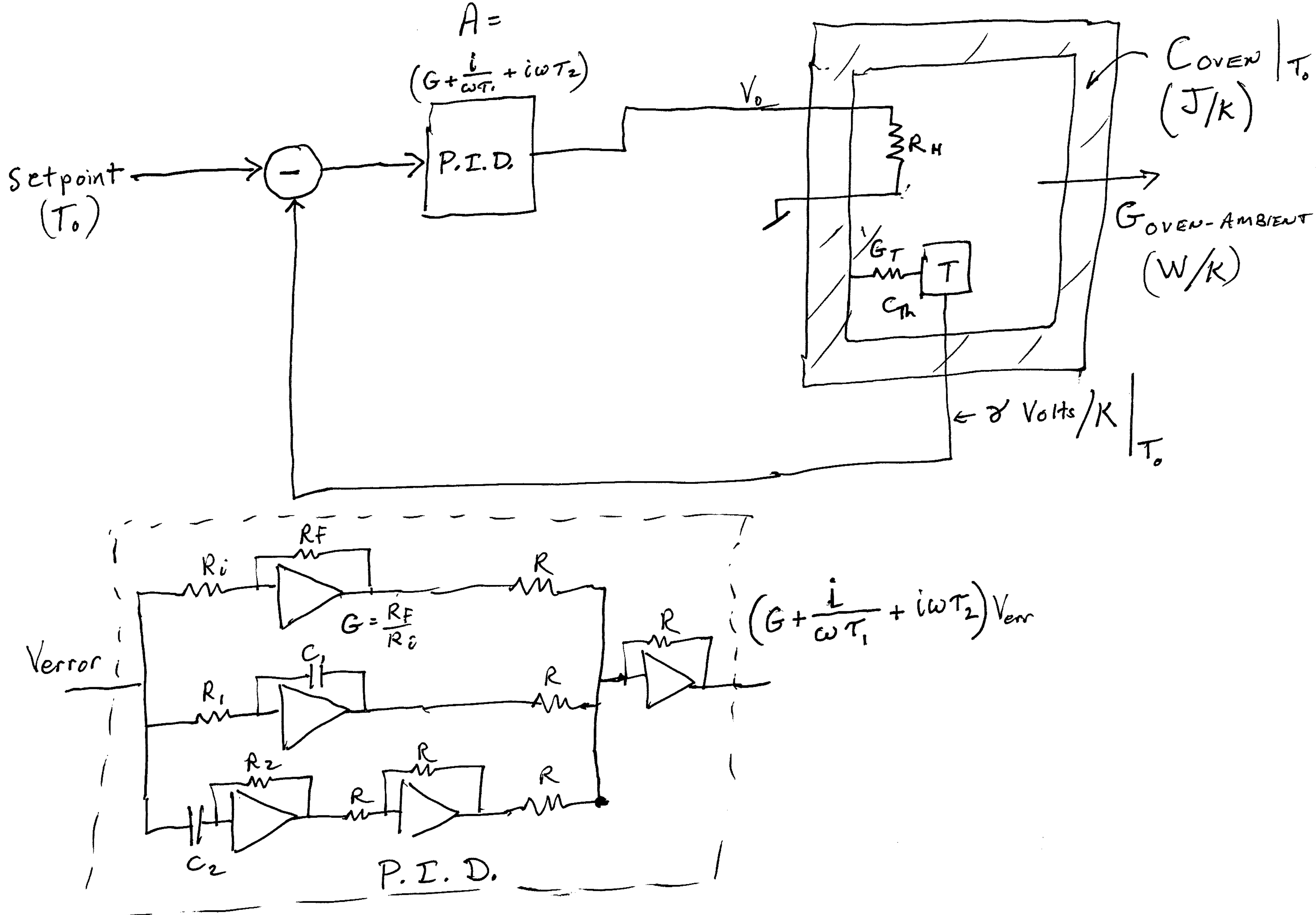
Analog is very good. Professional engineers seldom breadboard, even for high-frequency. ~~its~~ Almost always works.

Digital not as good. In very complex systems usually have problems even after ~~very good~~ best simulations.

- More potential states than you can test on computer - much longer time series than analog
- Timing is very critical
- Binary Threshold for interference

Simulators have various levels of physical abstraction:

- Transistor level
- Gate level
- Register (component) level ← multisim
- Behaviorial level (is logic correct?)



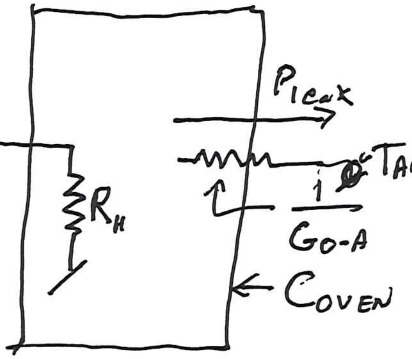
$A = \left(G + \frac{i}{\omega T_1} + i\omega T_2\right)$
Setpoint (T_0)
P.I.D.
V_0
R_H
$C_{OVEN}\big|_{T_0}$ (J/K)
$G_{OVEN-AMBIENT}$ (W/K)
$1/G_T$
T
C_{Th}
γ Volts/K$\big|_{T_0}$
R_i
RF
$G = \frac{R_F}{R_i}$
R
C_1
R_1
R_2
C_2
Verror
$\left(G + \frac{i}{\omega T_1} + i\omega T_2\right) V_{err}$
P. I. D.

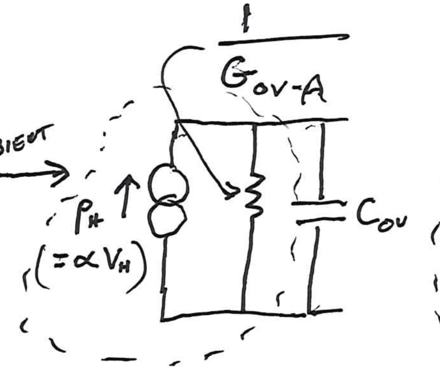
$V_H = V_0 + v_H$

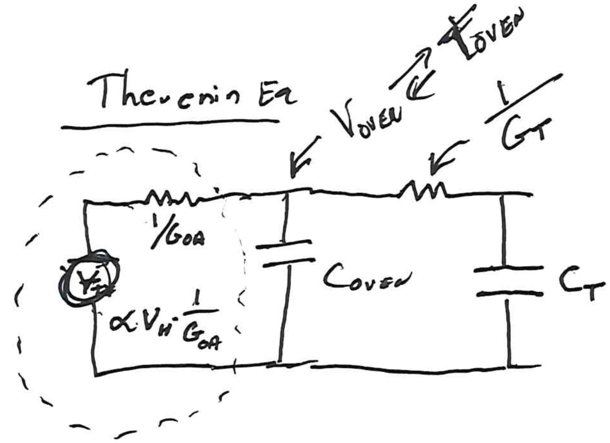
$P_H = P_0 + p_H$

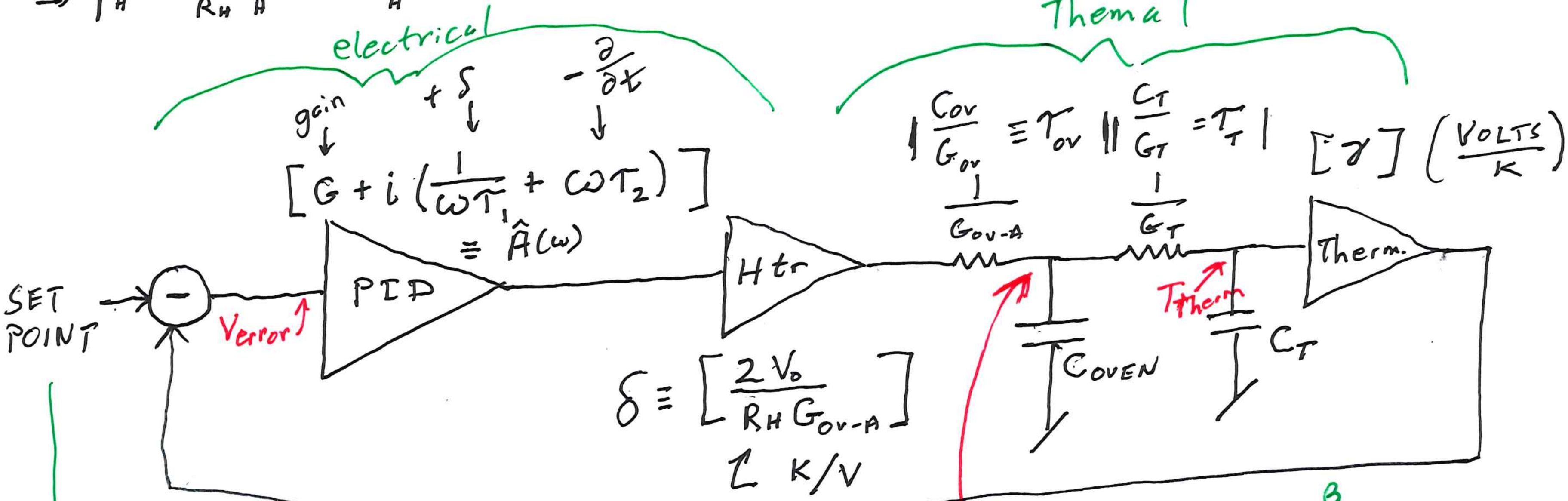
$\cong \frac{V_0^2}{R_H} + \frac{2V_0}{R_H} v$

$\Rightarrow p_H = \frac{2V_0}{R_H} v_H \equiv \alpha v_H$









Setpoint = "input"

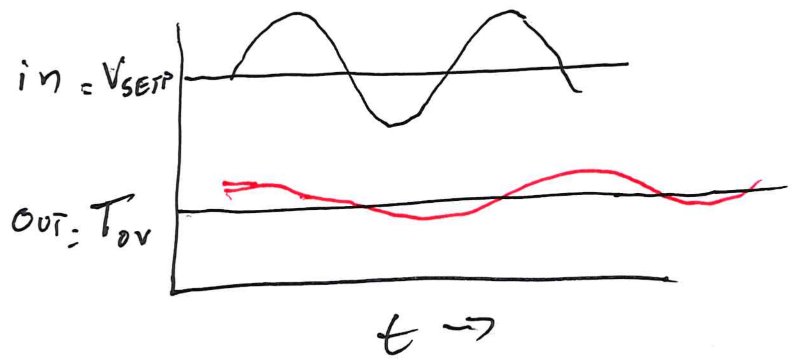
$$V_{er} \cdot \underbrace{\overbrace{\hat{A}(\omega) \cdot \delta \cdot \frac{1}{1+i\omega\tau_{OVEN}}}^{A} \cdot \overbrace{\frac{1}{1+i\omega\tau_{therm}} \cdot \gamma \cdot (-1)}^{\beta}}_{\beta A} \to V_{er}$$

T_{OVEN} = "OUTPUT"

T_{therm}

βA $\left(|\beta A| < 1\right.$ when "at Frequency where" positive real$\left.\right)$

$$A_{CL}(\omega) = \frac{A}{1-\beta A} \equiv \text{"transfer Function"}$$



$$A_{CL} \equiv \frac{T_{OVEN}}{V_{SETP}}$$

<u>Impulse Response</u> = inverse Laplace transform of transfer Function