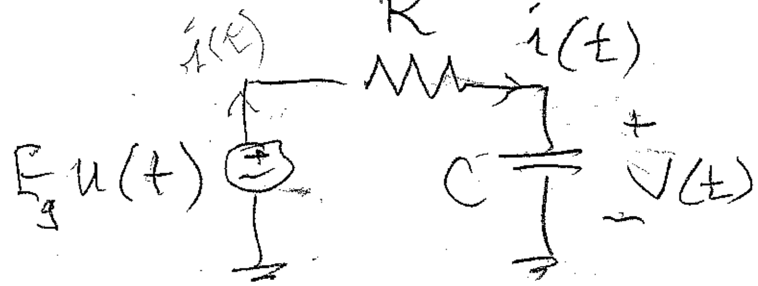


$$i(t) = C\, dv/dt$$

Power from gen. $E_g i(t)$

Energy from gen.

$$E = \int_0^{\infty} E_g C (dv/\cancel{dt})\, \cancel{dt} = E_g C \int_{V(0)}^{V(\infty)} dv = C E_g^2$$

Twice the energy stored in C!

Analysis:

$$E_g \frac{1}{s} = I(s)\left\{R + 1/(sC)\right\} \rightarrow I(s)$$

$$i(t) = (E_g/R)\, e^{-t/RC}$$

Power in R: $i^2(t) R$

Energy lost in R:

$$E_R = \int_0^{\infty} i^2(t) R\, dt = C E_g^2/2$$

Re: graphs, CAD

temes

From: "Kartikeya Mayaram" <karti@eecs.oregonstate.edu>
To: <eecs-grads@engr.orst.edu>
Cc: "Fiez, Terri / Oregon St " <terri@eecs.oregonstate.edu>; "Gabor Temes" <temes@eecs.oregonstate.edu>; "U. Moon" <moon@eecs.oregonstate.edu>; "Patrick Chiang" <pchiang@eecs.oregonstate.edu>; "PavanKumar Hanumolu" <hanumolu@eecs.oregonstate.edu>; "Kartikeya Mayaram" <karti@eecs.oregonstate.edu>; "Molly Shor" <shor@eecs.oregonstate.edu>
Sent: Tuesday, September 21, 2010 6:15 PM
Subject: Fall 2010 New Graduate Course: Analog Circuit Simulation

ECE 521 - Analog Circuit Simulation - Fall 2010 (MW 2-3:50pm)

How this course adds to the curriculum?

This course supplements other courses in the circuit design area such as ECE 4/522, ECE 4/523, ECE 520. Students use the circuit simulator SPICE extensively in these courses but are not aware of the theoretical and practical aspects of building a circuit simulator such as SPICE. This course provides them with an understanding of the key issues and also provides a stronger foundation in circuit theory and numerical methods. Essentially, this course addresses "Everything you wanted to know about SPICE but were afraid to ask!"

When/Where
Fall 2010: MW 2-3:50pm (Room TBD)

Prerequisites
A background in circuit theory, ability to write software in (C, C++, or Fortran), and an appreciation for numerical methods

Topics

1. Formulation of circuit equations using the following methods: nodal analysis (NA), modified nodal analysis (MNA), and sparse tableau approach (STA)

2. Solution of linear equations with direct and iterative methods and sparse-matrix solution techniques

3. DC analysis of circuits and solution of nonlinear equations and convergence issues

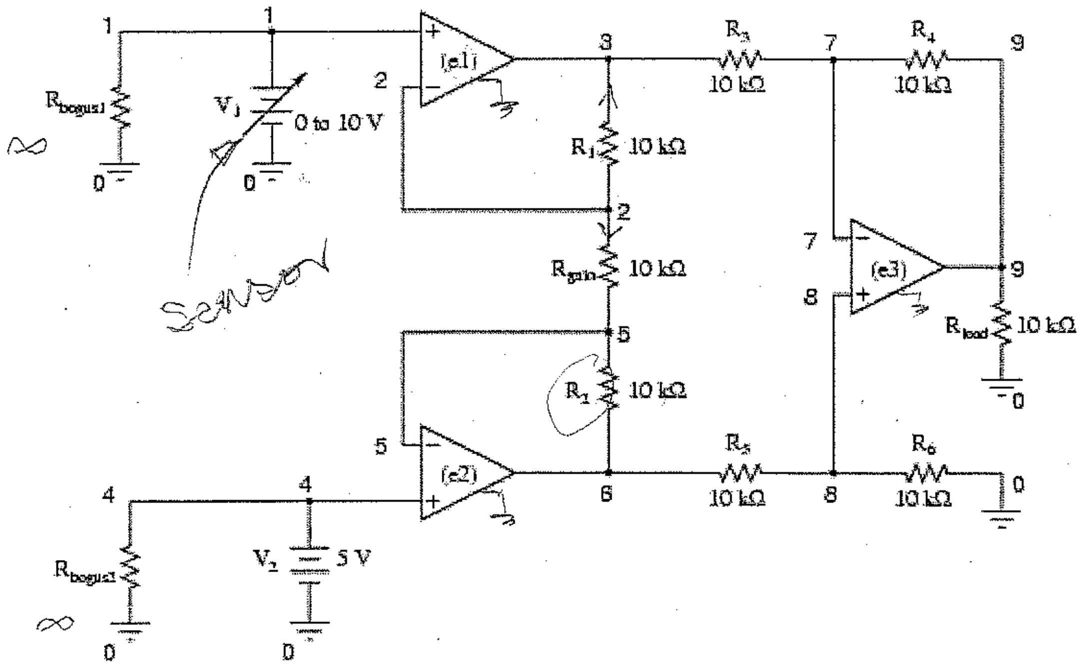
4. Small-signal ac, transient, sensitivity, noise, and pole/zero analyses

5. Analysis methods for RF circuits

12/E

CAD

Instrumentation amplifier



Note the very high-resistance R_{bogus1} and R_{bogus2} resistors in the netlist (not shown in schematic for brevity) across each input voltage source, to keepSPICE from thinking V_1 and V_2 were open-circuited, just like the other op-amp circuit examples.

ECAP — IBM
CANCER } UCB
SPICE
SCEPTRE

Netlist:

```
Instrumentation amplifier
v1 1 0
rbogus1 1 0 9e12
v2 4 0 dc 5
rbogus2 4 0 9e12
e1 3 0 1 2 999k
e2 6 0 4 5 999k
e3 9 0 8 7 999k
rload 9 0 10k
r1 2 3 10k
rgain 2 5 10k
r2 5 6 10k
r3 3 7 10k
r4 7 9 10k
r5 6 8 10k
r6 8 0 10k
.dc v1 0 10 1
.print dc v(9) v(3,6)
.end
```

VCVS – ampl.

Rs

voltage steps