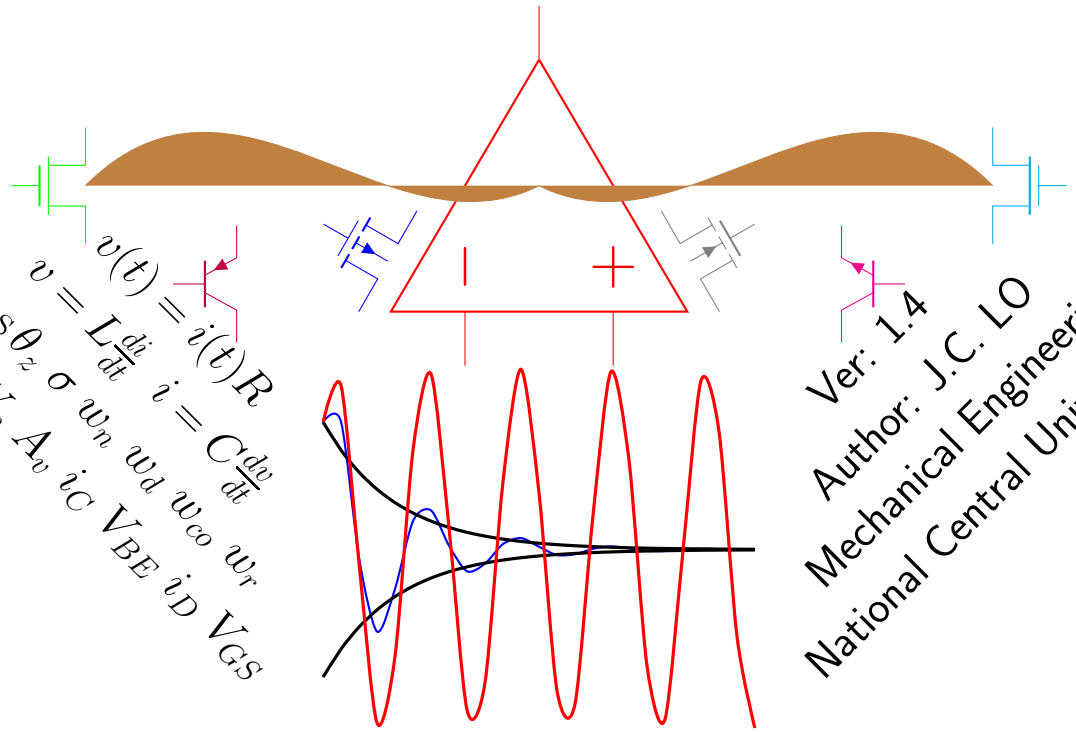
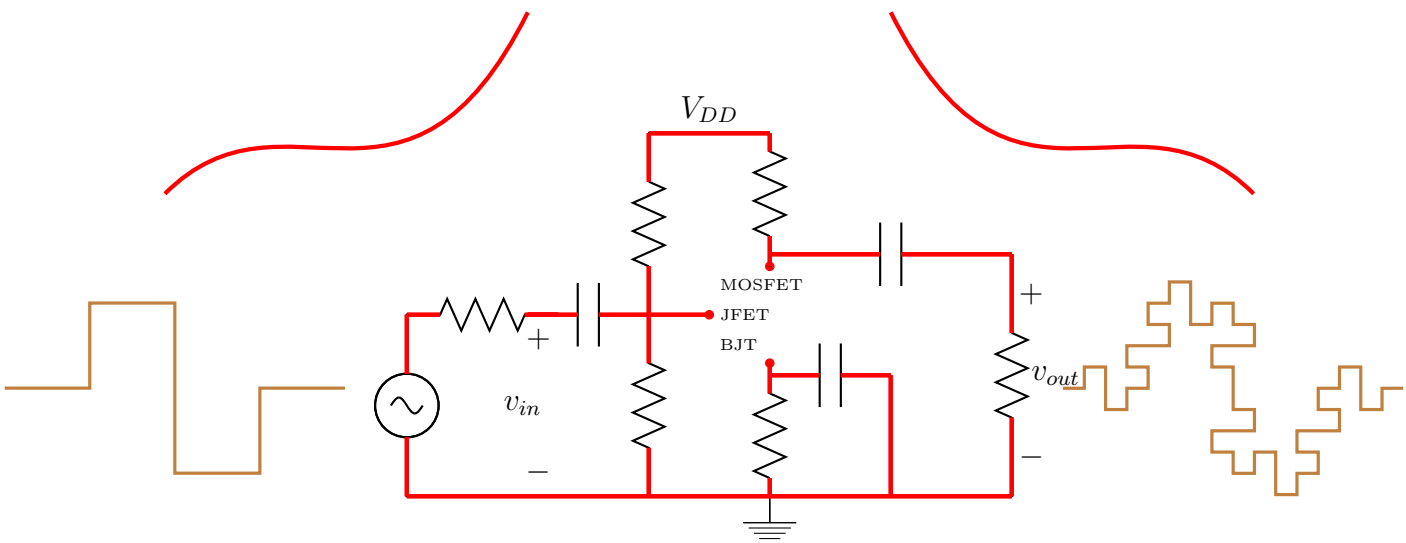


Ver: 1.4

Author: J.C. LO
Mechanical Engineering
National Central University, tw

eThinking in Circuits

with PSpice©



1 **Introduction**

2 **Circuit Laws**

3 **DC Circuits**

4 **AC Circuits**

5 **Transient Responses**

6 **Frequency Responses**

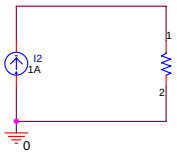
7 **Diodes**

8 **Operational Amplifiers**

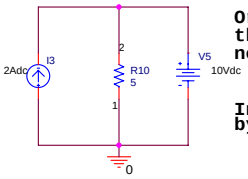
9 **Bipolar Transistors**

10 **Unipolar Transistors**

11 **PSpice Simulation Results**

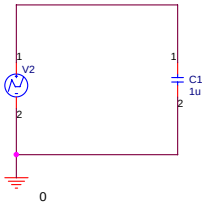


1. Construct the circuit on the left.
2. To verify Ohms' Law, open up the Simulation Settings and check DC Sweep/Primary Sweep, fill in the variable name I2, then Start value 0, End value 10, and increment 2 using linear scale.
3. Run the simulation by clicking the Big Blue Right Arrow.
4. When the probe window shows up, click Trace/Add Trace to find V1(R5) for output.
5. To find the numerical value of a particular point, click Trace/Cursor to get the cursor, move the cursor to the point you want, and click Plot/Label/Mark to add the numerical value on the graph.



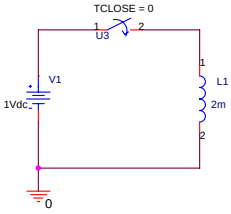
Or simply choose DC Bias Point analysis by editing Simulation Settings again. Then run the simulation. After that, click V and/or I to obtain the answer right away. But notice that this is valid only for one-time solution.

In the second example, there are two independent sources. You may explore the solution by using DC analysis again.



1. Construct the circuit on the left.
2. To verify Faradays' Law, open up the Simulation Settings and select Time Domain Transient analysis, set time period to 0.01ms.
3. Run the simulation.
4. When Probe window pops up, click Trace/Add Trace to select V1(V2) and I(C1), using comma to seperate them.

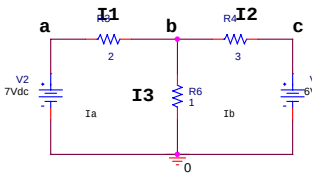
You may use the circuit above with C being replaced by L and voltage source replaced by IPWL, then verify the Henry's Law v=L di/dt. Or equivalently, check its intergral form as shown in the last circuit.



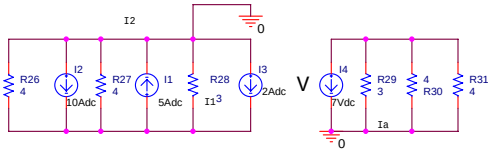
1. Construct the circuit on the left.
2. Open up the Simulation Settings, click Time Domain Transient and fill in Tstop=40ms.
3. Click the Blue Arrow.
4. When the Probe Window shows up, click Trace/Add Trace to add I(L1) and see the result.

Title: Electrical Laws		
Size: A4	Document Number: Chapter 1, JCLO	Rev: 1
Date: Wednesday, August 24, 2011	Sheet 1 of 1	

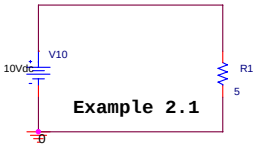
To solve DC circuits for current and voltage is to
1. Open up the Simulation Settings and select Bias Point analysis.
2. Run the simulation by clicking the Big Blue Arrow.
3. Once it is done, click V, I, W icons respectively to find solutions printed on the circuits.



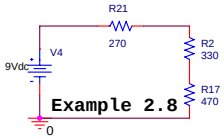
Example 2.5-2.7



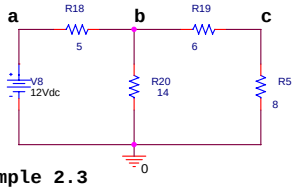
Example 2.13



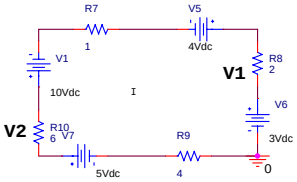
Example 2.1

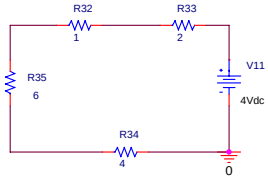


Example 2.8

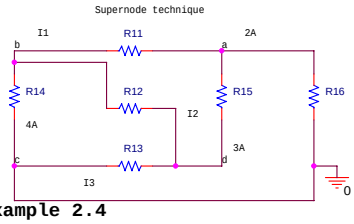


Example 2.3

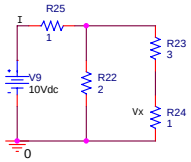




Example 2.10



Example 2.4

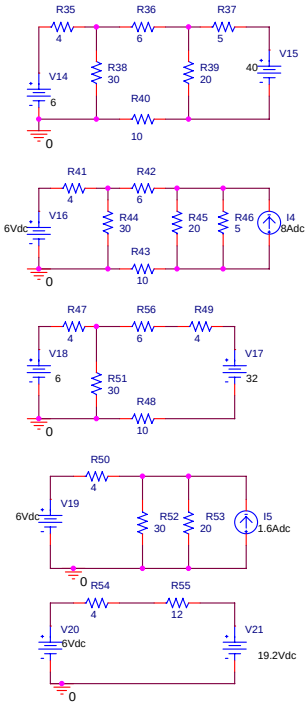


Example 2.12

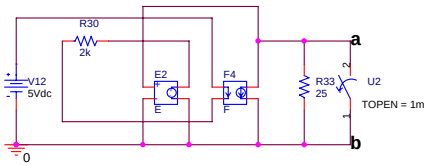
Title: Circuit Laws		
Size: A4	Document Number: Chapter 2, Page 1, JCLO	Rev: 1
Date: Wednesday, August 24, 2011	Sheet 1 of 1	

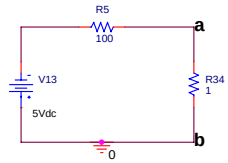
v-i Characteristics for Thevenin equivalent

1. **Construct the circuits below.**
2. **Open up the Simulation Settings, check the Time Domain Transient analysis, and set time period to 10ms.**
3. **Run the simulation. (it is noted that the switch yields short- and open-circuits)**
4. **Click Trace/Add Trace to select I(U?:2).for y-axis.**
5. **Since time domain t is the default x-axis, we need to change it. Click Plot/Axis Setting/Axis Variables to select x axis = V(U?:2). The slope is 1/R.**
6. **If one exahcnges x and y axis, then the slope is R.**

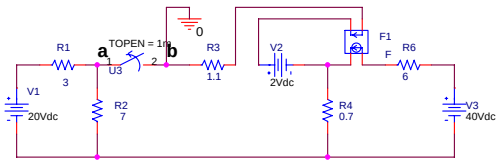


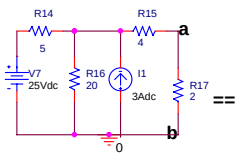
Example 3.2



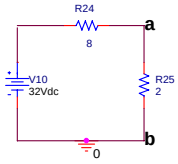


Examples 3.5

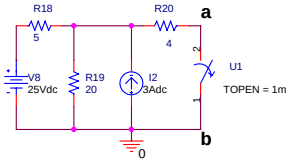




==

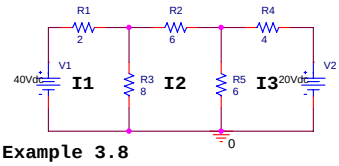


Examples 3.3-3.4

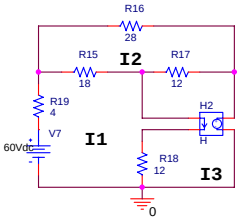


Title: DC Circuit Analysis, Equivalence and Thevenin		
Size: A4	Document Number: Chapter 3, Page 1, JCLO	Rev: 1
Date: Wednesday, August 24, 2011	Sheet 1 of 4	

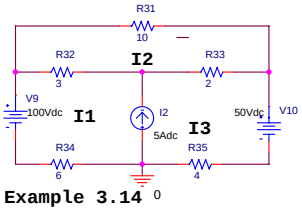
Loop-Current Approach
To find solutions for the the problems below is straightforward.
1. Open up Simulation Settings and choose Bias Point analysis.
2. Click the Big Blue Arrow to Run.
3. After simulation is complete, click V and I icon at the top of Capture menu.



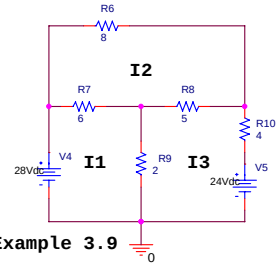
Example 3.8



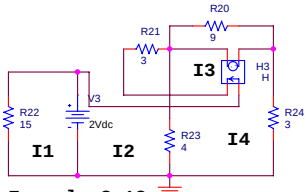
Example 3.11



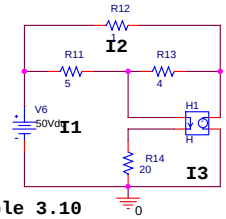
Example 3.14



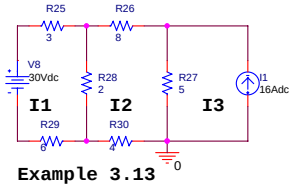
Example 3.9



Example 3.12



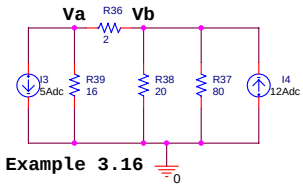
Example 3.10



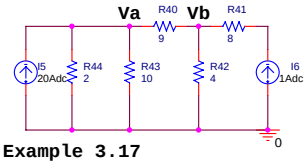
Example 3.13

Title	DC Circuit Analysis		
Size A4	Document Number Chapter 3, Page 2, JCLO		Rev 1
Date: Wednesday, August 24, 2011		Sheet 2 of 4	

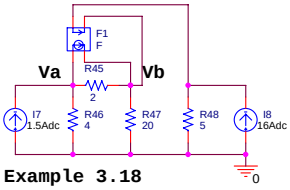
Node-Voltage Approach
To find solutions for the the problems below is straightforward.
1. Open up the Simulation Settings and choose Bias Point analysis.
2. Click the Big Blue Arrow to Run.
3. After simulation is complete, click V and I icon at the top of Capture menu.



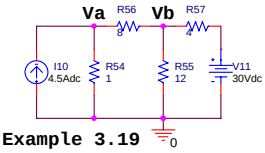
Example 3.16



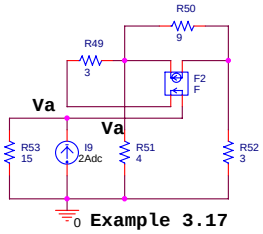
Example 3.17



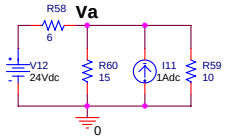
Example 3.18



Example 3.19



Example 3.17

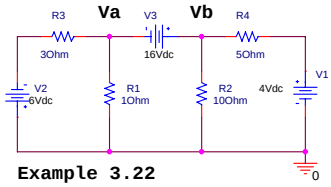


Example 3.21

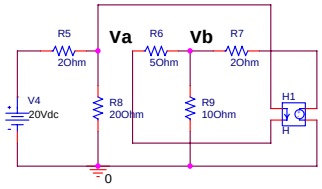
Title	Circuit Analysis		
Size A4	Document Number Chapter 3, Page 3, JCLO		Rev 1
Date: Tuesday, November 08, 2011		Sheet 3 of 4	

Node-Voltage Approach
To find solutions for the the problems below is straightforward.
1. Open up Simulation Settings and choose Bias Point analysis.
2. Click Run.
3. After simulation is complete, cick V and I icon at the top of Capture menu.

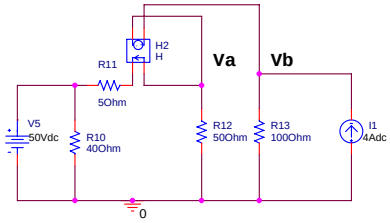
For supersition techniques
1. Basically this uses Bias Point analysis.
2. Replace on current source by an open circuit and replace an voltage source by an short circuit and leave only one active source to run.
3. Leave dependent sources alone.



Example 3.22

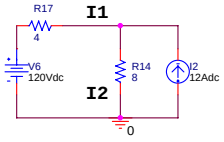


Example 3.23



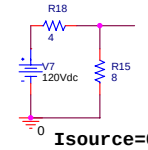
Example 3.24

Superposition



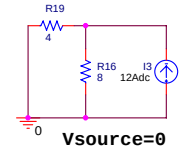
Example 3.26

===

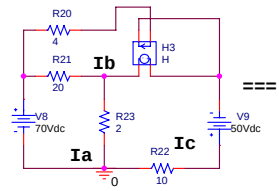


Isource=0

+

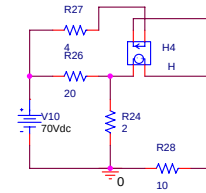


Vsource=0



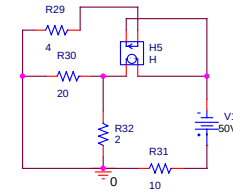
Example 3.27

===



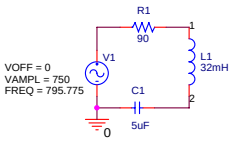
Vsource=0

+

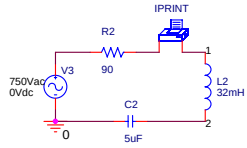


Vsource=0

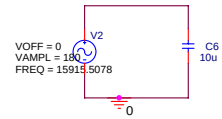
Title: DC Circuit Analysis		
Size: A4	Document Number: Chapter 3, Page 4, JCLO	Rev: 1
Date: Wednesday, August 24, 2011	Sheet 4 of 4	



Example 4.3



Freq=5000/2pi=795.775

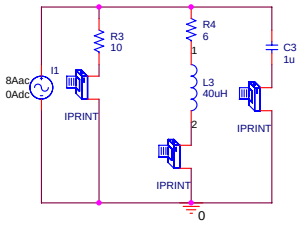


AC Power

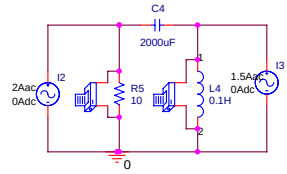
Real P=vi=180*180m/2=16.2W

Reactive P=vi=180*180*cos(-90/57.3)/2=0Var

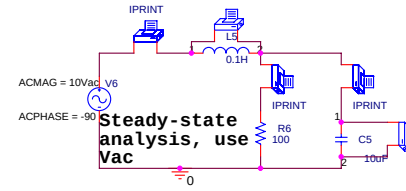
180cos(10^5t), R=1k, C=10u, L=10u



Example 4.4
Freq=200000/2pi=31831.015



Example 4.5
Freq=100/2pi=15.915

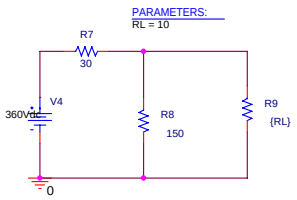


Steady-state analysis, use Vac

1. **To find transient response selects Vsin/Isin functions as sources.**
2. **draw the circuit.**
3. **Open up the Simulation Settings to select Time Domain Transient and fill in the time period.**
4. **Run the simulatiion.**
5. **When probe window appears, click Trace/Add Trace to add desired variables for plottings.**
5. **Actually, the steady-statee magnitude still can be found from the time trajectory.**

1. **To find steady-state response needs Vac/Iac functions**
2. **Select Simulation Settings/AC Sweep analysis and let Start = End frequency, using frequency in Hz, w=2\pi f, NOT in w.**
3. **Click PSpice/Markers/Advanced and select appropriate markers, connect them to the node.**
4. **Run the simulation.**
5. **Results will be shown on the probe window.**
6. **Another way is to place VPRINT2 and IPRINT in circuits and the outputs are saved in the output text fiile.**

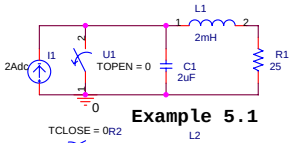
Title: Steady-State Anlysis		
Size: A4	Document Number: Chapter4, Page 1	Rev: 1
Date: Monday, October 31, 2011	Sheet 1 of 1	



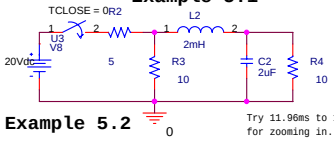
Maximum Power Transfer
1. Construct the circuit on the left.
2. Click R9 and fill in the value with {RL} (curly bracket is necessary)
3. Click Place Part/Special/PARAM and drag it to the place near R9.
4. Click the PARAMETERS to edit its property by adding a new column RL, insert a value except zero.
4. Click DC Sweep, check Global Parameter and give it a name RL, fill in (0:1:100) for sweeping range.
5. After Probe Window shows up, click Trace/Add Trace, select W(R9), to see the output vurve.

Title: Transient		
Size: A4	Document Number: Chapter 4, Page 2	Rev: 1
Date: Tuesday, November 08, 2011	Sheet 1 of 1	

Tranisent simulation
1. Open up Simulation Settings and choose Time Domain Transient analysis.
2. Fill in the Stop time. (Some suggestions are given)
3. Click Run.
4. Clici Trace/Add Trace to select the variables of concern.
5 .Check if the time trajectory is what you expect.

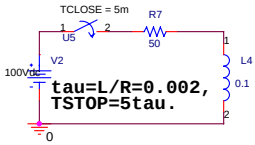


Example 5.1

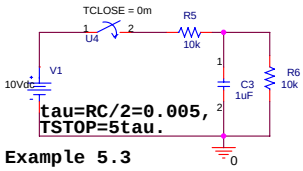


Example 5.2

Try 11.96ms to 12.06ms for zooming in.

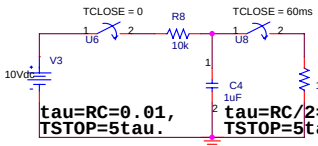


tau=L/R=0.002, TSTOP=5tau.



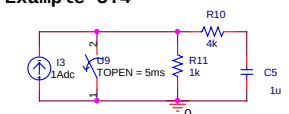
tau=RC/2=0.005, TSTOP=5tau.

Example 5.3



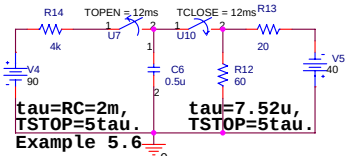
tau=RC=0.01, TSTOP=5tau. **tau=RC/2=0.005, TSTOP=5tau.**

Example 5.4



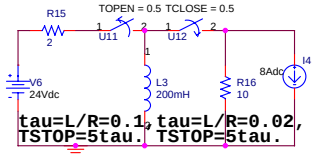
tau=(R1+R2)C=0.005, TSTOP=5tau.

Example 5.5



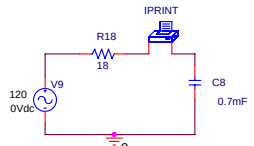
tau=RC=2m, TSTOP=5tau. **tau=7.52u, TSTOP=5tau.**

Example 5.6

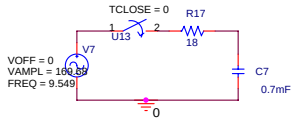


tau=L/R=0.1, TSTOP=5tau. **tau=L/R=0.02, TSTOP=5tau.**

Example 5.7



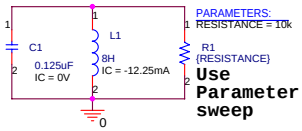
Steady-state selects Vac & AC sweep



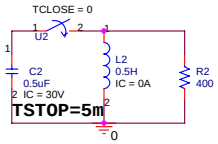
tau=RC=0.0126,TSTOP=5tau. Noting cos should be changed to sin, adding 90 in phase.

Example 5.8

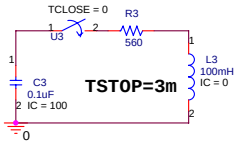
Title: Transient Analysis		
Size: A4	Document Number: Chapter 5, Page 1	Rev: 1
Date: Monday, October 31, 2011	Sheet 1 of 1	



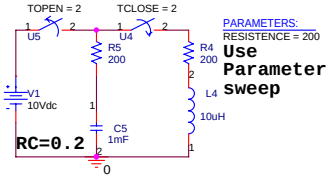
Examples 5.11-5.12



Example 5.13

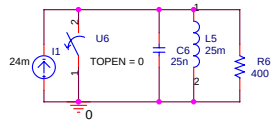


Example 5.14

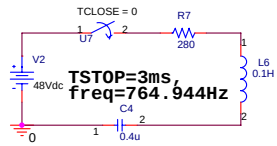


Example 5.15

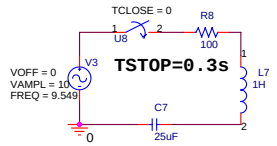
TSTOP=10. Also try 10uH for inductor whose charateristic equation is 10^-5s^2+400s+1000



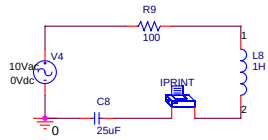
Example 5.16



Example 5.17



Example 5.18

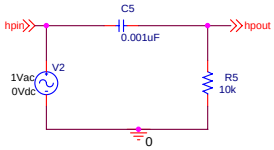


Example 5.19

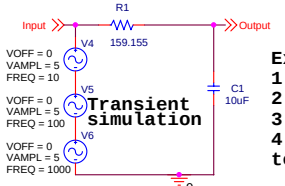
Tranisent simulation
1. Open up Simulation Settings and choose Time Domain Transient analysis.
2. Fill in the Stop time.
3. Click Run.
4. When Probe Window appears, click Trace/Add Trace to add variables of concerns to see the plot.

Parameter Sweep
1. Open up Simulation Settings and select Time Domain Transient/Parameter Sweep.
2. Fill in Parameter Name xxx. and select a Sweep type or key in a few values of interests in the List.
3. Go back to your circuit and click the element to name its value to be {xxx}
4. Click Run.
5. See the results.

Title: Switching Circuits			
Size: A4	Document Number: Chapter 5, Page2		Rev: 1
Date: Monday, October 31, 2011	Sheet 1 of 1		



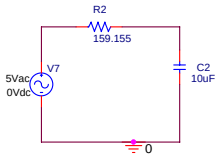
Example 6.2 -- Use VAC for frequency sweep
1. Construct the circuit on the left.
2. Click Simulation Settings, select Frequency Sweep, fill in (10:10:10E6).
3. When Probe Window appears, clcik Trace/Add Trace to add V(hpout) for linear scale or DB(V(hpout)) for log scale to generate the results.

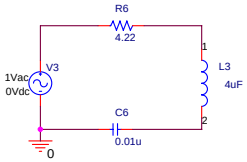


Example 6.3

Example 6.3
1. Construct the circuit on the left.
2. Open up Simulation Settings, select Time Domain Transient, fill in Stop time=100ms.
3. Run the Simulation.
4. When the Probe Window appears, click Trace/Add Trace to add V(output) and V(input) to see results, respectively.

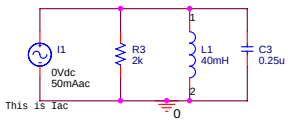
For Bode Plot
1. Use VAC source.
2. Open up Simulation Settings, select AC Sweep and fill in frequency range (0.1:10:10E4).
3. When Probe Window appears, click Trace/Add Trace to add DB(V1(C2)) for log scale.



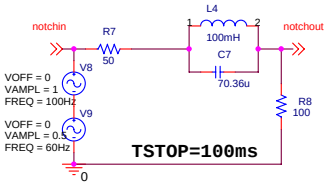


Example 6.4
1. Construct the circuit on the left.
2. Open up Simulation Settings, click AC Sweep, fill in (1K:10 10E6) and check log scale.
3. Run the Simulations.
4. When the Prob Windeow appears, click Trace/Add Trace to add DB(I(L3))
Freq=796159Hz<=>5002410rad/sec=w

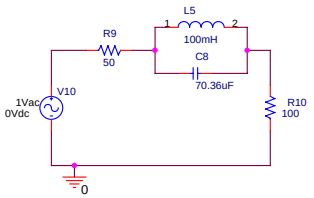
Title: Frequency Analysis		
Size: A4	Document Number: Chapter 6, Page 1	Rev: 1
Date: Monday, October 31, 2011	Sheet 1 of 1	



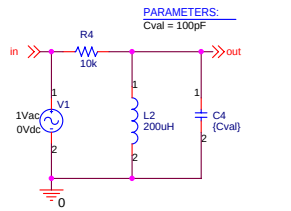
Example 6.6 -- Use IAC for frequency sweep
1. Construct the circuit on the left.
2. Click Simulation Settings, select Frequency Sweep, fill in (100:10:100K).
3. When Probe Window appears, clcik Trace/Add Trace to add I(R3) for linear scale or DB(I(R3)) for log scale to generate the results.



Example 6.7 Notch Filter
1. Construct the circuit on the left.
2. Open up Simulation Settings, select Time Domain Transient, fill in Stop time=100ms.
3. Run the Simulation.
4. When the Probe Window appears, click Trace/Add Trace to add V(notchout) and V(notchin)-V9 to see results, respectively.

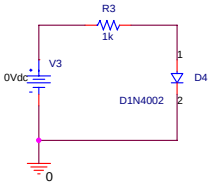


For Bode Plot
1. Use VAC source.
2. Open up Simulation Settings, select AC Sweep and fill in frequency range (10:10:100).
3. When Probe Window appears, click Trace/Add Trace to add DB(V1(R10)) for log scale.

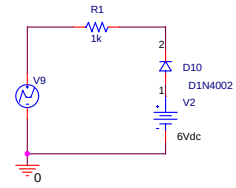


Example 6.9
1. Construct the circuit on the left.
2. Click C4 and fill in the value with {Cval} (the curly bracket is necessary)
3. Click Place Part/Special/PARAM and drag it to the place near C4.
4. Click the PARAMETERS to edit its property by adding a new column Cval, insert a value except zero.
5. Click dual AC Sweep, set (300K:10:3M) for primary, check Global Parameter and give it a name Cval, key in (100p, 200p, 300p) for Value list.
6. After Probe Window shows up, click Trace/Add Trace, select V(out), to see the output curve.

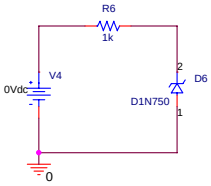
Title: Frequency Analysis		
Size: A4	Document Number: Chapter 6, Page 2	Rev: 1
Date: Tuesday, November 08, 2011	Sheet 1 of 1	



1. Construct the circuit diagram on the left.
2. Open up the Simiulation Settings by selecting DC Sweep where a dialog box will show up. Fill in sweep variable name V3 and sweep type (linear, start value -10, end value 2 and increments 0.1)
3. Run the simulation by clicking the blue solid right arrow.
4. After a blank graphic probe window appears, click Trace\Add Trace to add trace I(D4) for y-axis. Since the x-axis should be V(D4), we click Plot\Axis Settings to open up a dialog box and click axis variables to find V(D4:1).



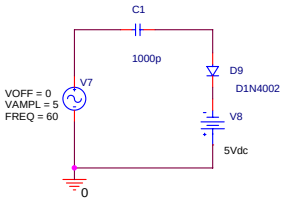
1. Construct the circuit on the left.
2. Use Transient analysis. Set final time to10ms.
3. Run the simulation by clcking the big right arrow on the menu.
4. Click Trace\Add Trace to add input and output signals, V1(V9) and V2(D10)), respectively.
5. To find the transfer function, change x-axis in time to V1(V9).



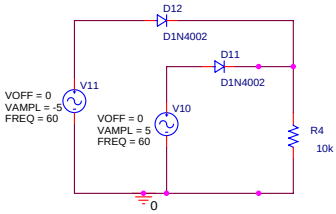
1. Construct the circuit diagram on the left.
2. Edit simiulation settings by selecting DC sweep where a dialog box will show up. Fill in Sweep Variable name V4 and Sweep Type (linear, start value 0, end value 20 and increments 0.1)
3. Run the simulation by clicking the blue solid right arrow.
4. After a blank graphic probe window appears, click Trace\Add Trace to add trace -I(D6) for y-axis. Since the x-axis should be V(D6), we click Plot\Axis Settings to open up a dialog box and click axis variables to find V(D6:2).
5. Draw a load line by clicking Plot\Label\Line and then draw the line from y-axis to x-axis or vice versa.

6. Find cursor from Trace\Cursur and click the intersection Q-point. Once this is done, go back to Plot\Label\Mark to insert the numerical value of the load line.

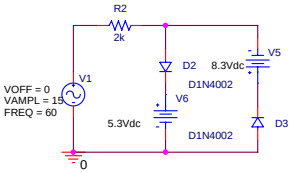
Title: Diodes		
Size: A4	Document Number: Chapter 7, Page 1, JCLO	Rev: 1
Date: Tuesday, November 08, 2011	Sheet 1 of 2	



1. Construct the circuit on the left.
2. Use Transient analysis. Set final time to 33.33m.
3. Run the simulation by clcking the big right arrow on the menu.
4. Click Trace\Add Trace to add input and output signals, V1(V7) and V1(D9), respectively.

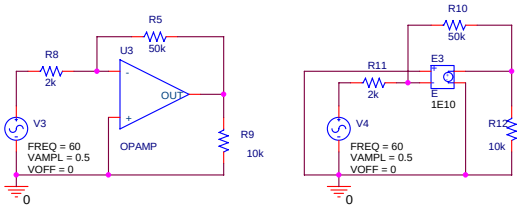


1. Construct the circuit on the left.
2. Use Transient analysis. Set final time to 33.33m.
3. Run the simulation by clcking the big right arrow on the menu.
4. Click Trace\Add Trace to add input and output signals, V1(R4).
5. Add one smoothing capacitor in parallel with the load.

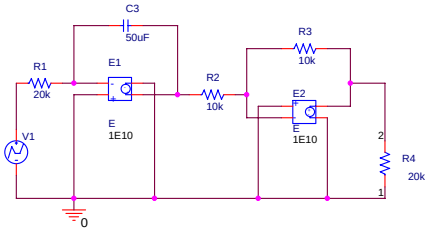


1. Use Transient analysis. Set final time to 33.33m.
2. Run simulation by clcking the big right arrow on the menu bar.
3. Click Trace/Add Trace to find V1(V1) and V1(D2)+V1(V6) respectively.
4. To find the transfer function Vout/Vin, clcik Plot/Add Plot to Window to generate a new blank plot and then Plot/Plot Settings/Axis Variables V1(V1) as x-axis and Add/Add Trace V1(D2)+V(V6) as y axis.

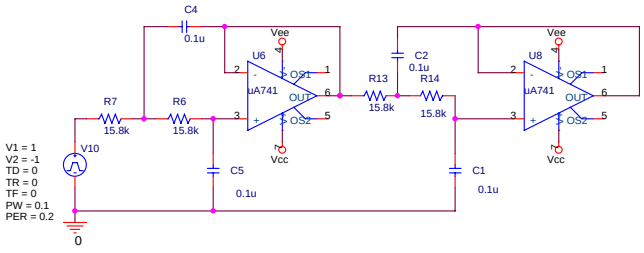
Title: Diodes		
Size: A4	Document Number: Chapter 7, Page 2, JCLO	Rev: <RevCo
Date: Monday, October 31, 2011	Sheet 2 of 2	



1. Construct the circuit on the left.
2. Open up the Simulation Settings and select the Traisnent analysis, setting simulation time period to 33.33ms.
3. Run the simulation.
4. Click Trace/Add Trace to add V1(R9) and V1(R12) to see identical plot, meaning VCVS = op-amp.
5. Click Trace/Cursor/Displace to find cursor and then click Plot/Label/mark to insert numerical values.



1. Construct the circuit on the left.
2. To verify integration, open up the Simulation Settings and select Time Domain Transient analysis, set time period to 4s. where pulse function of 4 second is given by VPWL.
3. Run the simulation.
4. When Probe window pops up, click Trace/Add Trace to select V1(V1) and V2(R2), using comma to seperate them.

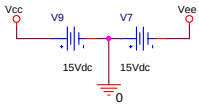


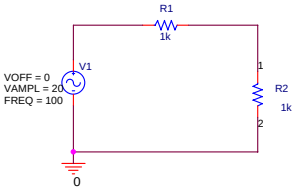
To obtain Frequency response, use VAC
1. Construct the circuit on the left.
2. Open up Simulation Settings, click AC Sweep, fill in (1k:10:10K).
3. Click the Big Right Arrow to run the simulation.
4. When the Probe Window appears, click Trace/Add Trace to add V(U6out) and V(u8out) to see the Bode plot.

To find transient response, use VPULSE
2. Open up Simulation Settings, click AC Sweep, fill in (1k:10:10K).
3. Click the Big Right Arrow to run the simulation.
4. When the Probe Window appears, click Trace/Add Trace to add V(U6out) and V(U8out) to see the Bode plot.

VCC_CIRCLE can be found at GNG/CAPSYM

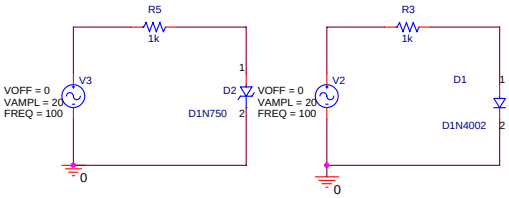
Title: Operational Amplifier			
Size: A4	Document Number: Chapter 8, Page 1, JCLO		Rev: 1
Date: Wednesday, August 24, 2011	Sheet 1 of 1		





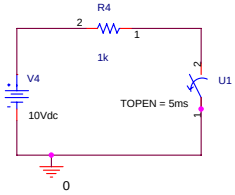
Linear element --- V-I characteristics

1. Construct the circuit on the left.
2. Open up the Simulation Settings, check the Time Domain Transient analysis, and set time period to 10ms.
3. Run the simulation.
4. Click Trace/Add Trace to select I(R2:1). A sine wave is shown.
5. Since time domain t is the default x-axis, we need to change it. Click Plot/Axis Setting/Axis Variables to select x axis =V(R2:1).



Nolinear element --- v-i characteristics

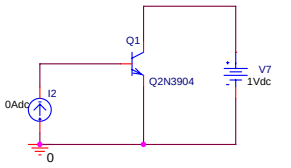
1. Construct the circuit on the left.
2. Open up the Simulation Settings, check the Time Domain Transient analysis, and set time period to 10ms.
3. Run the simulation.
4. Click Trace/Add Trace to select I(D1:1)
5. Since time domain t is the default x-axis, we need to change it. Click Plot/Axis Setting/Axis Variables to select x axis =V(D1:1).
5. Change D to D2, Zener v-i property is obtained.



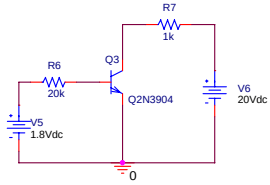
v-i Characteristics for Thevenin equivalent

1. Construct the circuit on the left.
2. Open up the Simulation Settings, check the Time Domain Transient analysis, and set time period to 10ms.
3. Run the simulation. (it is noted that the switch yields short- and open-circuits)
4. Click Trace/Add Trace to select I(U1:2).
5. Since time domain t is the default x-axis, we need to change it. Click Plot/Axis Setting/Axis Variables to select x axis = V(U1:2).

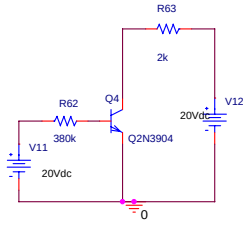
Title: BJT		
Size: A4	Document Number: Chapter 9, page 1	Rev: 1
Date: Tuesday, November 08, 2011	Sheet 1 of 1	



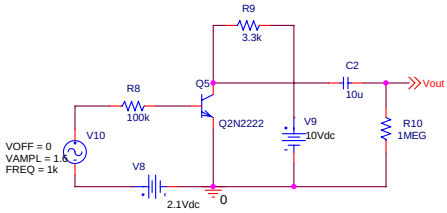
1. To find input characteristic Ib-Vbe, use DC sweep analysis by open up the Simulation Settings and select primary for current source I2(=Ib) using (0:10u:150u).
2. Run the simulation.
3. Click Trace/Add Trace to find IB(Q1) for y-axis.
4. Click Plot/Axis Settings/Axis Variable to find V(Q1:b) - V(Q1:e) for x-axis.
5. To find output characteristic Ic-Vce, use dual DC sweep where primary sweep is V7(=Vce) using (0:0.01:10) and the second sweep is I2(=Ib) using (0:10u:50u).
6. Click Trace/Add Trace to find IC(Q1) for y-axis.
7. Click Plot/Axis Settings/Axis Variable to find V1(V7) for x-axis.



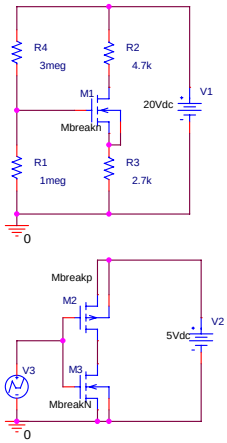
To find input characteristic Ib-Vbe, use DC sweep on v5 (0:0.01:10)
To find output characteristic Ic-Vce, use dual DC sweep on v6 (0:0.1:10) and V5 (0:0.2:2)



To find Q point, simply run bias point analysis and click V and I button.



Title: BJT		
Size: A4	Document Number: Chapter 9, Page 2	Rev: 1
Date: Wednesday, August 24, 2011	Sheet 2 of 2	



To verify input characteristic:
1. Click Place Part/Breakout/Mbreakn or Mbreakp, drag it to the place you favor. Click the Mbreakn once and from top menu, choose Edit/Model to add/copy the following
.MODEL Mbreakn NMOS (KP = 50E-6,Vto=2,lambda=0.01)
.MODEL Mbreakp PMOS (KP = 20E-6,Vto=-2,lambda=0.01)
to the model editor and then save it.
2. Click twice to edit NMOS, fill in W and L values.
3. Open up the Simulation Settings and select the Primary DC Sweep for V1 (1:1:5).
4. Run the Simulation by click the Big Blue Arrow.
5. When the probe window appears, click Trace/Add Trace and select ID(M1) for y-axis. Click Plot/Axis Setting/Axis Variables and select V(M1,g)-V(M1,s).
6. Click Plot/Label/Line to draw the load line. When drawing, change range of x- and y-axis if necessary.

To verify output characteristic:
1. Open up the Simulation Settings and use DC Dual Sweep. Let Primary Sweep be V2 (1:1:30) and secondarysweep be V1 (1:1:5).
2. Run the simulation.
3. When the probe window appears, click Trace/Add Traceand select ID(M1) for y-axis. Click Plot/AxisSetting/Axis Variables and select V(M1,d)-V(M1,s).
4. Click Plot/Label/Line to draw the load line. When drawing, change range of x- and y-axis if necessary.

DC Bias Point analysis
1. The easist way is to use Bias Point and then click the icons V and I to find Q point.

CMOS Inverter Gate
1. Construct the circuit on the left.
2. Open up Simulation Setting and choose Time Domain Transient analysis for 5 sec.
3. Run the simulation by clicking the big blue arrow.
4. After the probe window appears, click Trace/Add Trace to select V1(V3) and V(M2:d) for outputs.
5. Adjust range of y-axis for better presentation.

Title: MOSFET		
Size: A4	Document Number: Chapter 10, Page 1	Rev: 1
Date: Monday, October 31, 2011	Sheet 1 of 1	

Index

$\Delta - Y$ transform, 84
n-type material, 233, 309
p-type material, 233, 309
pn junction, 233
3-dB bandwidth, 213, 218, 223
  parallel, 219
  series, 214

AC excitation, 243
Admittance, 93
Amplifiers
  BJT, 305
  NMOS, 327
  op-amp
    CCCS/current amp, 266
    CCVS/transresistance amp, 268
    VCCS/transcondance amp, 267
    VCVS/voltage amp, 263
Average value, 89

Bias circuit, 236, 291
BJT
  active region, 289
  base, 287
  base bias, 290
  collector, 287
  common-base, 305
  common-collector, 307
  common-emitter, 303
  cutoff region, 289
  emitter, 287
  emitter follower, 307
  feedback bias
    collector-feedback, 297
    emitter-feedback, 297
  load line, 291
  saturation current, 288
  saturation region, 289
  self-bias circuit, 297
  switch, 294
  thermal voltage, 288
Bode plot, 201, 210
  high frequency asymptote, 205
  line approximation, 203
  low frequency asymptote, 205
Bypassing capacitor, 245, 302

Capacitance, 10, 101, 112, 128, 217
Characteristic equations, 135, 140, 151, 159, 165, 172, 223
Circuit analysis, 51
  loop-current method, 51
  mesh method, 51
  node-voltage method, 57
  nodeal method, 57
Circuit element laws, 8, 91
  passive
    Faraday's law, 9
    generalized Ohm's law, 166
    Henry's law, 11
    Ohm's law, 8, 98
Circuit elements
  active
    BJT, 287
    FET, 317
    op-amp, 263
  passive
    capacitor, 9

diode, 233
inductor, 11
resistor, 8
Circuit network laws
KCL, 22
KVL, 22
Circuit quantities
charge, 3
current, 4
energy, 6
power, 6
voltage, 5
Circuit theorems
equivalence, 37
Norton equivalent, 13, 44
source transformation, 38, 58
Thevenin equivalent, 13, 43, 44, 107
Clipping, 251
Complex conjugate, 104, 114
Conductance, 9, 31
Continuity of energy, 12
Conventions
current convention, 4
passive convention, 5, 8
power convention, 6
voltage convention, 5
Current, 4
alternating current (AC), 4
direct current (DC), 4
Cutoff frequency, 210

Deactivation, 65
Definitions
$\alpha = \frac{i_C}{i_E}$, 288
$\beta = \frac{i_C}{i_B}$, 288
(oc) transresistance gain, 268
(oc) voltage gain, 263
(sc) current gain, 266
(sc) transconductance gain, 267
3-dB bandwidth, 212
active elements, 3, 287
apparent power, 102
branch, 21
charge, 4
circuit, 21
current, 4
current gain, 263
decade, 202, 207, 215
decibels, 201
energy, 6
ideal diode, 240
ideal op-amp/voltage amp, 264
impedance, 91
linearity, 65
loop, 21
node, 21
octave, 204
passive elements, 3
phasor, 93
pinch-off voltage, 318
power, 6
RMS (sine function), 89
RMS (triangular function), 122
voltage, 5
Zener diode, 234
Dependent sources, 14, 45, 49, 57
current-controlled current source/CCCS, 15, 300
current-controlled voltage source/CCVS, 15
voltage-controlled current source/VCCS, 14, 300, 327
voltage-controlled voltage source VCVS, 14
voltage-controlled voltage source/VCVS, 272
Depletion
pJFET, 320
diode, 233
Diffusion, 233
Diodes
breakdown voltage, 234

clampers
negative, 250
negative-biased, 259
positive, 250
positive-biased, 259
clippers
negative, 249
negative-biased, 259
positive, 249
positive-biased, 259
forward bias, 234
full-wave rectifier, 248
half-wave rectifier, 247
knee/threshold voltage, 234, 237
load line, 236
piecewise-linear model, 242
reverse bias, 234
voltage attenuator, 245
Dynamic equation, 145

Euler's formula, 90

Filters
active
Butterworth filter, 277
low-pass filter, 277
passive, 227
band-pass filter, 212
band-reject filter, 222
high-pass filter, 198
low-pass filter, 195
notch filter, 220, 222
Frequencies
angular frequency, 89
break frequency, 199, 205
corner frequency, 196, 199, 205
cutoff frequency, 199, 205
damped frequency, 159, 223
half power frequency, 199, 205
natural frequency, 159, 218
resonant frequency, 212, 218

Hertz, 90

Ideal independent sources
ideal current source, 13
ideal voltage source, 12
Impedance, 93, 166, 214
Inductance, 11, 99, 128, 217

JFET
*n*JFET, 317
*p*JFET, 320
depletion region, 318
drain, 317
gate, 317
linear region, 319
pinch-off voltage, 318
reverse bias, 317
saturation region, 319
source, 317
voltage-controlled resistor, 319

Laplace transform, 158, 165
Large-signal analysis
BJT, 289
diodes, 245
NMOS, 323
PMOS, 323
Load
capacitive load, 99
inductive load, 99
resistive load, 98
Loading effect, 13, 276
Logic gates
BJT
NOT, 294
CMOS
NAND, 332
NOR, 332
NOT, 332
diode
AND, 241
OR, 257

Maximum value, 102
Maximum/peak value, 89
Mobile carriers
  electrons, 233, 318
  holes, 233, 318

NMOS
  bias circuit, 323
  body, 321
  common-gate, 330
  common-source, 327
  cutoff region, 323
  drain, 321
  gate, 321
  large-signal circuit, 324
  load line, 323
  saturation region, 323
  saturation-triode boundary, 323
  small-signal circuit, 324
  source, 321
  source follower, 328
  switch, 332
  triode/linear region, 323
Norton, 44

Op-amp
  averaging amp, 272
  comparator amp, 277
  difference amp, 274
  differentiator amp, 280
  integrator amp, 280, 282
  inverting amp, 270
  non-inverting amp, 273
  summing amp, 272
  VCVS, 272
  voltage follower, 276
Open circuit, 38, 44, 65, 302
OrCAD, 335

Permeability, 11
Permittivity, 10
Phasor diagram, 95, 104, 105, 217
Polarity, 5, 321
Power, 6
  apparent power, 102
  average reactive power, 102
  average real power, 102
  capacitive power, 99
  complex power, 102
  inductive power, 99
  instaneous power, 101
  maximum power transferred, 107
  power angle, 103
  power triangle, 103
  resistive power, 98
  Volt-Amperes Reactive (VAR), 102
Power factor
  lagging power factor, 103
  leading power factor, 103
PSpice, 335
  analyses
    AC Sweep, 97, 216, 278
    bias point, 9, 235, 239, 249, 251
    DC sweep, 9, 235, 239, 249, 251
    dual DC sweep, 291
    time domain transient, 11
  commands
    PAREMETER, 109
    plot/axis settings/axis variables, 9, 235, 239, 249, 251
    plot/label/line, 325
    plot/label/mark, 9, 235, 239, 249, 251, 325
    simulation settings, 9, 11, 235, 239, 249, 251
    trace/add trace, 9, 11, 235, 239, 249, 251
    trace/cursor, 9, 235, 239, 249, 251
  edit model
    NMOS, 332
    PMOS, 332
  edit property
    add new column, 109

dependent source, 17
IPRINT, 114
VPRINT2, 114
elements
C, 11
D*, 234
D1N4002, 235, 249
D1N750, 239
D_HI, 278
D_LO, 278
Do, 234
GND, 9
L, 12
Mbreakn, 324, 325, 332
Mbreakp, 324, 332
opamp, 265
R, 9
sD, 234
Sw_tClose, 44
Sw_tOpen, 44
UA471, 265, 278
VCC_CIRCLE, 278
sources
E, 14, 272
F, 15
G, 14
H, 15
IAC, 13, 97
IDC, 9, 13
IPWL, 11
ISIN, 13, 251
VAC, 12, 278
VDC, 9, 12
VPULSE, 278, 294
VPWL, 10, 11, 237
VSIN, 12, 251

Quality factor
parallel, 219
series, 214
Quiescent point, 245
DC operating point, 236, 291, 292, 323

Resistivity, 8
Responses
forced/steady-state, 127
natural/transient, 127
RMS/Effective value, 89
Root Means Square value (RMS), 102
Roots
second-order system, 159

Sallen-Key topology, 277
Saturation, 251
Schockley equation, 234
Second-order systems, 157
parallel, 158
series, 164
Selective networks, 195, 213
Short circuit, 39, 44, 65, 302
Signals
complex, 89
exponential, 89
sinusoidal, 90
Small-signal analysis
BJT, 299
diodes, 245
NMOS, 326
Steady-state error, 142
Summing-point constraint, 264
Supermesh, 22, 25, 55
Supernode, 23, 25, 63, 314
Superposition, 65, 245, 274
Systems
first-order systems, 130, 155, 157
forced systems, 157
second-order systems, 157, 278
unforced systems, 157

Thermal voltage, 234
Thevenin, 43, 108, 177, 236, 271, 295
Time constant, 132, 135
Time line, 129

Transfer function, 196, 236, 249
Transient responses
    critically damped response, 160, 166
    first-order systems, 130
    overdamped response, 159, 166
    second-order systems, 157
    underdamped response, 160, 166
Transistors
    bipolar
      npn BJT, 287
      pnp BJT, 287, 309
    unipolar
      *n*JFET, 317
      *p*JFET, 317
      IGFET/MOSFET, 317
      NMOS, 317
      PMOS, 317

Virtual open circuit, 264
Virtual short circuit, 264

Wheatstone bridge, 64, 275

Zener
    voltage regulator, 237

Bibliography

[1] A. R. Hambley, *Electrical Engineering, Principles and Applications, third Edition.* Prentice Hall, 2005.

[2] C. R. Paul, *Fundamentals of Electric Circuits Anslyis.* Wiley, 2001.

[3] J. W. Nilsson, *Electric Circuits.* Addison Wesley, 1984.

[4] P. Z. Peebles and T. A. Giuma, *Principles of Elecrical Engineering.* McGraw Hill, 1991.

[5] J. Clemons, F. Kerr, F. Evangelisti and C. Klingensmith *Introductory Electronic Devices and Circuits.* Prentice Hall, 1994.

[6] http://www.ee.cyu.edu.tw/garylee/pspice/animation/index.htm

[7] http://www.seas.upenn.edu/ jan/spice/PSpicePrimer.pdf

[8] http://www.electronics-lab.com/downloads/schematic/013/tutorial/PSPICE.pdf

[9] http://people.msoe.edu/ saadat/pspiceintro253.htm

[10] http://www1bpt.bridgeport.edu/ thapa/eleg458/tut/pspice/ PSPICE_MOS_TUT5.pdf

[11] http://www.ee.nmt.edu/ rison/ee321_fall02/Tutorial.html

書名: **eThinking in Circuits with PSpice©**
ISBN 978-957-41-8721-8
著作人: 羅吉昌
封面設計: 羅吉昌
著作完成日: 西元2011年10月31日

著作人資訊:
出版者: 羅吉昌

印製者: 淵明印刷有限公司
地址: 台北縣永和市福和路164號4樓
網址: http://www.ym.com.tw/
電話: 02-89255555

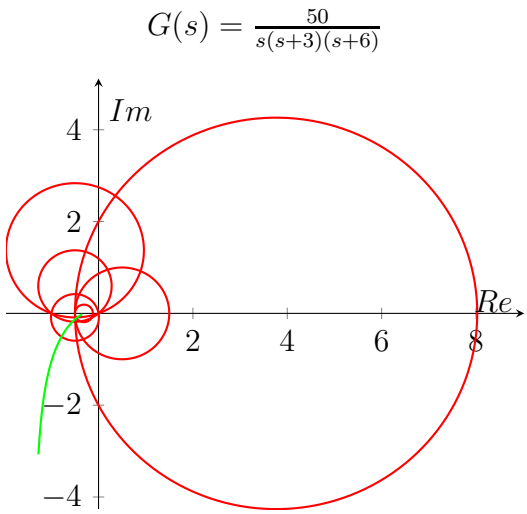
出版日期: 西元2012年01月

Date

Dept.

Institution

Name



About the author

J.C. LO is a Professor of Mechanical Engineering at National Central University, Taiwan, receiving his Ph.D. degree in Electrical Engineering from Michigan State University, East Lansing, Michigan, in 1990. Since graduation, he jointed the Mechanical Engineering, actively teaching in circuits, microprocessor, and automatic control and deeply indulging in the area of fuzzy, robust, linear and nonlinear control.

He has served as a regular peer reviewer for many renowned technical Journals and Institutions, domestic and abroad. His current interests focus on homogenously polynomial parameter dependent Lyapunov methods analyzing stability/stabilization problems via Linear Matrix Inequalities and Sum of Squares, with publication credits for numerous papers and reports in his area of expertise.

ISBN 978-957-41-8721-8

9 789574 187218

National Central University `http://www.ncu.edu.tw` **Cover Illustration by J.C. LO**