Circuit Simulation Using SPICE

ECE222
Circuit Design Flow

1. Idea Conception
2. Specification
3. Initial Circuit Design
4. Circuit Simulation
5. Meet Spec?
   - Modify Circuit Design
   - Circuit Implementation
Circuit Simulation

- Voltage and current signals are represented as continuous variables.
- Circuit elements are represented by linear or nonlinear device models.
- Circuit is described by a set of nonlinear differential equations.
- Time-domain circuit simulation means solving the differential equations at each time step iteratively based on previous step.
- Initial condition is required.
- Accuracy depends on the error criterion set.
Example:

Node voltages: \( V_{e1} \quad V_{e2} \)

Branch currents: \( i_{Vs} \quad i_R \quad i_C \)

Device models:
\[
\begin{align*}
\quad i_R &= G V_R \\
\quad i_C &= C \frac{dV_C}{dt}
\end{align*}
\]

Circuit equations using Modified Modal Analysis:
\[
\begin{align*}
G(V_{e1} - V_{e2}) + i_{Vs} &= 0 \\
C \frac{dV_{e2}}{dt} - G(V_{e1} - V_{e2}) &= 0 \\
V_{e1} &= V_S
\end{align*}
\]

Circuit equations for numerical analysis:
\[
\begin{bmatrix}
0 & 0 & 0 \\
0 & C & 0 \\
0 & 0 & 0
\end{bmatrix}
\begin{bmatrix}
\frac{d}{dt} V_{e1} \\
\frac{d}{dt} V_{e2} \\
\frac{d}{dt} i_{Vs}
\end{bmatrix}
+ 
\begin{bmatrix}
G & -G & 1 \\
-G & G & 0 \\
1 & 0 & 0
\end{bmatrix}
\begin{bmatrix}
V_{e1} \\
V_{e2} \\
i_{Vs}
\end{bmatrix}
= 
\begin{bmatrix}
0 \\
0 \\
V_S
\end{bmatrix}
\]
SPICE

- **Simulation Program with Integrated Circuit Emphasis**
  - Developed at UC Berkeley
  - Coincided with emergence of silicon valley
  - Industry-standard general-purpose circuit simulator

- **Numerical simulation**
  - Modified nodal analysis
  - Trapezoidal or Gear integration method
  - External nodes define boundary conditions

- **Circuit elements represented by device models**

- **Text input, text or graphical output**
Circuit Elements in SPICE

• Resistors
• Capacitors
• Inductors and coupled inductors
• Independent sources (V, I)
• Dependent sources
• Transmission lines
• Switches
• Uniform distributed RC lines
• Active devices
  – Diodes, BJT s, JFETs, MOSFETs, MESFETs
Analysis Modes

• DC analysis (operating points): \texttt{.DC}, \texttt{.OP}, \texttt{.TF}
  – Also before transient and AC
  – Sweep bias points

• AC small-signal analysis: \texttt{.AC}
  – Frequency response
  – Linearized small-signal device models

• Transient analysis: \texttt{.TRAN}
  – Most frequently used in digital circuit simulation
  – Tool to determine time-domain parameters
Analysis Modes (cont.)

• Pole-zero analysis: .PZ
  – Poles/zeros in the small-signal ac transfer function

• Small-signal distortion analysis: .DISTO
  – Steady-state harmonic and intermodulation products for small input signal magnitudes

• Sensitivity analysis: .SENS
  – Sensitivity of DC operating-point or AC small-signal of an output variable with respect to circuit variables

• Noise: .NOISE

• Analysis at different temperature
  – Nominal temperature of 27ºC
A SPICE file is made of a series of statements.
First line is a title statement.
Last line is an end statement.
The order of other statement is arbitrary.
Element statements describe the circuit.
Control statements describe model parameters and execute analyses.
SPICE Syntax

• Each statement is on one line, unless continued with ‘+’ as the first character.
• Each statement consists of fields.
• Fields are separated by , = ( ) or spaces.
• Fields consist of keywords, symbols, names, numbers, or scale-factors.
• Scale-factors:
  – T=1e12, G=1e9, MEG=1e6, K=1e3
  – M=1e-3, U=1e-6, N=1e-9, P=1e-12, F=1e-15
• Comment lines start with * or $.
SPICE File Conventions

- Upper-case letters for SPICE keywords.
- Lower-case letters for user-defined names.
- File structure:
  - Title
  - Parameters
  - Circuit description
  - Input control
  - Analyses
  - Output control
  - Models
  - End
- Comment inserted for documentation.
Circuit Elements

• General syntax
  – NAME node1 node2 … nodeN <model_ref> value
    +=<parameter1=val> < <parameter2=val> …

• Nodes defined by unique numbering (or names in HSPICE).
• Node number 0 (or GND in HSPICE) reserved for ground.
Circuit Elements (cont.)

• Resistor
  – $R_{xxx \ n1 \ n2 \ value}$

• Capacitor
  – $C_{xxx \ n1 \ n2 \ value \ <IC=initial\_condition>}$

• Inductor
  – $L_{xxx \ n1 \ n2 \ value \ <IC=initial\_condition>}$

• Coupled inductor
  – $K_{xxx \ Lyyy \ Lzzz \ value}$
Circuit Elements (cont.)

• Independent voltage/current source
  – $V_{xxx}$ n+ n- <DC> DC_value
  – $I_{xxx}$ n+ n- <DC> DC_value>
  – $V_{xxx}$ n+ n- <<DC> DC_value> AC AC_value +<phase>
  – $I_{xxx}$ n+ n- <<DC> DC_value> AC AC_value +<phase>
Circuit Elements (cont.)

- Dependent sources
  - VCVS: Exxx n+ n- nc1 nc2 value
  - VCCS: Gxxx n+ n- nc1 nc2 value
  - CCVS: Hxxx n+ n- nc1 nc2 value
  - CCCS: Fxxx n+ n- nc1 nc2 value
Circuit Elements (cont.)

- Sinusoidal source
  - $V_{xxx} n^+ n^- \text{ SIN}(v_0 v_a f_o <t_d> <a> <\text{phase}>)$

- Piece-wise linear sources
  - $V_{xxx} n^+ n^- \text{ PWL}(t_1 v_1 t_2 v_2 t_3 v_3 \ldots)$

- Pulse sources
  - $V_{xxx} n^+ n^- \text{ PULSE}(v_0 v_1 t_d t_r t_f t_w t_0)$
Circuit Elements (cont.)

• Diode
  – Dxxx n+ n- <model_name>

• BJT
  – Qxxx nc nb ne <model_name>

• MOSFET
  – Mxxx nd ng ns nb <model_name> L=val W=val
    +<AD=val> <AS=val> <PD=val> <PS=val>…
Circuit Elements (cont.)

- **Subcircuit**
  
  - `Xxxx n1 n2 n3 ... subcircuit_name`

```
Subcircuit Definition
.SUBCKT subcircuit_name n1 n2 n3 ...
  element statements
.ENDS subcircuit_name
```
Output Statements

• .PRINT type ov1 ov2 ov3 ...
• .PLOT type ov1 ov2 ov3 ...

• Node voltage, current through voltage sources
• Type: DC, AC, or TRAN
• Suffix:
  – M: Magnitude
  – DB: Magnitude in dB
  – P: Phase
  – R: Real part
  – I: Imaginary part
HSPICE Specials

• Node names
• .PARAM
  – Define parameters
• .INCLUDE
  – Include external SPICE file
• .LIB
  – Call library entries
• .OPTIONS POST
  – Create graphic plots for AvanWaves/CosmosScope
HSPICE Specials

• .MEASURE
  – Rise, Fall and Time Delay
  – Average, RMS, min, max, peak-to-peak and integral
  – Find X when Y occurs
  – Derivative and Integral Evaluations
  – Equation Evaluations
  – Relative Error

• See HSPICE manuals for more details
Device Models

• A wide variety of active device models
• Process parameter variation
  – Process corners
  – Statistical spread
• Behavioral modeling
SPICE Diode Model

\[ i_D = I_S \left( e^{\frac{v_D}{nV_T}} - 1 \right) \]

\[ C_D = C_d + C_j = \frac{\tau_T}{V_T} I_S e^{\frac{v_D}{nV_T}} + C_j \left( 1 - \frac{v_D}{V_0} \right)^m \]
SPICE Diode Model

- .MODEL model_name D (IS=val N=val RS=val +CJ0=val TT=val BV=val IBV=val)
  - IS saturation current (default=1e-14 A)
  - N emission coefficient (=1)
  - RS series resistance (=0 ohm)
  - CJ0 junction capacitance (=0 F)
  - TT transit time (=0 sec)
  - BV reverse bias breakdown voltage (=inf V)
  - IBV reverse bias breakdown current (=1e-10 A)
Equivalent Circuit for Zener Diode
SPICE Netlist

```plaintext
.SUBCKT zener_diode 1 2
*connections:       | |
* anode |
* cathode

Dforward 1 2 mA_diode
Dreverse 2 4 ideal_diode
Vz0 4 3 DC 4.9V
Rz 1 3 10

* diode model statements
.model mA_diode D (Is=100pA n=1.679)
.model ideal_diode D (Is=100pA n=0.01)

.ENDS zener_diode
```
Example: Voltage Doubler

PARAMETERS:
C1 = 1u
C2 = 1u
VOFF = 0
VAMPL = 10V
FREQ = 1K

(a)
Example: Voltage Doubler
Further Reading

• Sedra & Smith 1.8, 2.9, 3.9
• “The Spice Page”,