Analog Circuit Modeling & Simulation with LTSpice IV

Introduction

- Linear Technologies LT Spice is free.
- SPICE simulator with schematic capture, originally developed to model switcher power supplies.
- 75 circuit examples, 14 assembly examples included.
- All LTC op-amp models
- Many other assemblies, libraries, and circuit models available for free – Google ‘ltspice’
- Good tutorial at Wilfrid Laurier University web site: denethor.wlu.ca/ltspice

PC Board Layout

- LTSpice exports netlist files which may be used for PC board layout and manufacturing, for example, to ExpressPCB, which is also free. However, symbols need to be defined, and this is not trivial.

Examples

- Phase inverter: general modeling technique.
- Voltage-multiplier Power Supply: ideal components and component parasitics
- DG MOSFET Mixer/IF Amplifier: frequency-domain display of transient analysis
- Tube Audio Amplifier: non-standard component models

SPICE Commands

- Analysis (.TRAN, .AC)
- Coefficient of Coupling (Mutual Inductance) K
- .INCLUDE special models (DG MOSFET)
- .param – use {} to replace with values

Parasitics

- Inductor Q
- Voltage Source Resistance
- Crystal motional inductance, holder capacitance, series resistance
- Leakage inductance/coupling coefficient

Phase Inverter

- Start LTSpice, new schematic
- Component->NPN; set to 2N2222
- Resistors; label Re, Rc, Rbe, Rbc
- Wire up resistors
- Grounds
- VCC=12V; Vsig=1V 1kHz
- Capacitors all 1uF

- Ve=3V; Vc=12-(6-3)=9V
- Ic=1mA≈Ie
- Re=Rc=Ic/Vc=3k
- β=100
- I_{Bias} =10*Ic/β=100μA
- Rbe=(3+0.7)/ I_{Bias}=37k
- Rbc=(12-3-0.7)/I_{Bias}=83k
- Transient plot outputs
- AC Analysis 10-100 MHz

Power Supply

- Need 500V @ 20mA
- Only 6.3V and 12.6V secondary transformers are available.
- Winding resistances, and floating nodes.
- With k=.99 quadrupler is << 500V.
- Doubler is better, but even so…
- K (difficult to measure) is very important.
- Use of parameters – increase k to 0.995.

Mixer

- Crystal is really a capacitor which uses L_{series}, R_{series}, and $C_{parallel}$ parasitics.
- Add crystal filter on output: L_M=40 mH; C_M=3.06 pF; R_S=50; C_P=7 pF; RL=600Ω
- Change it to a ceramic filter, L_M=1.23 mH; C_M=100 pF; R_S=50; C_P=15 pF; RL=1500Ω
- Subtract voltages to display differential amplifier output.

Tube Audio Amp

- Show freq response simulation, change C1 from 1 uF to 5 uF to flatten it out.