

Open source CAD tool for electronic and electrical engineers

Yogesh Dilip Save

Indian Institute of Technology, Bombay

October 3, 2012

Motivation

Objective

To develop a tool (open source) which is capable of performing following tasks:

- Circuit Design and drawing
- PCB Layout
- Circuit Simulation (Analog, Digital and Mix mode)

Plan

- Integrate existing open source tools for Circuit Design and drawing, PCB Layout, Circuit Simulation.
- Create additional modules if required.
- Create user friendly graphical interface.
- Spoken tutorials

Modules

- eeschema – Schematic Editor
- CvPCB – Component-Footprint mapper
- pcbnew – PCB Layout Editor
- Kicad to Ngspice netlist converter
- Analysis Inserter
- Component Model Builder
- Component Sub-circuit Builder
- Circuit Simulator – Ngspice
- Scilab based circuit simulator – SMCSim

eeschema

Problems

- No fictitious components (sources)

Sol: Build a library of different kind of voltage and current sources (pulse, sine, exponential etc.)

- Too many components

Sol: Build own libraries (include the components supported by ngspice (explicitly or implicitly)).

Libraries (analogSpice (analog components) and digitalSpice (digital components)) can be built by

- 1 Creating own components.
 - 2 Copying components from existing libraries.
- No measurement modules

Sol: Build a library which gives you functionality of printing and plotting solution.

PCB Layout

CvPCB and pcbnew

- Add footprint for new components.
- Special treatment for measurement modules.

Netlist Converter

- Insert parameters for fictitious components
- Convert IC into discrete blocks
- Insert D-A and A-D converter at appropriate place,
- Insert plotting and printing statements in netlist.
- Find current through all components.

Analysis Inserter

- Insert type of analysis
- Option of analysis
- Option of simulator

Model Editor

- Provides facility to define new model.
 - Diode
 - Bipolar Junction Transistor (BJT)
 - Metal Oxide Semiconductor (MOS)
 - Junction Field Effect Transistor (JFET)
 - IGBT
 - Magnetic core
- Provides facility to edit existing model.
- Provides help related to model parameter.

Sub-circuit Editor

- Provides facility to define new components.
 - Op-amp
 - Timer-IC555
- Provides facility to edit existing sub-circuit.
- Provides help related to components parameters.

Circuit Simulator

Ngspice

- Ngspice is a mixed-level/mixed-signal circuit simulator.
- Code is based on three open source software packages:
 - Spice3f5 – analog circuit simulator
 - Cider1b1 – couples Spice3f5 circuit simulator to DSIM device simulator
 - Xspice – provides code modeling support and simulation of digital components through an event driven algorithm.

Scilab based Mini Circuit Simulator (SMCSim)

Objective

To assist students in improving their knowledge in field of circuit simulation.

Problem with commercial simulators

- Generally software codes are not available.
- Software codes are written in higher level language (C Programming and Fortran....).
- Complex due to implementation of many features and complex modeling.

Motivation

Objective

To assist students in improving their knowledge in field of circuit simulation.

Mini simulator

- used Scilab for coding.
- integrated least number of component.
- different versions for add-on features.

Plan

Display Symbolic Equations

Display Numerical Values

Complete Report Generation

GUI for simulator option

Spoken Tutorial

Plan for releasing version 1.0

- More consolidation of Kicad NgSpice netlist converter (nearly 7 days).
- Build a descent GUI and remove the bugs in integration (nearly 10 days).
- Simulate more examples which cover most of the syllabus of undergraduate in circuit theory (nearly 10 days).