

FOSS EDA Analog Simulation

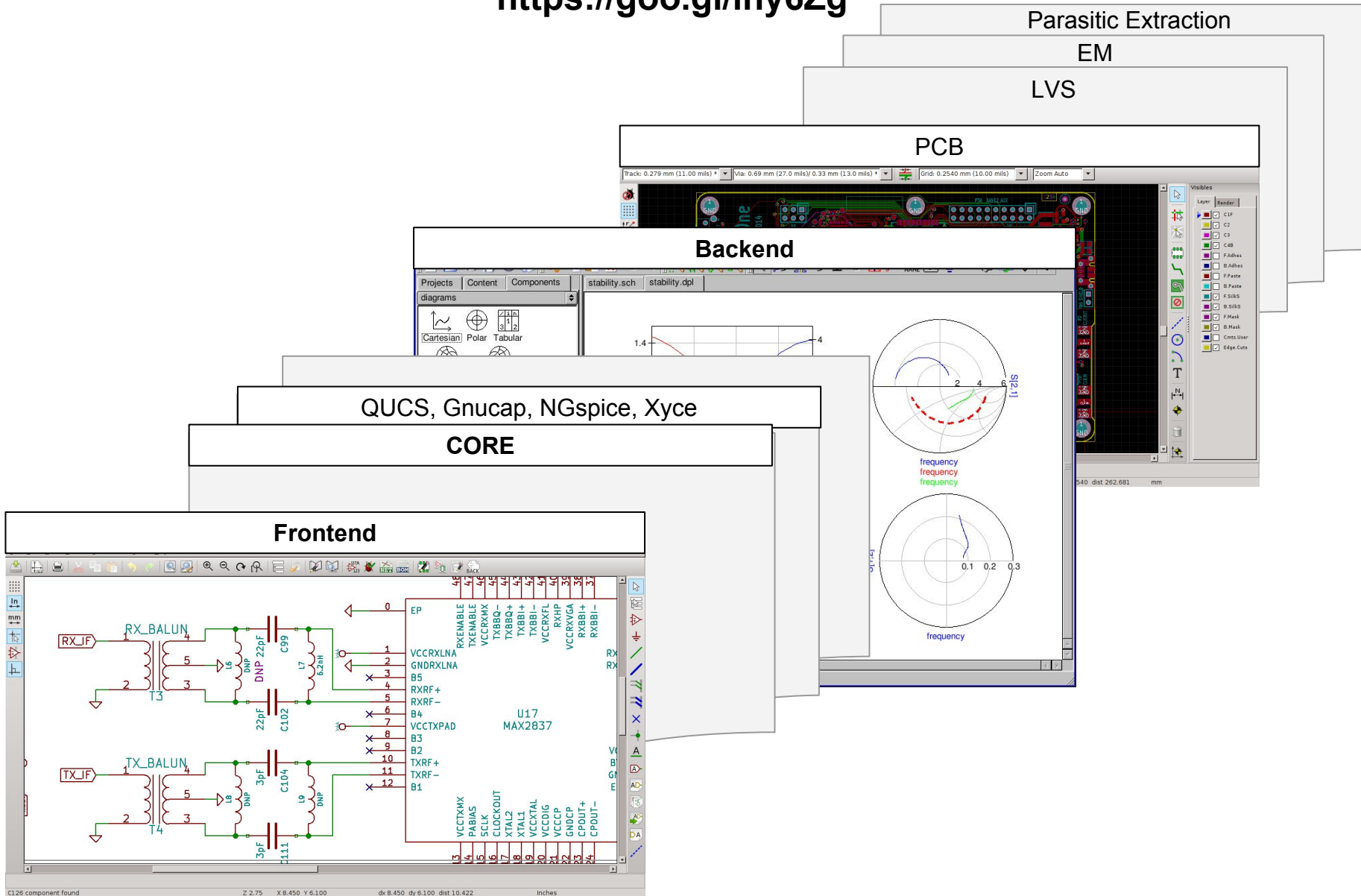
Intercompatibility Matrix

Paolo Nenzi, Francesco Lannutti, Mike Brinson, Guilherme Torri,
Vadim Kusnetsov, Daniel Tomaszewski, Al Davis, Felix Salfelder,
Jason C. Verley, Eric R. Keiter, Wayne Stambaugh, Maciej Suminski,
Tomasz Wlostowski, Javier Serrano, Wladek Grabinski (Speaker)

FOSDEM'17 EDA Devroom; Brussels Feb.5

FOSS EDA Intercompatibility Matrix

<https://goo.gl/iny6Zg>



Frontend:

TOOL			Frontend		
	<i>Sch.Capture</i>	<i>Symbols</i>	<i>Netlist/Parser</i>	<i>UI/scripting</i>	<i>drop-in replacement for</i>
NGSPICE	Kicad,Qucs-S		SPICE3	batch, interactive, ngnutmeg [3]	?
QUCS [13]	Qucs,Qucs-S	variant of XML	Qucs/SPICE3 [5]	batch mode, equations system [6]	
GNUCAP	gEDA (native), verilog (native) [0]		acs, spice, verilog, spectre, gEDA (.sch), qucsator (.net)	interactive cli, command plugins, function plugins [8]	spice, qucsator, spectre
XYCE	gEDA, QUCS-S [14] [15]		SPICE3f5 (PSPICE, HSPICE; not 100% compatible)	batch mode	
SpiceOpus [12]	Qucs-S		Spice3f5	batch, interactive, ngnutmeg [3]	
<i>other...</i>					

CORE:

TOOL			Core	
	<i>model support</i>	<i>Solver</i>	<i>GPU/multicore</i>	<i>simulations</i>
NGSPICE	ADMS-va, XSPICE [4]	SPARSE 1.3 - KLU	GPU - OpenMP	tran, op, ac, pz, linear noise, sensitivity, dc sweep, disto, S-param. could be emulated with nutmeg
QUCS [13]	C++, ADMS-va, EDD [7]	QUCSATOR proof-of-concept solver, for educational purpose	no	op, ac, tran, s-param, HB (limited), noise
GNUCAP	ADMS-va [10], modelgen, bm, spice (binary blobs) [1]	sparse, mixed, post-spice	no demand (yet?)	tran, ac, dc, op, fourier, real-time [9]
XYCE	ADMS-va	sandia labs, industrial stuff	MPI	op, tran, ac, FOUR (Fourier analysis), HB, multi-time PDE, MOR (model order reduction)
SpiceOpus [12]	XSPICE (?)			tran, op, ac, pz, linear noise, sensitivity, dc sweep, disto
<i>other...</i>				

Backend:

TOOL		Backend	
	<i>Visualization</i>	<i>Backannotation</i>	<i>Netlist/Schematic conversion</i>
NGSPICE	Kicad,Qucs-S,Octave[11]	Kicad/interactive	Octave [11]
QUCS [13]	Qucs/Octave, Qucs-S	no	
GNUCAP	gaw, oscopy	[2]	gschem, verilog-a, spectre
XYCE	Qucs-S, support PROBE output (PSPICE [15]), Tecplot	no	no
SpiceOpus [12]	Qucs-S		
<i>other...</i>			

footnotes:

- [0] gnucae aims at a proper schematic/netlist format and translation from/to others
- [1] the intent is, to support more models, e.g. full verilog-ams, systemC(-ams) etc. gnucae supports binary blobs, evades licence issues
- [2] backannotation is trivial if names are consistent between schematic and simulation. e.g. gnucae-geda
- [3] an extended and updated version of nutmeg that includes functions, macros and new commands
- [4] licensing of xspice extension for ngspice seems unclear. does it qualify as "foss"?
- [5] qucsconv is used to convert SPICE into qucsator netlist (SPICE support is limited)
- [6] equation system with several functions (basic math, statistics, matrix, conversions, diff and integration,...) are available for pre- and post-processing
- [7] equation defined device (EDD) is a primitive that when put together allow to build very complex models in a graphical way
- [8] built-in (pluggable) wave processor (fast!), user defined functions. easy to extend (e.g. statistical functions in gnucae-random)
- [9] "simulations" are "commands" are "plugins" and easy to add. sensitivity, ageing transient, and state space inspection are available in gnucae-uf, must be ported.
- [10] gucae-adms does not use the spice-style jacobian-stamping templates
- [11] Octave can control the Ngspice and Qucsator simulation, and obtain data from raw-SPICE3f5 files using the Octave-circuittools package https://github.com/ra3xdh/octave_circuittools/ and convraw CLI utility <https://github.com/ra3xdh/convraw>
- [12] There is no indication (on the website) that spiceOpus is free software. could not even obtain the source code
- [13] not a circuit simulator like the others listed. maybe that should be "qucsator" for consistency
- [14] Xyce is also a supported simulator in the Electric VLSI package (<http://www.staticfreesoft.com>), which is a GPLv3 code
- [15] PSpice (OrCAD), LTSpice = "competition".

Free Computational Electromagnetic Modeling Codes

online source: <http://www.clemson.edu/ces/cvel/modeling/EMAG/free-codes.html>

FAtaiTec Free 2D Field Solver

http://ataitec.com/free_2d_solver/

ATLC - Arbitrary Transmission Line Calculator

<http://atlc.sourceforge.net/>

ATLC2 - Arbitrary Transmission Line Calculator 2

<http://www.hdtvprimer.com/KQ6QV/atlc2html>

emAnalyze

http://www.cemtach.com/index.php?option=com_content&view=article&id=1&Itemid=101

EMAP

<http://www.cvel.clemson.edu/modeling/EMAG/EMAP/>

EMCoS Antenna VLab SV

<http://www.emcos.com/?products=student-version>

EM Explorer

<http://www.emexplorer.net/>

emGine Environment

<http://www.petr-lorenz.com/emgine/>

ERMES

<http://tts.cimne.com/ermes/index.html>

FastCap and FastHenry

http://www.rle.mit.edu/cpg/research_codes.htm

FEKO LITE

<http://www.feko.info/>

FEMM - Finite Element Method Magnetics

<http://www.femm.info/wiki/HomePage>

gprMax

<http://www.gprmax.com/about.shtml>

MagNet (Infolytica)

<http://www.infolytica.com/en/products/trial/>

MMANA-GAL (basic version)

<http://hamsoft.ca/pages/mmana-gal.php>

MEEP

<http://ab-initio.mit.edu/meep/>

MMTL

<http://mmtl.sourceforge.net/>

Multiple Multipole (MMP) Algorithms

<http://alphard.ethz.ch/Hafner/mmp/mmp.htm>

NEC2

<http://www.nec2org/>

NEC2 - the Numerical Electromagnetics Modeling code is a widely used 3D

<http://www.nic.funet.fi/pub/ham/antenna/NEC/swindex.html>

NEC2 uses a text interface. There are several free or inexpensive

<http://www.qsl.net/4nec2/>

newFasant (silver version)

<https://www.fasant.com/editions/silver>

openEMS

<http://openems.de/start/index.php>

Pic2Mag

<http://www.pic2mag.com/>

pdnMesh

<http://pdnmesh.sourceforge.net/>

uma-EM

<https://sourceforge.net/projects/puma-em/>

Qsci

http://boccellengineering.altervista.org/software/software_page.html

SATE Static Field Analysis Toolkit (Educational)

<http://www.fieldp.com/sate.html>

Students' QuickField

http://www.quickfield.com/free_soft.htm

Sonnet Lite

<http://www.sonnetsoftware.com/products/lite/>

Trace Analyzer

<http://www.eecircle.com/downloads/ta.html>

Related FOSS Groups/Websites/Tools

FOSSEE (Free and Open Source Software in Education)

<http://fossee.in/>

gEDA; GPL toolkit of EDA tools

<http://geda-project.org/>

eSim (previously known as Oscad / FreeEDA: <http://freeeda.in>)

<http://esim.fossee.in/>

Free Computational Electromagnetic Modeling Codes

<http://www.clemson.edu/ces/cvel/modeling/EMAG/free-codes.html>

DEVSIM TCAD Semiconductor Device Simulator

<http://www.devsim.org/>

Open Circuit Design Software (including Magic, the VLSI layout editor, extraction, and DRC tool)

<http://opencircuitdesign.com/>

Electric VLSI package

<http://www.staticfreesoft.com>

gSpiceUI

<http://users.tpg.com.au/micksw012/gspiceui.html>

Open Schematic Capture

<http://openschcapt.sourceforge.net>

Open Lab Hardware

EspoTek Labrador

<https://www.crowdsupply.com/espotek/labrador>

Portable Electronics Lab (4 analog channels, two analog outputs, two digital outputs, power supply)

<http://www.nscope.org/explore/>

Analog Discovery 2 100MS/s USB Oscilloscope, Logic Analyzer and Variable Power Supply

<http://store.digilentinc.com/analog-discovery-2-100msps-usb-oscilloscope-logic-analyzer-and-variable-power-supply/>