Simulation Cheatsheet

Harald Pretl, Michael Köfinger
Institute for Integrated Circuits, Johannes Kepler University Linz

This cheat sheet compiles basic simulation and measurement commands, as well a collection of useful code snippets for Ngspice and Xyce. In addition, bind keys and a summary of the mouse operations for Xschem are provieded as well. This compilation relates to Ngspice 42, Xyce 7.8 and Xschem 3.4.5, see:

https://ngspice.sourceforge.io/
https://xyce.sandia.gov/
http://repo.hu/projects/xschem/index.html

A comprehensive documentation and description of Ngspice, Xyce and Xschem can be found at:

https://ngspice.sourceforge.io/docs/ngspice-42-manual.pdf
https://xyce.sandia.gov/files/xyce/Xyce_Users_Guide_7.8.pdf
http://repo.hu/projects/xschem/xschem_man/xschem_man.html

Introductionary notes on Ngspice

Simulation based on the typical SPICE commands, which are preceded by a dot, is known in Ngspice as batch mode. In addition to this basic mode of operation, Ngspice does also offer a so called interactive mode with control file or control section, which allows to either process the results from batch mode, or use all the known simulation commands, measurement statements for scripting. Note that commands in the interactive mode are not preceded by a dot! Lastly, it should be noted, that Ngspice creates a so called *plot* for every simulation run, e. g. *op1, ac1, tran2*. These contain all the respective vectors. In case of a small-signal noise analysis, two plots are generated, the second *plot* contains the integrated results. The inital plot is called *const*. Vectors may be accessed across plots by adding <plotname>. infront of the vector.

Simulation options:		
Ngspice	Xyce	Comment
.temp		
.IC		set initial conditions
.LIB		include a library
.INCLUDE		include part of the netlist
.OPTIONS		
.NODESET		set initial conditions
.PARAM		define parameter
.CSPARAM	–	define parameter(s), also available in control section
.SAVE		save simulation result vector
.PROBE		save device currents, voltages and differntial voltages

Analysis commands: Ngspice	Xyce	Comment
.OP		operating point
.DC		
.AC		small-signal ac
.NOISE		small-signal noise
.TF		transfer function
.PZ node1 node2 node3 node4 cur/vol pol/zer/pz		Pole-zero
.SP		S parameter
.TRAN		transient
.DISTO		distortion
.FOUR		
.PSS		Periodic steady state
.SENS		sensitivity

Measurement snippets (ngspice):

General commands (ngspice):	
`display`	print all vectors of the current plot
`set`	print all plots and set variables
`show`	print all dc operating point model parameters for all device instances
`show` [<*devicename*>]	print dc operating point model parameters of <devicename>

Bind keys (Xschem):

Mouse operations (Xschem):