

MCAST Workshop Manual

1.0 Introduction:

MCAST is a compact model compiler for general devices. MCAST generates device codes automatically for different circuit simulators from the VHDL-AMS description of the compact model. The target simulator used in this workshop is Berkeley's freeware Spice3f5. The MCAST installation files include Spice3f5, so there is no need to download additional files.

A user of MCAST should have a working knowledge of VHDL-AMS and Spice analysis. Useful tutorials can be found at:

http://www.vhdl.org/analog/ftp_files/documentation/tutdac99.pdf, for VHDL-AMS and <http://bwrc.eecs.berkeley.edu/Classes/IcBook/SPICE/>, for Spice3f5.

Proficient users of VHDL-AMS should refer to the appendix, where the limits of MCAST's ability to interpret VHDL-AMS are discussed.

2.0 Quick start:

- 1) Copy spice and device model files to your directory, eg. /homes/demo1
 - a. `cp -r /s0/MCAST/spice3f5 /homes/demo1`
 - b. `cp -r /s0/MCAST/BMACTEST /homes/demo1`

- 2) Setup environment variables
 - a. `setenv BMACROOT /s0/MCAST/BMAC/`
 - b. `setenv SPICEDIR /homes/demo1/spice3f5`
 - c. `setenv SPICEROOT /homes`
 - d. `set path = ($BMACROOT/bin $SPICEDIR/src/bin $path)`

Notes:

 - SPICEROOT is two levels above the SPICEDIR
 - You can either setup in your .tcshrc file or type them in your terminals (need to run "rehash" to active it)

- 3) Three easy steps to compile a compact device model, eg. bmacres.vhdl
 - a. `cd /homes/demo1/BMACTEST/normalDevices/resistor`
 - b. `bmac bmacres.vhdl`
You should see a directory named "gen" created under your current dir.
 - c. `install.spice3f5`
This will install your generated model into spice3f5

- 4) Recompile spice3f5
 - a. `cd $SPICEDIR`

- b. util/build sun4 gcc
- c. Check if any errors. If not, Congratulations! You have successfully add a new compact model into spice3f5 and it is ready to run.

5) Testing your new simulator

- a. cd /homes/demo1/BMACTEST/normalDevices/resistor
- b. Check *.sp files, they are the netlist files for the same test circuits but using different models: one is spice3 built-in model and another is the new model generated by MCAST
- c. spice3 -b rc.sp > result_spice
Simulate using built-in model
- d. spice3 -b rc_bmac.sp > result_bmac
Simulate using auto model
- e. "bmacextract.pl" can be used to extract results from the simulation result to plot in MATLAB
bmacextract.pl result_spice plot_spice
bmacextract.pl result_bmac plot_bmac
You can using MATLAB to plot and compare the results