**Synthesis**

**/PMA/examples/** folder contains mixed-signal digital and analog circuit simulation examples. In addition, a set of tools has been developed by the authors to simplify mixed-signal synthesis, optimization and verification.

As an example, please refer to **64b66b/SERDES64\_4\_1** folder, which contains both digital designs of Parallel-In-Serial-Out (PISO) shift registers synthesized using standard ASIC flow, coming as other parts of the toolkit, as well as custom-designed MCML tree-based SerDes blocks:

* For synthesis and simulation of CMOS circuits in HSPICE environment, please refer to **PMA/cmos\_synth\_and\_sim/** part of the toolkit, while,
* For MCML library optimization, modelling and final simulation of MCML blocks, please refer to **PMA/mcml\_optimization/** and **PMA/examples/** parts of the toolkit.

**Post-synthesis mixed-signal HSPICE simulation**

Once both CMOS and MCML circuits are synthesized, optimized and verified as standalone modules, a mixed digital-analog circuit simulation is possible. Please refer to **/PMA/examples/64b66b/SERDES64\_4\_1** design as an example.

“Digital Signal Injection” tool can be used in hybrid digital-analog circuit HSPICE simulations. The tool automatically generates Spice stimulus waveforms using higher level abstraction file (\*.dsi) format. Please check the DSI tool documentation for signal annotation examples.

In order to run mixed-signal simulation, please use the following steps:

1. Copy synthesized CMOS circuit HSPICE-presented and fully-optimized MCML cell library designs with their instantiations into **hspice.source/** folder inside **SERDES64\_4\_1/** root catalogue;
2. Modify **P6441464.instantiation.spi** file in a way that all CMOS and MCML modules are interconnected to each other; all input interfaces have correct input stimuli signals allocated, and all output interfaces are loaded;
3. Produce appropriate input stimuli via updating test.dsi file, using DSI format. The names inside the input stimuli should be identical to the names used inside instantiation file. The resulting Piecewise Linear Source (PWL) HSPICE input signals are kept in **tempdir.stimulus/** folder and generated when the HSPICE simulation starts. Please refer to **flow\_dsi\_v0p7.pl** Perl script file for details.
4. Once input stimuli file is updated, please, open and update main HSPICE simulation design file - **test.spi**, which defines:
5. HSPICE versions of standard CMOS technology process libraries used; optimized MCML cell library locations.
6. HSPICE simulation options: time of simulation, output signals that need to be probed, location of all library, design and dependency files, including input stimulus and signal variation measurement files;
7. Location of energy-profile measurement files, if produced in custom manner. Otherwise **flow\_pwr\_v0p2.pl** Perl power measurement tool will be invoked.
8. Location of MCML cell library optimization parameters that are usually kept inside a **params\_cells.spi** file.
9. Finally, define voltage supplies for every circuit in use (CMOS and MCML). Please refer to the VDD-level voltage source annotation rules below.
10. Please create symbolic links of all files used in **hspice.source/** folders into the main **SERDES64\_4\_1/** folder.
11. Run HSPICE simulation using **run\_hspice.sh** script file.

Once the simulation is completed the resulting waveforms can be analysed using either Synopsys CosmosScope or an open-source Gwave waveform viewers. Simulation outputs can be also reviewed using **hspice.out** output file.

**VDD Voltage source annotations:** In order for a energy-measurement script (**flow\_pwr\_v0p2.pl**) to produce correct linking to the voltage sources instantiated, a specific voltage source annotation should be used in **test.spi** file.For instance: All MCML power supplies should contain **“vdd\_mcml”** and allCMOS voltage supplies **“vdd\_cmos”** **tags** inside their names.Please refer to **flow\_pwr\_v0p2.pl** file for details about circuit naming.

The described naming rules can be completely ignored if energy-measurements are performed using custom-designed measurement script.