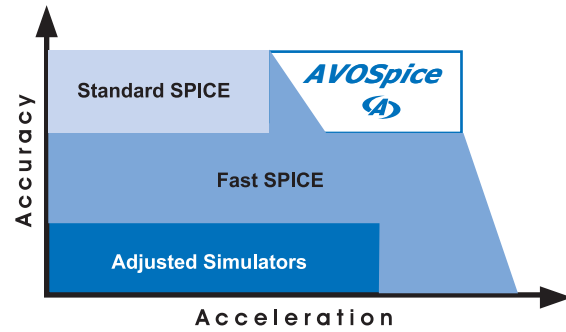


The full-chip circuit simulation is an essential stage for SOC (System-On-Chip) design of high-end integrated circuits (ICs). Only using full-chip simulation you can design various types of SOC circuits. AVOSpice is a simulator which helps engineers to face this challenge.



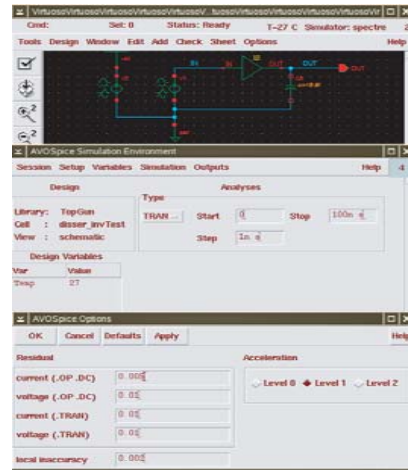
### Advantages:

AVOSpice applies new technology of transient analysis that allows IC designers to reduce time for VLSI circuits simulation substantially. Forthcoming effect lies in reduction in time-to-market for new projects. This can help increase competitive capacity, raise the efficiency of business processes and pave the way to new markets.

AVOSpice can be applied:

- for functional verification of complex digital-to-analog projects such as SOC projects using precise (SPICE) simulation methods;
- for final verification of circuits extracted from layout which contains a great number of parasitic elements;
- for parametric optimization of large electrical circuits fragments.

### Specifications

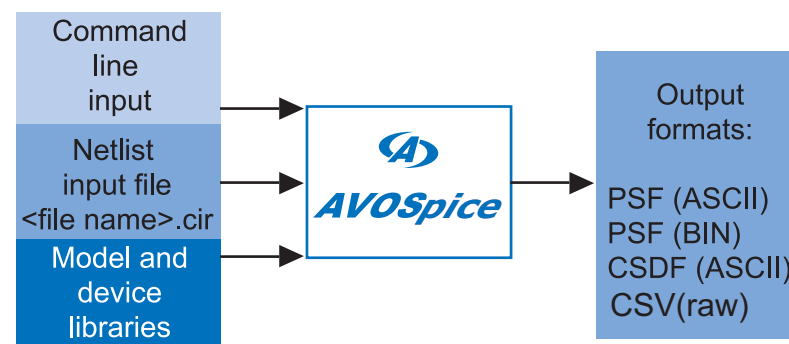
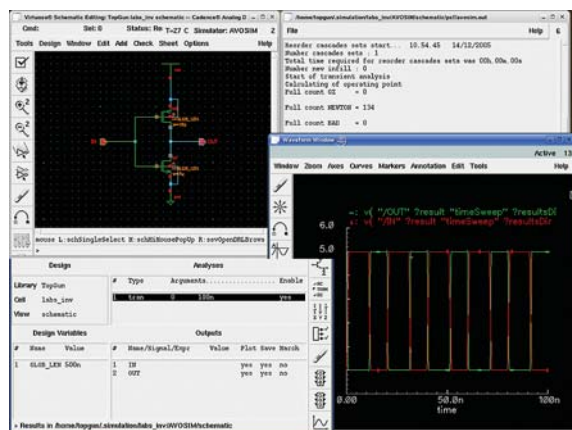


Analyses supported: DC analysis, TRAN analysis, AC analysis and parametric SWEEP for all these analyses.

- Analyses supported in test mode: PSS analysis, S-parameters analysis.
- Models supported: MOS transistor (BSIM3v3.2, BSIM4.40, BSIMSOI3.2), bipolar transistor (Gummel-Poon), resistor, physical resistor, capacitor, inductance, diode, current and voltage sources (DC, EXP, PULSE, SIN, PWL, SFFM, VCVS, VCCS, CCVS, CCCS). All models are HSPICE compatible.
- Additional functions: RC network reduction; multiprocessing (in a test mode).
- Input NETLIST formats: HSPICE and SPECTRE formats.
- Output simulation result formats: PSFBIN/PSFASCII format, CSDF (ASCII) and CSV (raw) formats.
- OS supported: Linux, Solaris, Windows.

### Applications

1. AVOSpice can be launched directly from Cadence ADE (Analog Design Environment). For postprocessing Cadence Results Browser can be used.
2. AVOSpice can be launched from command line. In this case, results are viewed with either Cadence Results Browser or other programs which can visualize PSF, CSDF, or CSV formats.



### Effectiveness

Table 1. Sample circuits simulation time for AVOSpice.

Computer AMD64 Opteron 2.2GHz	Generator -5713- transistors (BSIM3)	PLL -1607- transistors (BSIM3)	ADC10 -4933- transistors (BSIM3)	USB Cable -12141- transistors (BSIM3)	Counter 1000 -64000- transistors
AVOSpice 4 core	3h 16min	50min	1h 18min	2h 41min	3min 12sec
AVOSpice 1 core	8h 08min	2h 13min	3h 02min	6h 46min	6min 58sec
SPICE simulator 1 Core	2days 14h	7h 09min	9h 53min	2days 4h	8days
Multicore acceleration	19 times faster	7 times faster	9 times faster	20 times faster	3840 times faster

Full advantage of AVOSpice is obtained for very large-scale ICs where the majority of signals are staying unchanged during a long time: the more elements - the higher efficiency.

### RC-reduction

Usually RC-reduction is used for final verification of circuits extracted from layout. The size of such circuits can be reduced dramatically. For example, digital circuit X[16]^2 which is automatically synthesized after layout extraction consists of about 300.000 elements. After RC-reduction the number of elements reduces down to 30.000. With manually designed digital-to-analog circuit the effect might be less. To activate RC-reduction the designer should put a few additional directives to the netlist:

```
*@ PRS_OPTIONS fold-all
*@ PRS_R_MODELS res_ad
*@ PRS_PWR_NODES VSS gnd
```

### RF simulation

In the nearest future AVOSpice will allow to perform RF analysis. Periodic Steady State (PSS) and S-parameters analyses are already available in a test mode.

### Multiprocessing

During last few years manufacturers of multicore/multiprocessing systems increased their business activity. Running AVOSpice under two core system you might accelerate your computational process up to 250% additionally. For some types of simulations acceleration will increase proportionally to the number of processors (cores).

### Success story

- Examples of commercial projects successfully designed with AVOSim:
  - Analog-to-Digital Converters 3bit, 6bit, 8bit, 10bit, 16bit (pipeline, delta-sigma);
  - USB1.1 interface;
  - PLL 2-100MHz;
  - Optical Mouse.

To make sure that your circuits can be successfully simulated with AVOSpice you can send us test circuit and we'll simulate it for you. Please address any questions to [support@avocad.com](mailto:support@avocad.com)

You can download trial version of AVOSpice from our WEB site [www.avocad.com](http://www.avocad.com) to see how AVOSpice integrates into your design flow.