TechModeler

Device Modeling for New Technologies

SILVACO

Today engineers who develop organic transistors, OLED, solar cells or other new devices face a new challenge. Unlike the silicon based technology, where engineers use existing, standarized SPICE compact models to fit the behaviour of their transistors, no physical model is able to fit the behaviour of these new devices. Developing physical models is very complex and time consuming. TechModeler adresses this issue through an innovative compact modeling approach. TechModeler modeling solution brings this missing piece and fills the lack of physical models thanks to its unique compact modeling technology.

Key Features

- Fully automated Model generation
- Semi-physical modeling
- Batch mode available
- Input sampling to explore customer data
- Incrementally optimized database and modeling
- Famous EDA formats model export for circuit simulation (Encrypted Verilog-A, C library)
- Drastic reduction of costs and time-to-market

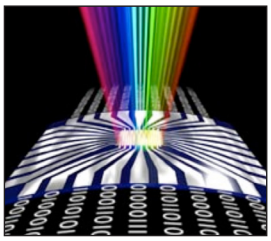
Benefits

- Innovative patented technology
- Modeling from measurements in minutes
- Unique advanced models validation
- Generalization even in high dimension with strong coupled variables
- Preserving accurate extrapolation properties
- Parameterized models for simulation
- Encrypted models for IP preserve
- Global data exploration, all space covered
- Constraint multi-objective on input and output parameters

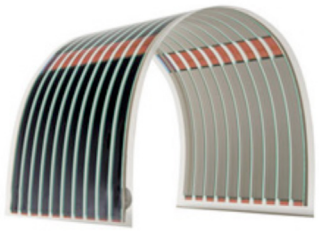
Typical Applications

- Organic transistors, OLED, solar cells, sensors, memory…

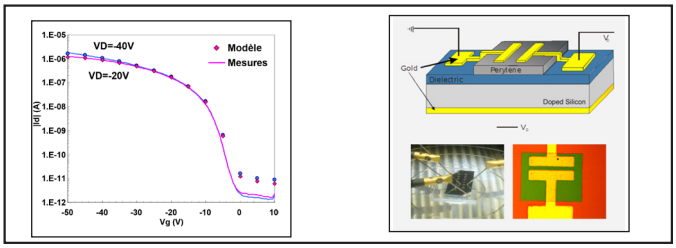
Sensors



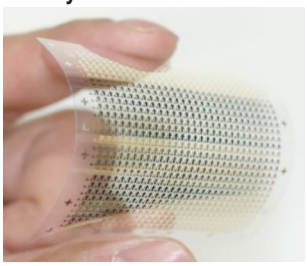
Photovoltaic Cells



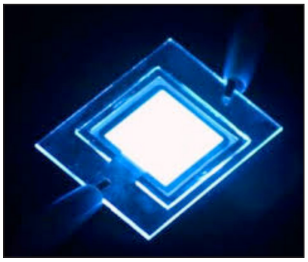
Transistors



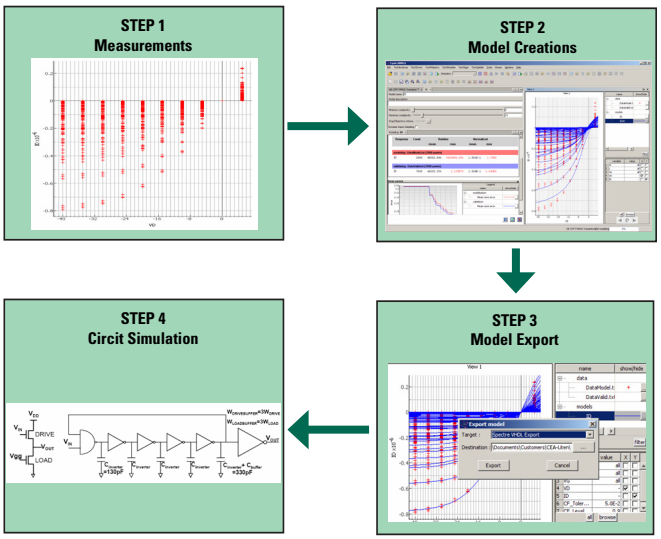
Memory



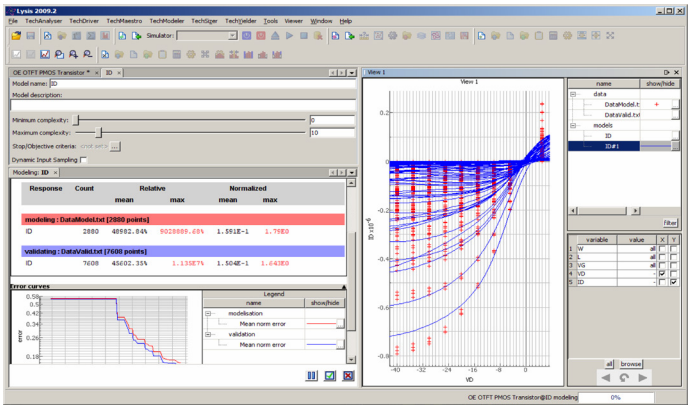
OLED



TechModeler Modeling Flow



A Typical Application – OTFT Modeling



From the measurements made on the transistor prototypes, transistor models have been created.

Model parameters are:

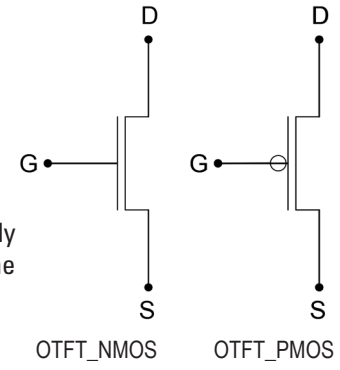
Name	Definition	Min Value	Max Value
W	Channel width	500 µm	12000 µm
L	Channel length	10 µm	100 µm
Nf	Number of fingers	1	10
VDS	Drain Source voltage	-50	50
VGS	Gate Source voltage	-50	50

The model accuracy is above 99% (which means less than 1% difference between model and data).

Once the models are created, they are exported to the spice simulator, SmartSpice, Eldo, Spectre or HSPICE. They are provided as an encrypted Verilog-A and a C library files.

The transistor is represented as a sub-circuit with three connection ports:

- D: Transistor drain
- G: Transistor gate
- S: Transistor source

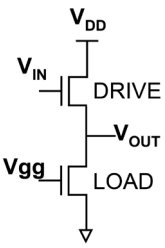


OTFT_NMOS OTFT_PMOS

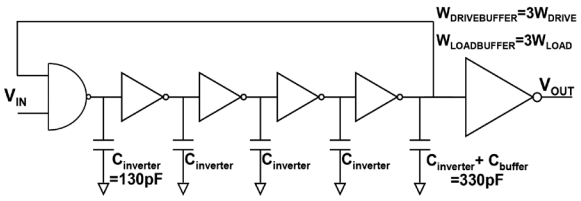
To use these transistors, simply connect them to the rest of the circuit:

```
X1 VDD VGG VSS OTFT_PMOS W=800U L=50U NF=2
```

These transistors models have been used for several circuit simulations and the simulations have been compared with measurement done on prototypes:

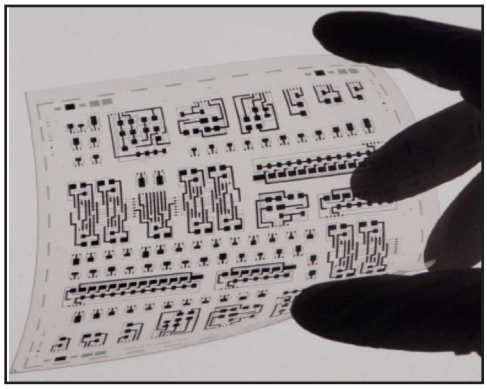


P-Type Inverter

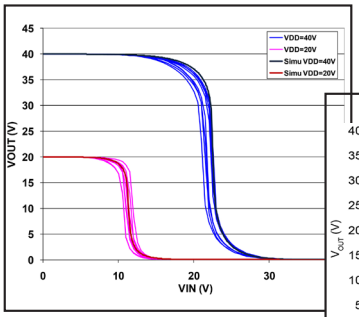


P-Type Ring Oscillator

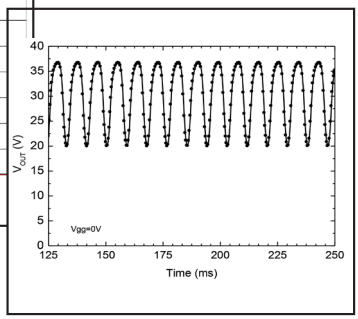
The prototypes of the circuits have been fabricated and then measured:



The comparisons between measures and simulation show good compatibility:



Comparison for an inverter (DC operation)



Comparison for an oscillator (Transient operation)

SILVACO
HEADQUARTERS
4701 Patrick Henry Drive, Bldg #23
Santa Clara, CA 95054

NORTH AMERICA sales@silvaco.com
BRAZIL br_sales@silvaco.com
EUROPE eusales@silvaco.com

JAPAN jpsales@silvaco.com
KOREA krsales@silvaco.com
TAIWAN twsales@silvaco.com
SINGAPORE sgsales@silvaco.com
CHINA cn_sales@silvaco.com

WWW.SILVACO.COM

Rev 042820_02