

# T-Spice Pro™

The Windows® solution for mixed-signal IC design

Tanner EDA's powerful T-Spice Circuit Simulator generates fast and accurate simulations of analog and mixed-signal IC designs.

Support for over 40 models including:

## FET Models

Berkeley BSIM3SOI 3.1 User Manual  
Berkeley BSIM3SOI 3.2 New Features  
Berkeley BSIMSOI 4.0 User Manual  
Berkeley BSIM3 3.3 User Manual  
Berkeley BSIM4 4.5 User Manual

## EKV Model Equations

Philips MOS 9  
Philips MOS 11  
Philips MOS 20  
Philips MOS 30  
Philips MOS 31  
Philips MOS 40  
Philips PSP  
RPI a-Si TFT 15  
RPI Poly-Si TFT 16

## Bipolar Models

Philips Mextram  
Philips Modella  
VBIC Fundamentals  
VBIC Model Equations

## Diode Models

Philips Diode 500  
Philips Juncap

Additional models available on request.

For more information:  
[www.eda-solutions.com](http://www.eda-solutions.com)



from software to silicon

