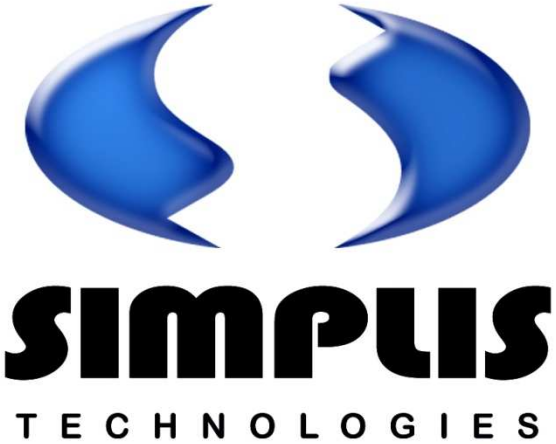
What is the difference between SIMPLIS and Spice?



SIMPLIS

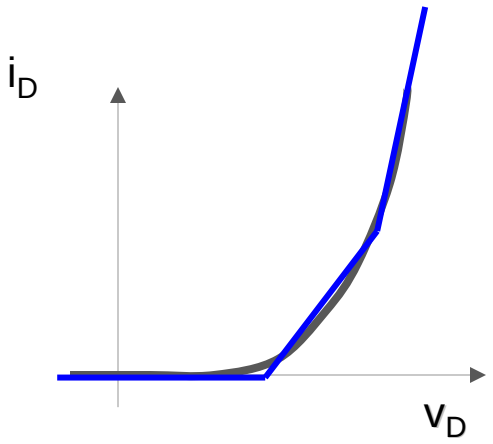
- SIMPLIS uses piecewise linear (PWL) analysis and modeling techniques
- All nonlinearities are modeled with piecewise linear approximations
- At any instant in time, the system is a linear network
- That linear network can change radically at any time

PWL analysis of SIMPLIS facilitates

- Very fast transient simulations
  - 10 to 50 times faster than Spice
- Very fast determination of steady-state
  - Periodic Operating Point analysis
    - No need for long transient simulations to reach steady-state
- Very fast AC analysis on full nonlinear time-domain switching model
  - Same procedure as in lab
    - No need to derive separate average model

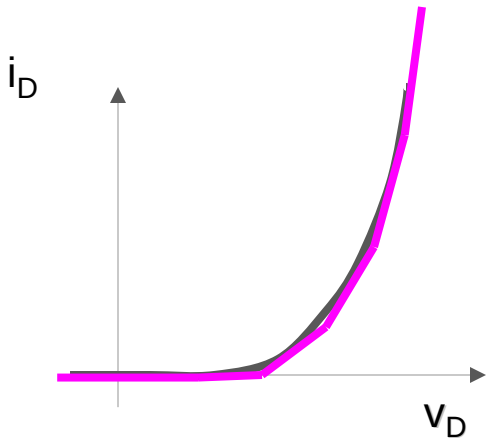
… for highly nonlinear systems such as switching power supplies

Piecewise linear modeling



PWL → Exponential expressions are approximated by a series of straight line segments

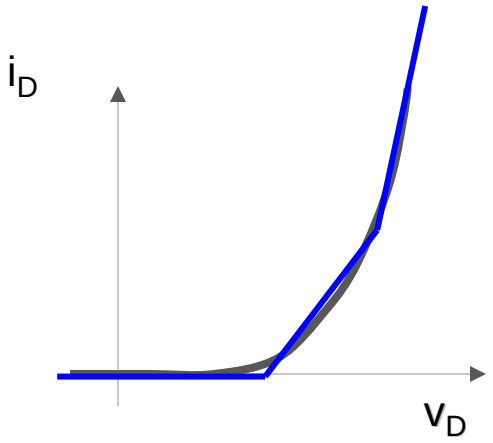
Piecewise linear modeling



PWL → Exponential expressions are approximated by a series of straight line segments

More straight line segments yield higher accuracy and slower simulation times

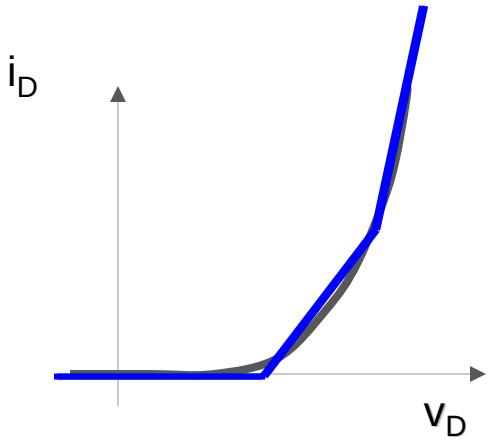
Piecewise linear modeling



PWL → Exponential expressions are approximated by a series of straight line segments

If a switching system spends very little time in the area where the device models are less accurate, often the system behavior can be accurately simulated despite the piecewise linear approximations.

Piecewise linear modeling



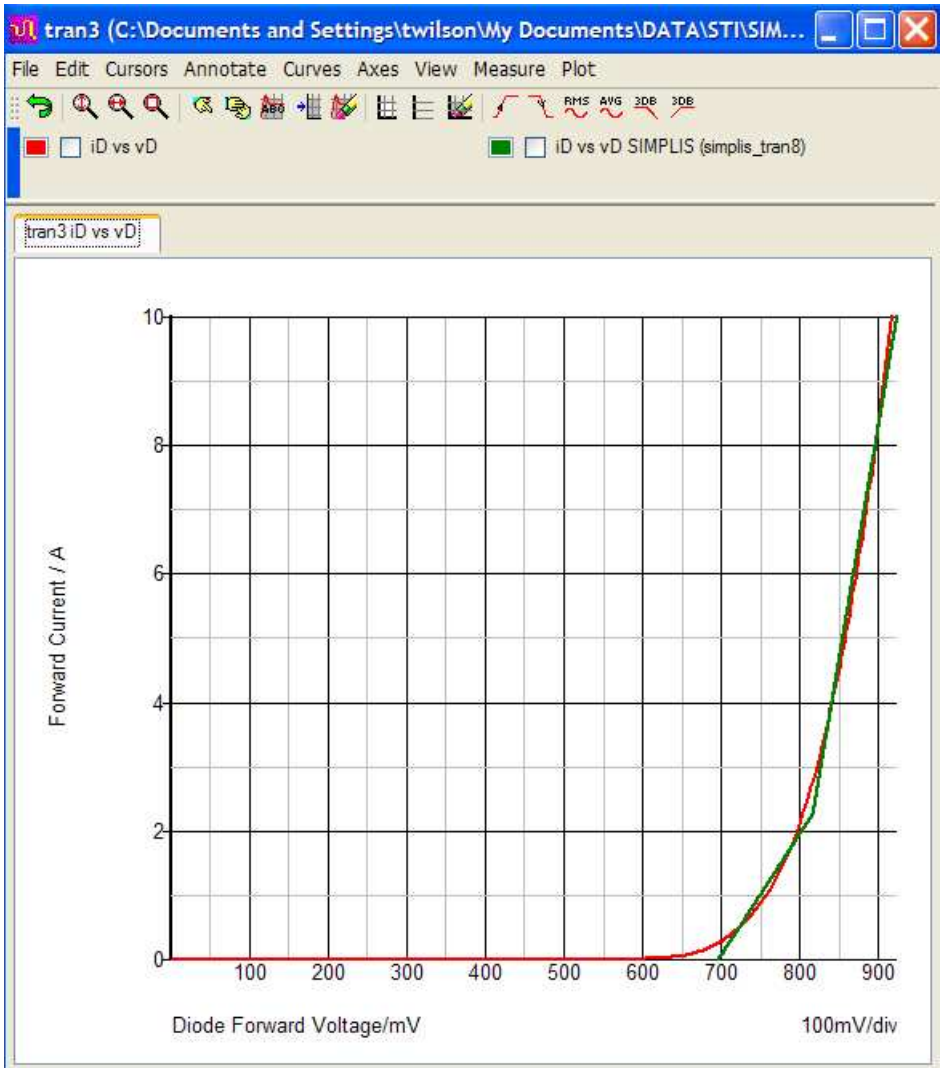
PWL → Exponential expressions are approximated by a series of straight line segments

Goal: Achieve desired accuracy of results in shortest simulation time

If a switching system spends very little time in the area where the device models are less accurate, often the system behavior can be accurately simulated despite the piecewise linear approximations.

Diode PWL Model

- Three-segment SIMPLIS diode model (green)
- Compared to Spice model (red)
- Fit optimized over 4A – 8A range



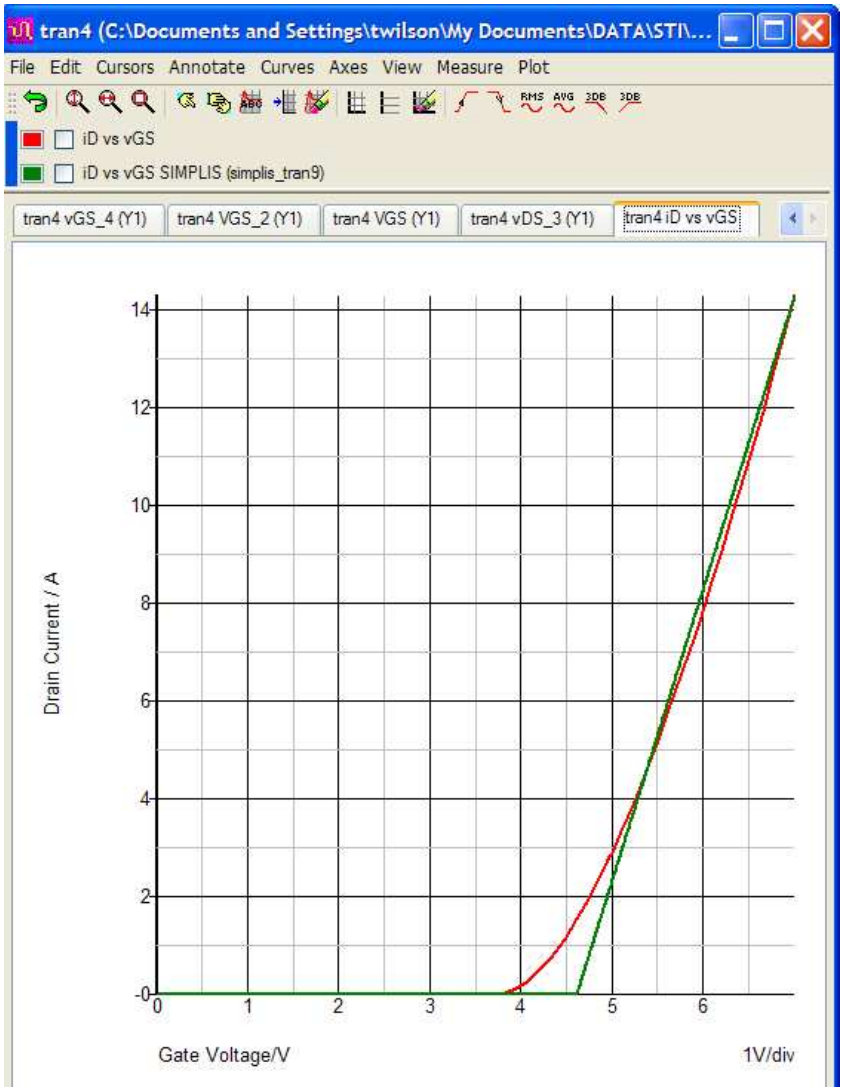
SIMPLIS TECHNOLOGIES

Simulation Software for Power Electronics

Component Design • Circuit Design • System Design

FET PWL Model

- Two-segment SIMPLIS iD vs. vGS (green)
- Compared to Spice model (red)
- Fit optimized over 33% - 100% of Imax



SIMPLIS TECHNOLOGIES

Simulation Software for Power Electronics

Component Design • Circuit Design • System Design

Model Accuracy

- **Accuracy of Physics:** the Physics behind the equations that describe semiconductors
  - Spice - very accurate
  - SIMPLIS – piecewise linear behavioral approximations
- **Accuracy of Parameters:** the value of the constants used in the model
  - Spice – mere mortals cannot typically create their own Spice models
  - SIMPLIS – models are based on data sheet info and can be created by average user in ~20 minutes
- **Accuracy of solution of circuit equations:**
  - Spice – in order to get practical speed, Spice must accept more numerical error (RelTol) per computational step
  - SIMPLIS – takes advantage of piecewise linear system to get extremely accurate numerical solutions of circuit equations

SIMPLIS

- **Because SIMPLIS can get extremely accurate numerical solutions to the circuit equations:**
  - **SIMPLIS can solve numerically for the steady-state limit cycle of a closed-loop switching power supply very quickly**
    - % change in all state variables from beginning to end of one conversion cycle < **10^{-10} %**
  - **SIMPLIS can perform a small signal analysis on the full large signal model of the power supply** (no need to derive a separate averaged model)

When SIMPLIS? When Spice?

- Spice is optimized for:
  - Circuits that can be linearized around a DC operating point
    - e.g. Op amp circuits, …
  - IC circuits where Spice models can relate device parameters to process parameters

- SIMPLIS is optimized for switching systems that have:
  - Very fast switching transients and relatively large amounts of energy storage
    - e.g. Closed-loop switching power supplies, Switched capacitor filters, Phase-Locked Loops

SIMPLIS makes Virtual Prototyping Feasible

- For closed-loop switching power supplies,
  - **Vastly superior speed** for transient simulations
    - Typically 10X to 50X faster than Spice
  - **Vastly superior convergence** properties
    - Convergence ceases to be an ever-present concern
  - Ability to quickly find **steady-state** and do **AC analysis** for circuits that have a stable steady-state limit cycle using only the large signal time-domain model – ***no derivation of an average model required***

… make Virtual Prototyping Feasible