



## Chapter 1

# Introducing Star-Hspice

---

The Star-Hspice optimizing analog circuit simulator is Avant!'s industrial-grade circuit analysis product for the simulation of electrical circuits in steady-state, transient, and frequency domains. Circuits are accurately simulated, analyzed, and optimized from DC to microwave frequencies greater than 100 GHz.

Star-Hspice is ideal for cell design and process modeling and is the tool of choice for signal-integrity and transmission-line analysis.

This chapter covers the following topics:

- [Star-Hspice Applications](#)
- [Star-Hspice Features](#)

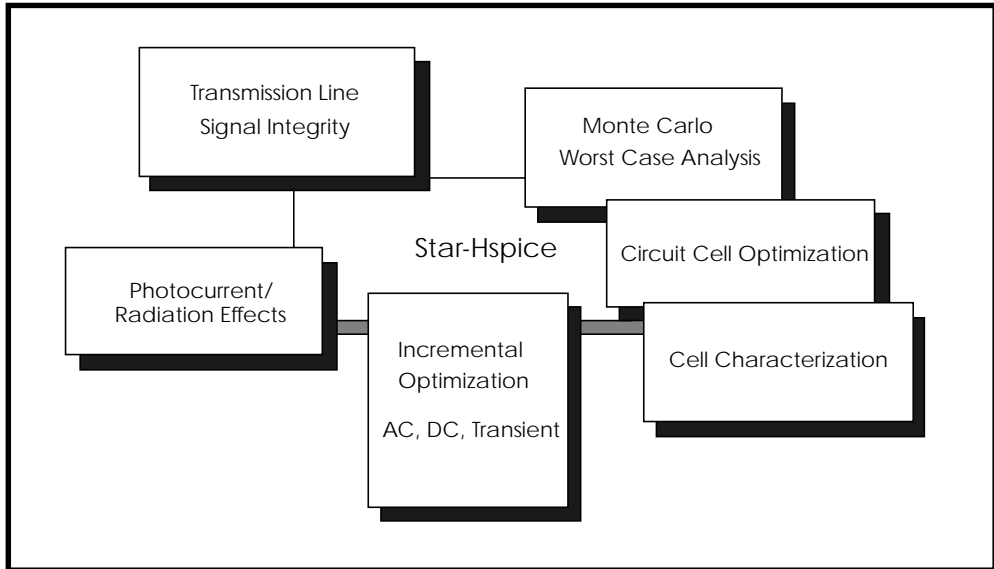
---

## **Star-Hspice Applications**

Star-Hspice is unequalled for fast, accurate circuit and behavioral simulation. It facilitates circuit-level analysis of performance and yield utilizing Monte Carlo, worst case, parametric sweep, and data-table sweep analysis while employing the most reliable automatic convergence capability. Star-Hspice forms the cornerstone of a suite of Avant! tools and services that allow accurate calibration of logic and circuit model libraries to actual silicon performance.

The size of the circuits simulated by Star-Hspice is limited only by the virtual memory of the computer being used. Star-Hspice software is optimized for each computer platform with interfaces available to a variety of design frameworks.

## Star-Hspice Features

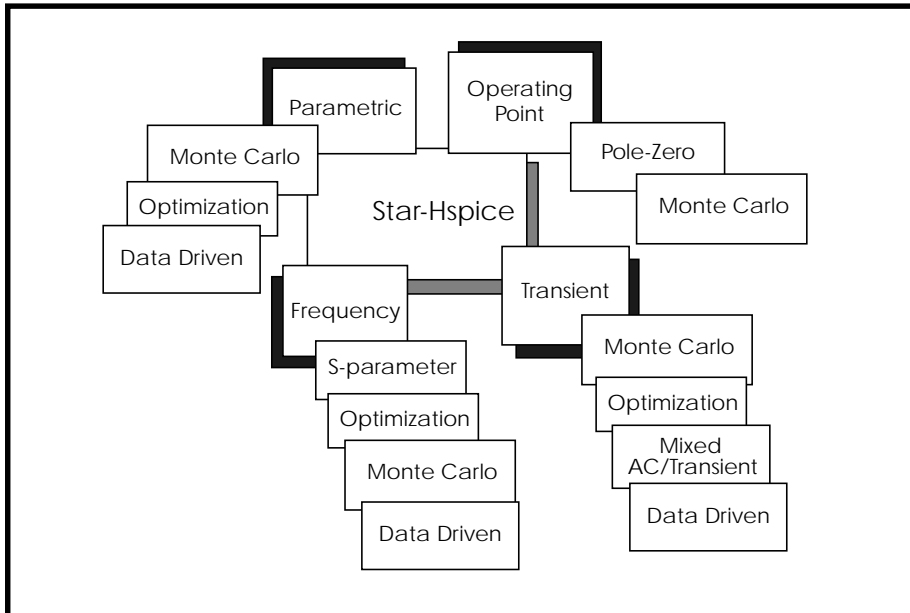


**Figure 1-1: Star-Hspice Design Features**

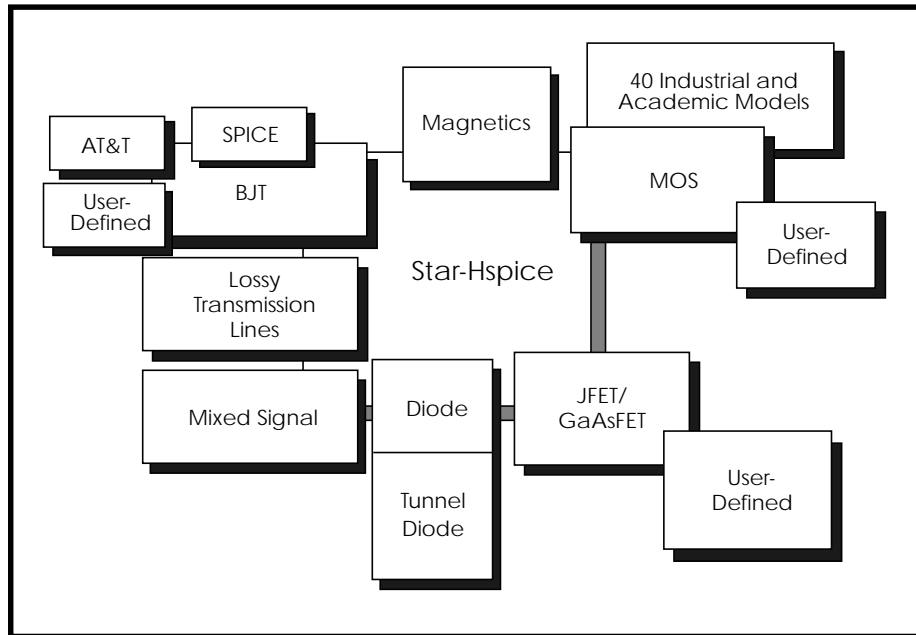
Star-Hspice is compatible with most SPICE variations, and has the following additional features:

- Superior convergence
- Accurate modeling, including many foundry models
- Hierarchical node naming and reference
- Circuit optimization for models and cells, with incremental or simultaneous multiparameter optimizations in AC, DC, and transient simulations
- Interpreted Monte Carlo and worst-case design support
- Input, output, and behavioral algebraics for parameterizable cells
- Cell characterization tools for calibrating library models for higher level logic simulators

- Geometric lossy coupled transmission lines for PCB, multi-chip, package, and IC technologies
- Discrete component, pin, package, and vendor IC libraries
- AvanWaves interactive waveform graphing and analysis from multiple simulations



**Figure 1-2: Star-Hspice Circuit Analysis Types**



**Figure 1-3: Star-Hspice Modeling Technologies**

Simulation at the integrated circuit level and at the system level requires careful planning of the organization and interaction between transistor models and subcircuits. Methods that worked for small circuits might have too many limitations when applied to higher level simulations.

You can organize simulation circuits and models to run using the following Star-Hspice features

- Explicit include files – .INC statement
- Implicit include files – .OPTION SEARCH='lib\_directory'
- Algebraics and parameters for devices and models – .PARAM statement
- Parameter library files – .LIB statement
- Automatic model selector – LMIN, LMAX, WMIN, WMAX model parameters
- Parameter sweep – SWEEP analysis statement
- Statistical analysis – SWEEP MONTE analysis statement

