主辦 | Graser 協辦 | cadence

Getting Confidence
in Your Design Early!

Graser User Conference

Taipei

2016 7.14

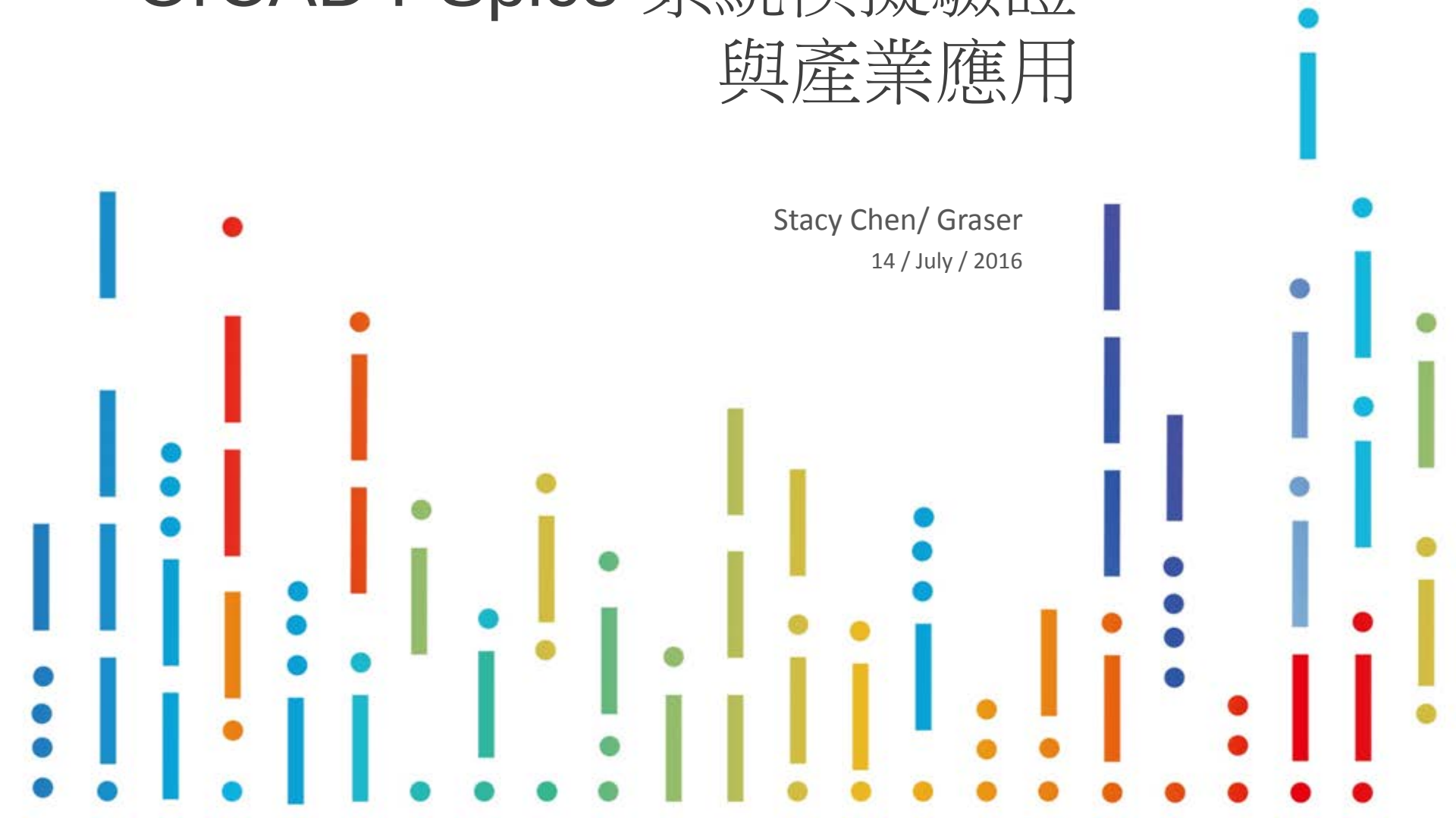
跨界整合 翻轉設計

Graser User Conference Taipei

OrCAD PSpice 系統模擬驗證與產業應用

Stacy Chen/ Graser

14 / July / 2016



OrCAD PSpice System Solution and Industry Application

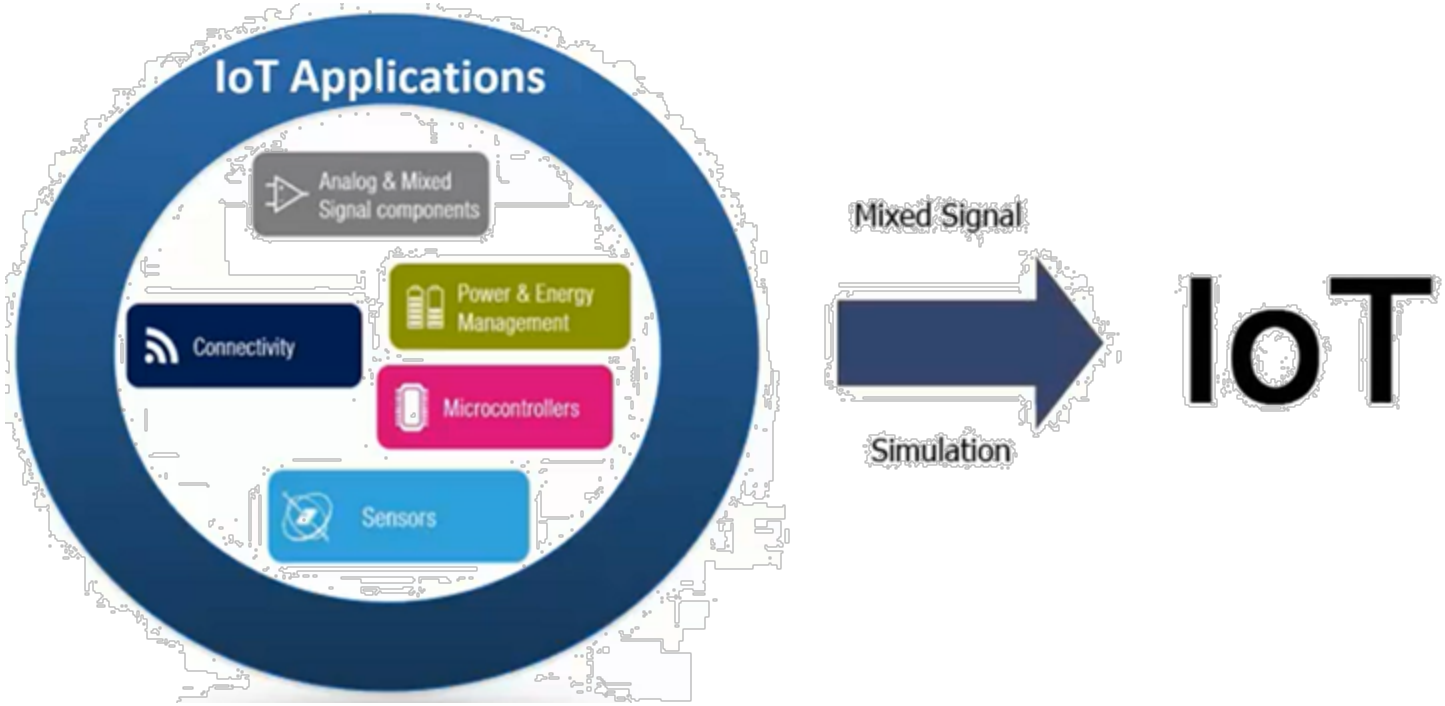
- OrCAD® PSpice® Technologies
- OrCAD PSpice Virtual Prototyping Systems
- OrCAD PSpice Advanced Analysis
- Industry Application

Graser User Conference Taipei

OrCAD PSpice Technologies

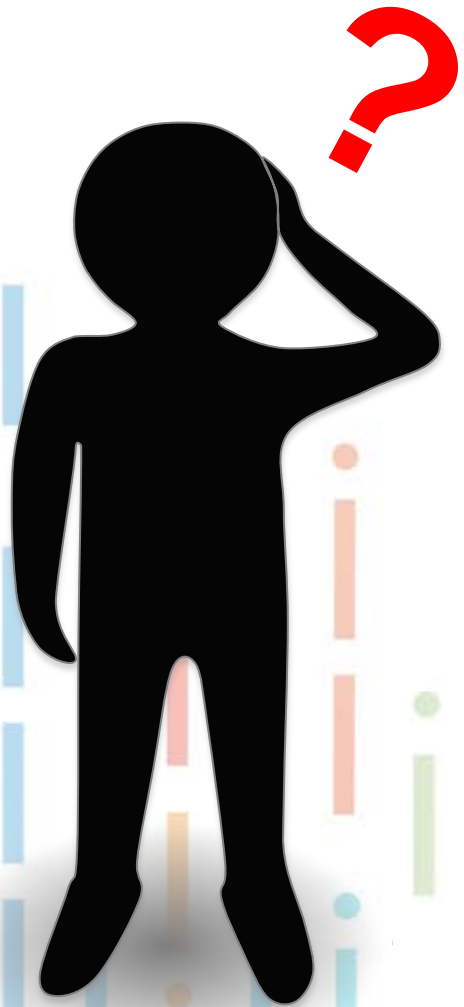
Internet of Things

Internet of Things



Challenges

Graser User Conference Taipei



Reduce errors

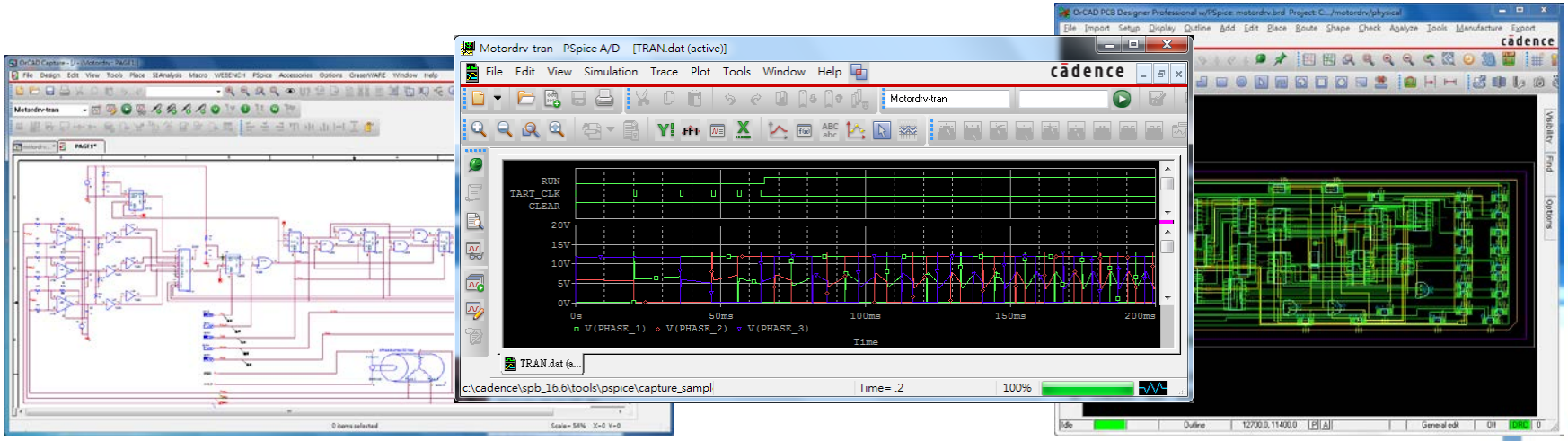
Increase reliability

Model development

Increase Design Productivity

OrCAD Design Solution

- Powerful and Widely Used Design Solution
  - Front-to-Back Integration
    - Fast and intuitive schematic design entry, OrCAD® Capture
    - **Mixed-signal simulator, OrCAD® PSpice®**
    - Comprehensive PCB solution, OrCAD® PCB Designer



Schematic
OrCAD Capture

Simulation
OrCAD PSpice

Layout
OrCAD PCB Designer

Graser User Conference Taipei

OrCAD PSpice

- Mixed-signal simulator
  - Full integration with OrCAD® Capture improve productivity and data integrity.
  - Powerful waveform viewing and post-processing expression support speed review and analysis without having to rerun simulations.

Graser User Conference Taipei

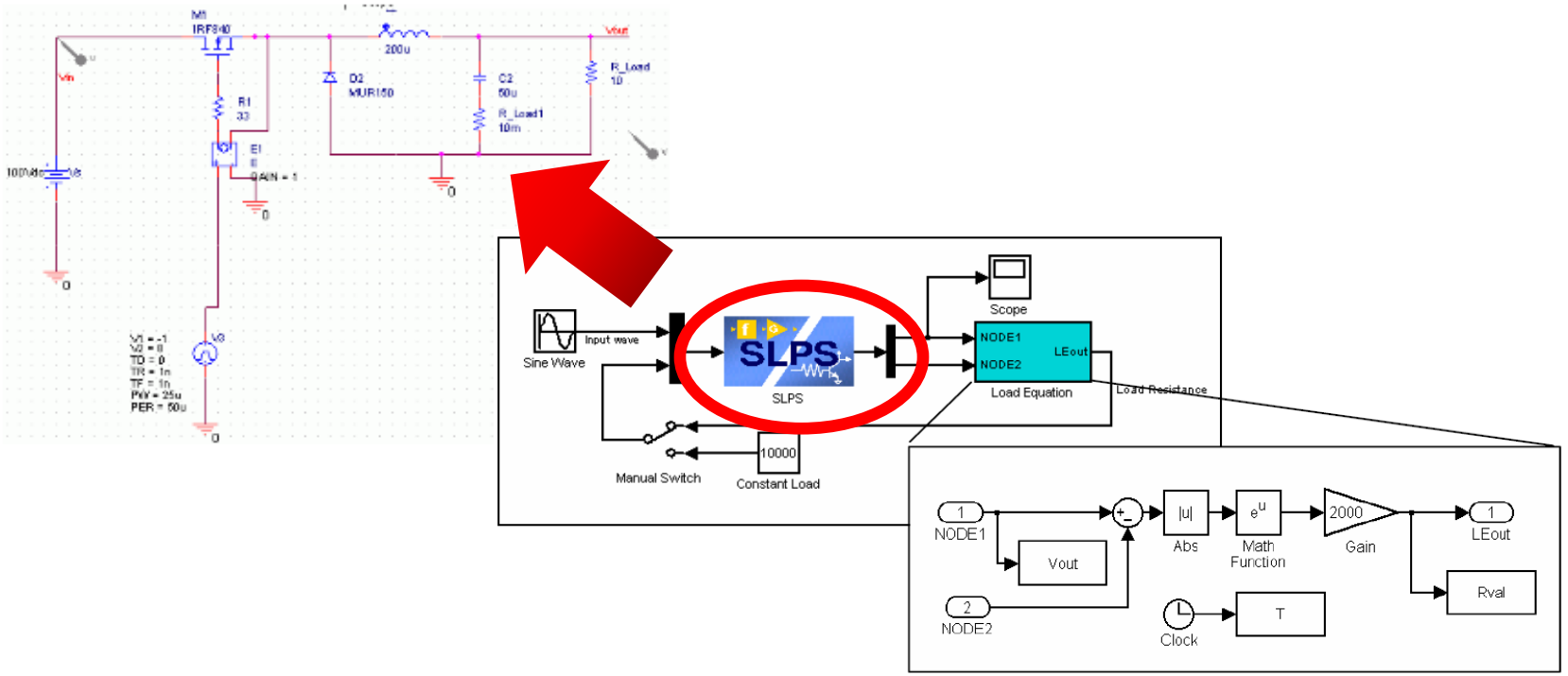
OrCAD PSpice

- Mixed-signal simulator
  - Multi-vendor models, built-in mathematical functions, and behavioral modeling techniques.
  - Support multi-modeling type,
    - Algorithmic Models: Matlab/C/C++
    - System Models: SystemC
    - Digital Models with IO/Timing/Constraint
    - Digital Function Model
    - Verilog-A
    - PSpice® Behavioral Models

Graser User Conference Taipei

OrCAD PSpice Mixed Domain Integration

- OrCAD® PSpice® w/ Matlab
  - Systems Simulation
  - Integration with MATLAB Simulink brings two industry-leading simulation tools, electromechanical systems & electrical, in a co-simulation environment.

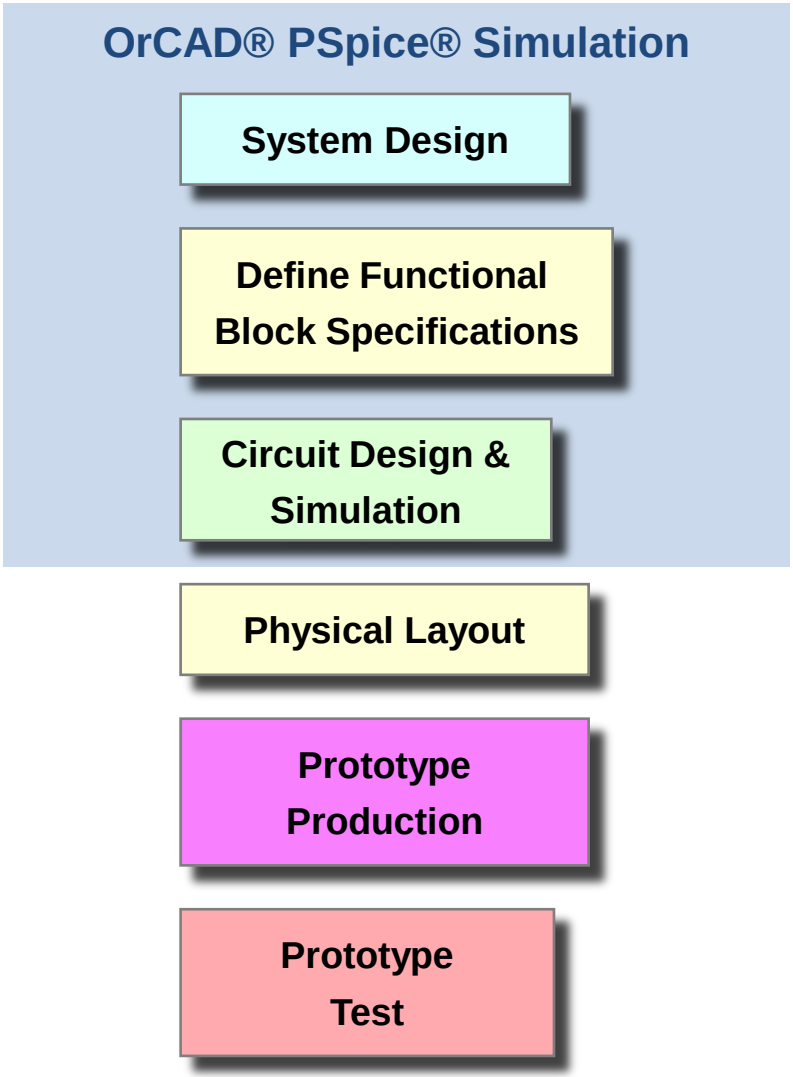


Graser User Conference Taipei

Graser User Conference Taipei

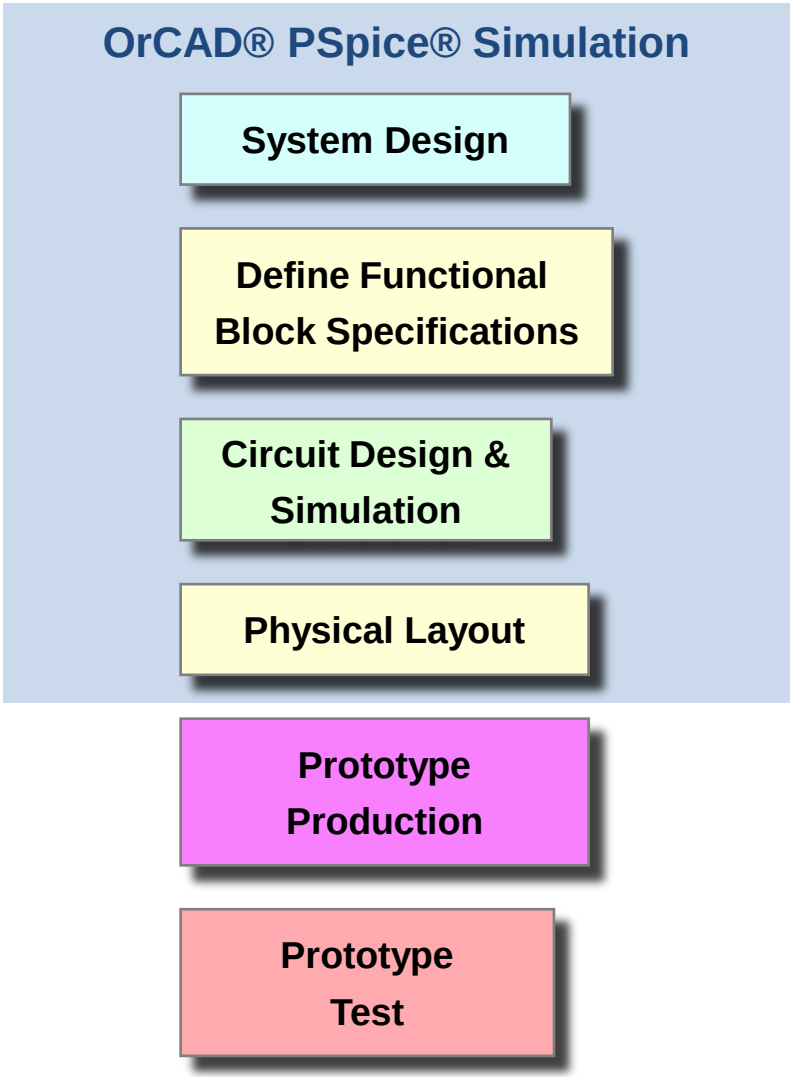
OrCAD PSpice
Virtual Prototyping Systems

OrCAD PSpice System Simulation Solution Flow



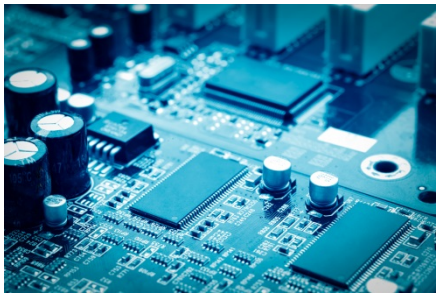
Graser User Conference Taipei

OrCAD PSpice System Simulation Solution Flow

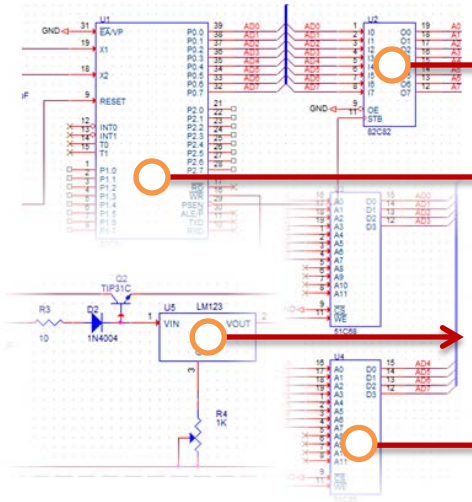


Graser User Conference Taipei

OrCAD PSpice Virtual Prototyping Systems



Components with different levels Abstraction Modeled in PSpice PCB simulation



PSpice Analog Behavioral

SystemC with Embedded S/W

PSpice Spice MacroModel

PSpice Functional Block defined in C

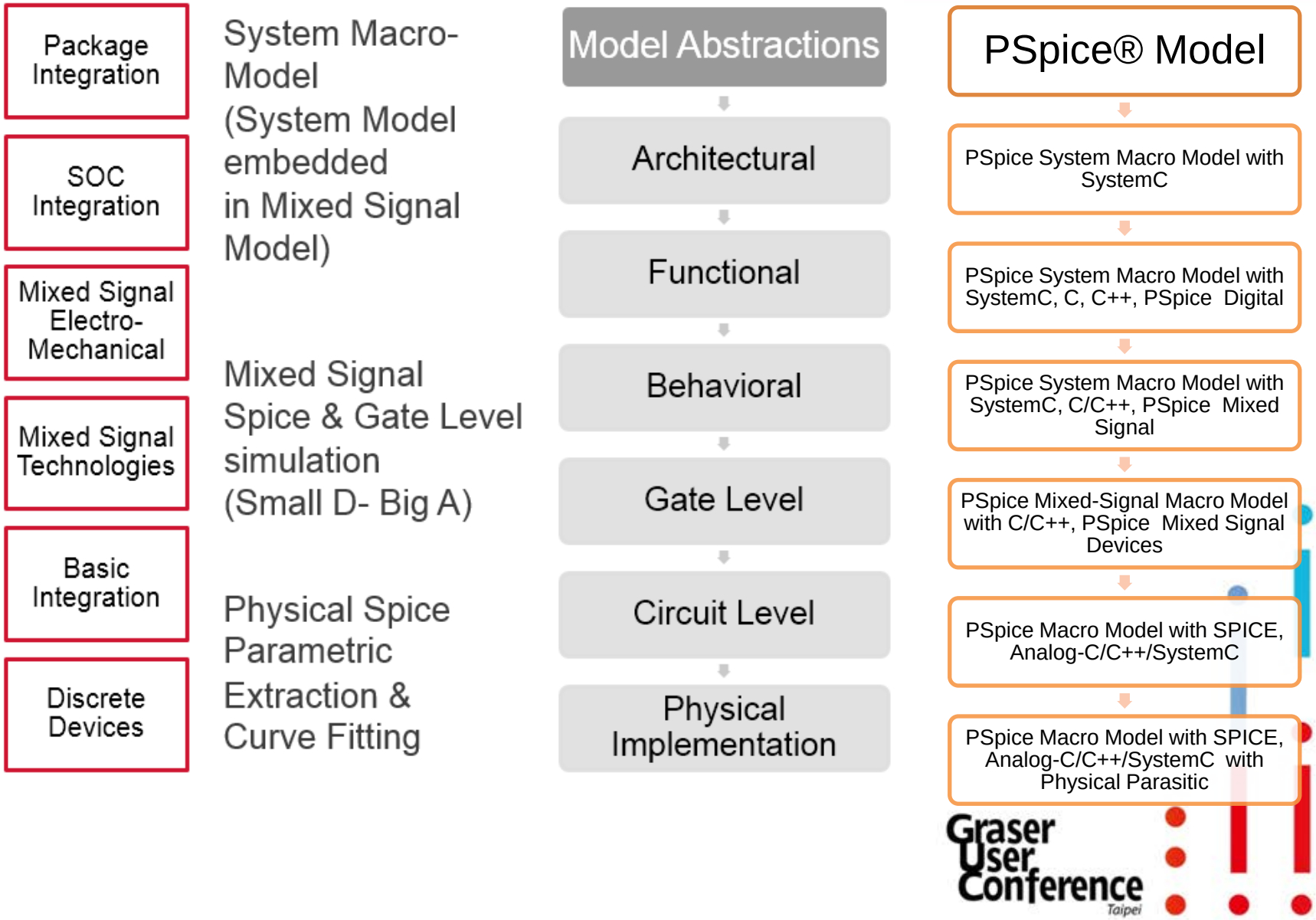
Model Abstractions

- Architectural
- Functional
- Behavioral
- Gate Level
- Circuit Level
- Physical Implementation

- **Multi-Level Block Abstractions**

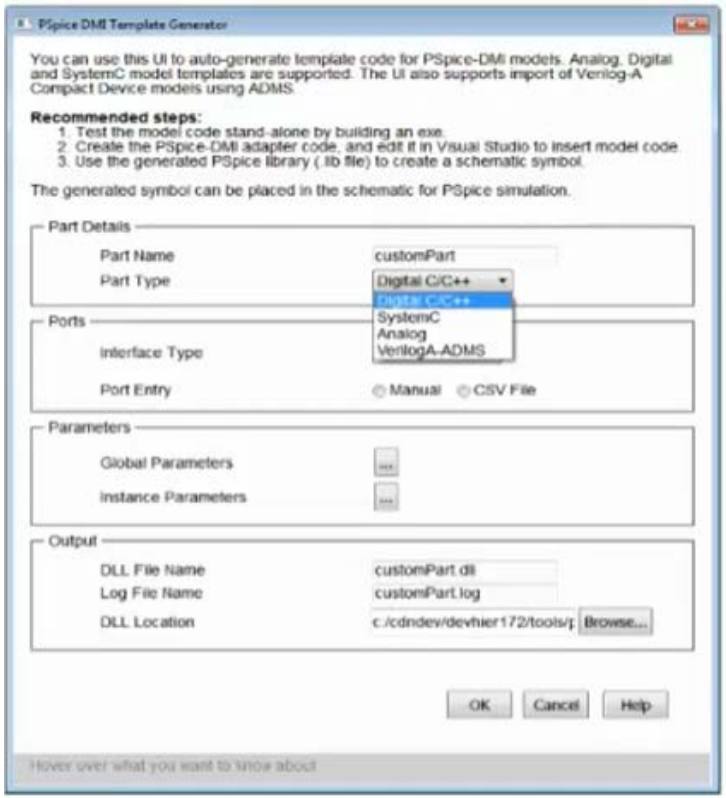
Graser User Conference Taipei

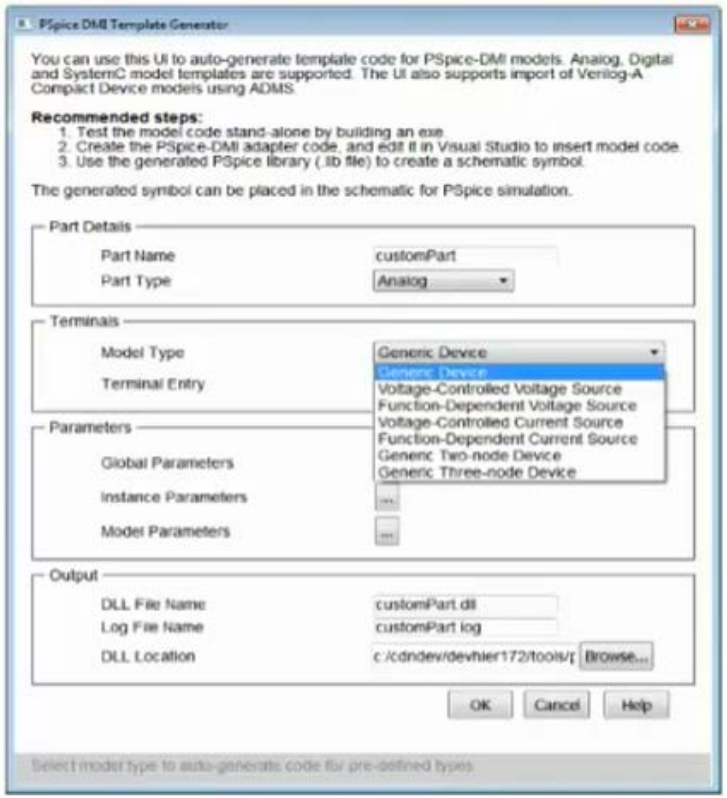
OrCAD PSpice Virtual Prototyping Systems



OrCAD PSpice Device Model Interface

- Generate PSpice® Adaptor code in PSpice Model Editor
  - Analog, Digital C/C++, SystemC and Verilog-A.



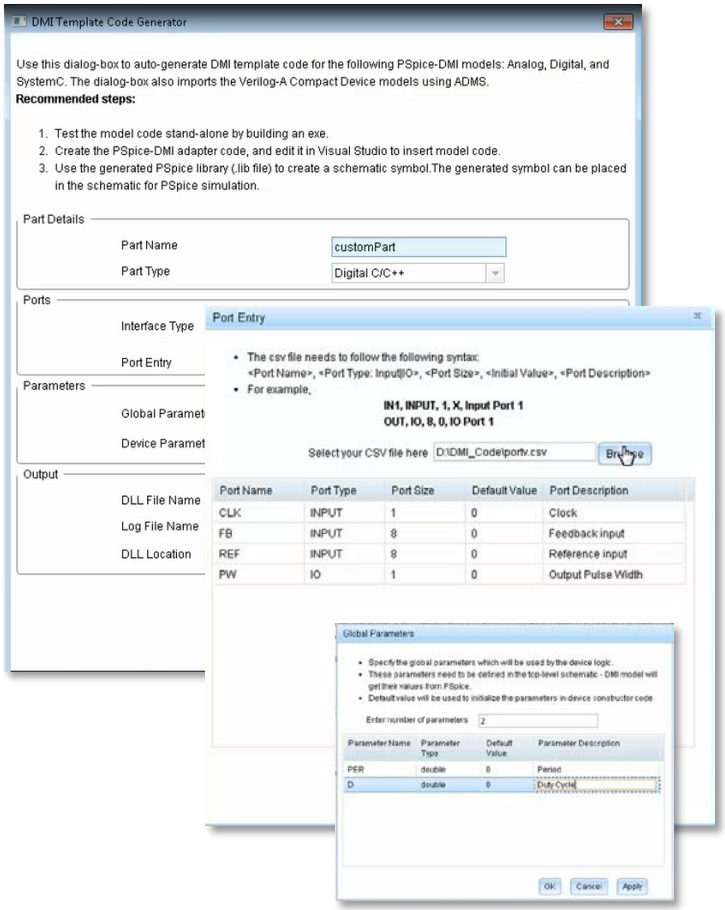


Graser User Conference Taipei

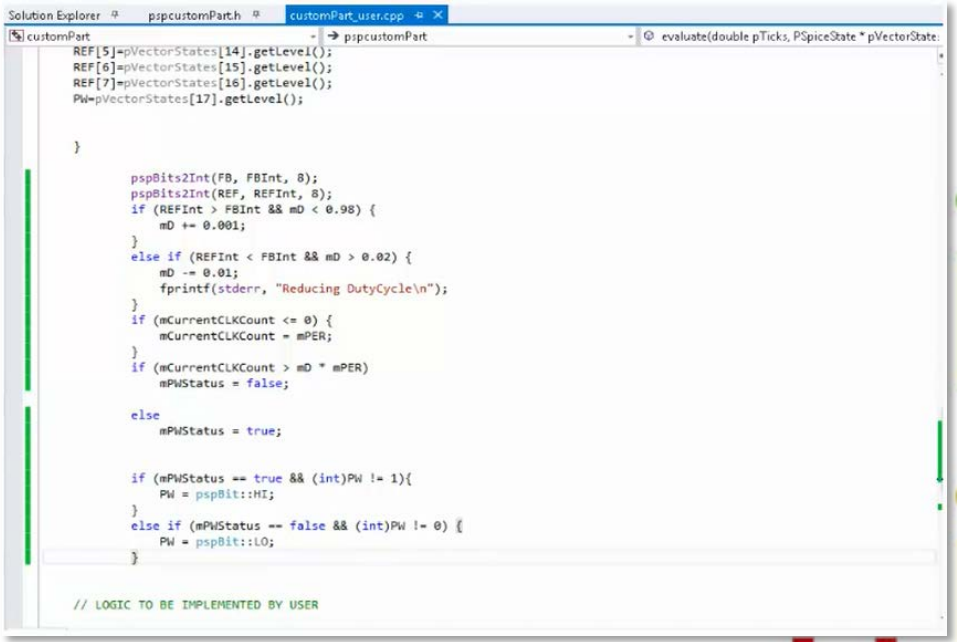
OrCAD PSpice Simulation

- Create Model

PSpice® DMI



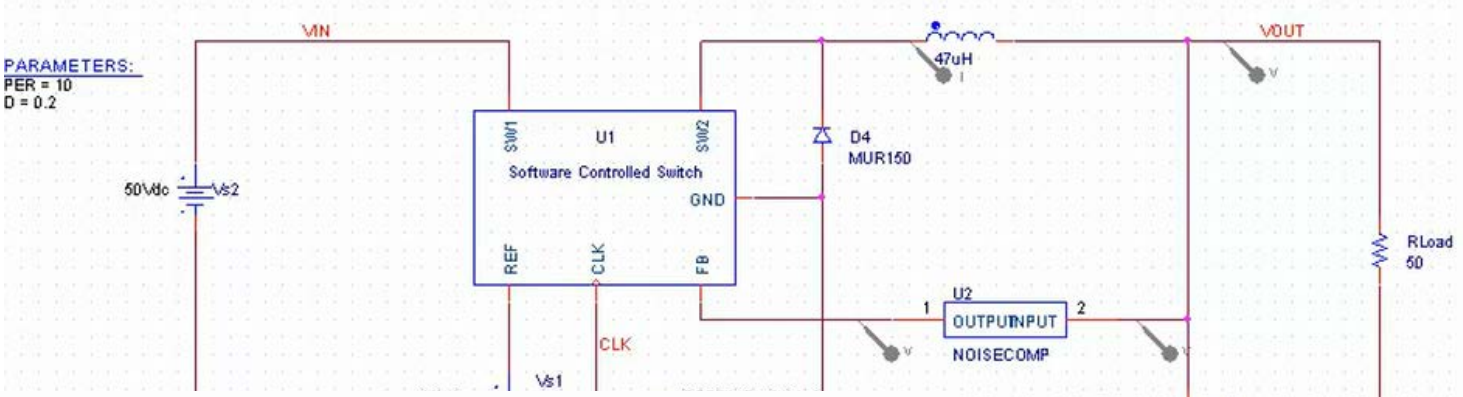
Microsoft Visual Studio Community 2013

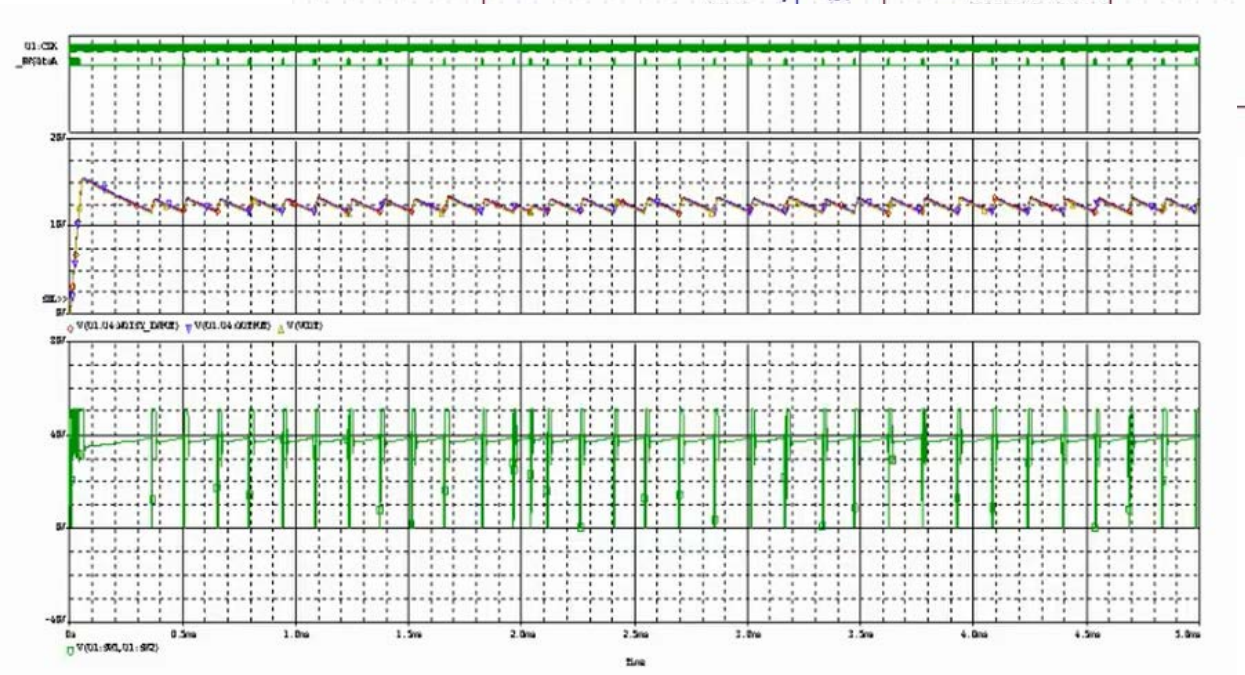


Graser User Conference Taipei

OrCAD PSpice Simulation

- Result





Graser User Conference Taipei

Graser User Conference Taipei

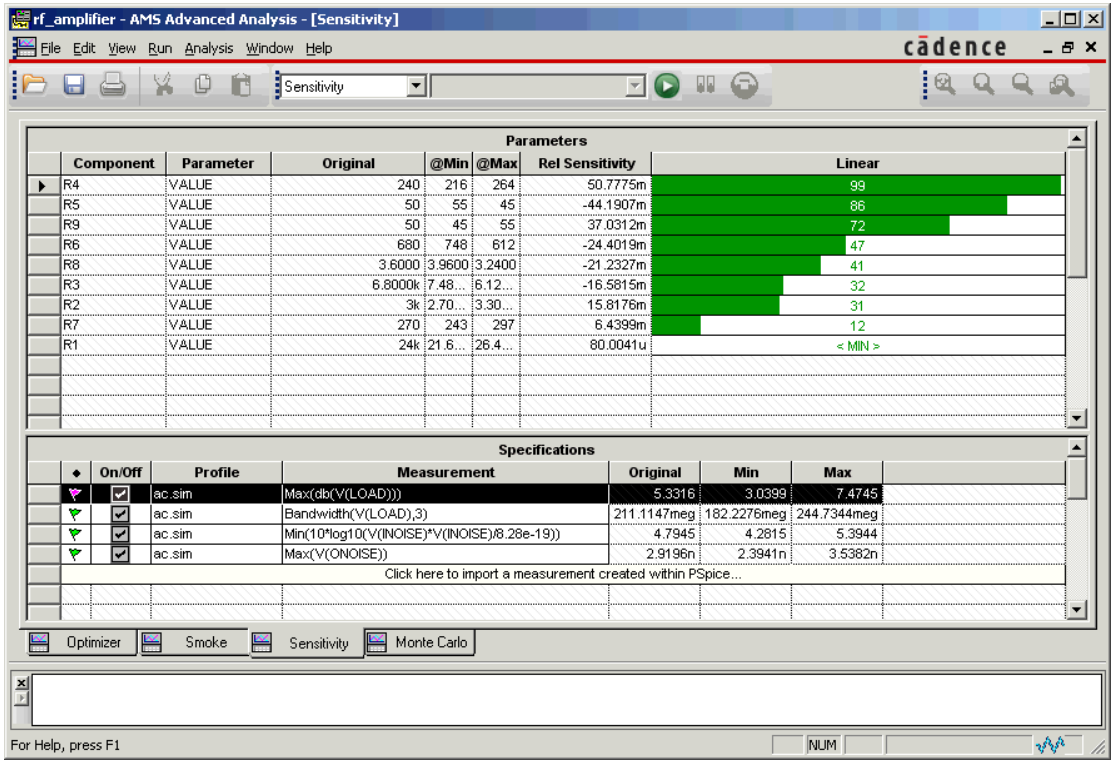
OrCAD PSpice Advanced Analysis

Powerful Advanced Analysis

- Combine OrCAD® PSpice® A/D functionality with the powerful Advanced Analysis environment.

Sensitivity Analysis

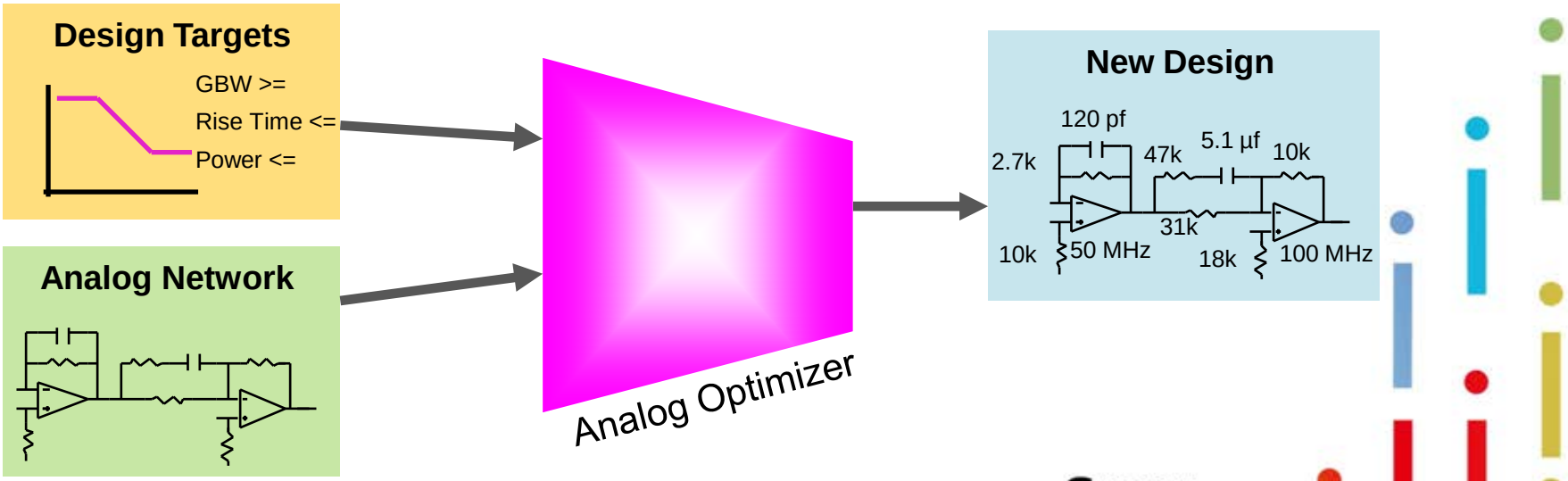
- Easily identify components impacting key circuit goals and specs.
- Estimate worst case performance of the circuit given the device tolerances.
- Identify components whose tolerance does not matter.
  - **Reduce cost** by choosing components with relaxed tolerance.



Graser User Conference Taipei

Circuit Optimization

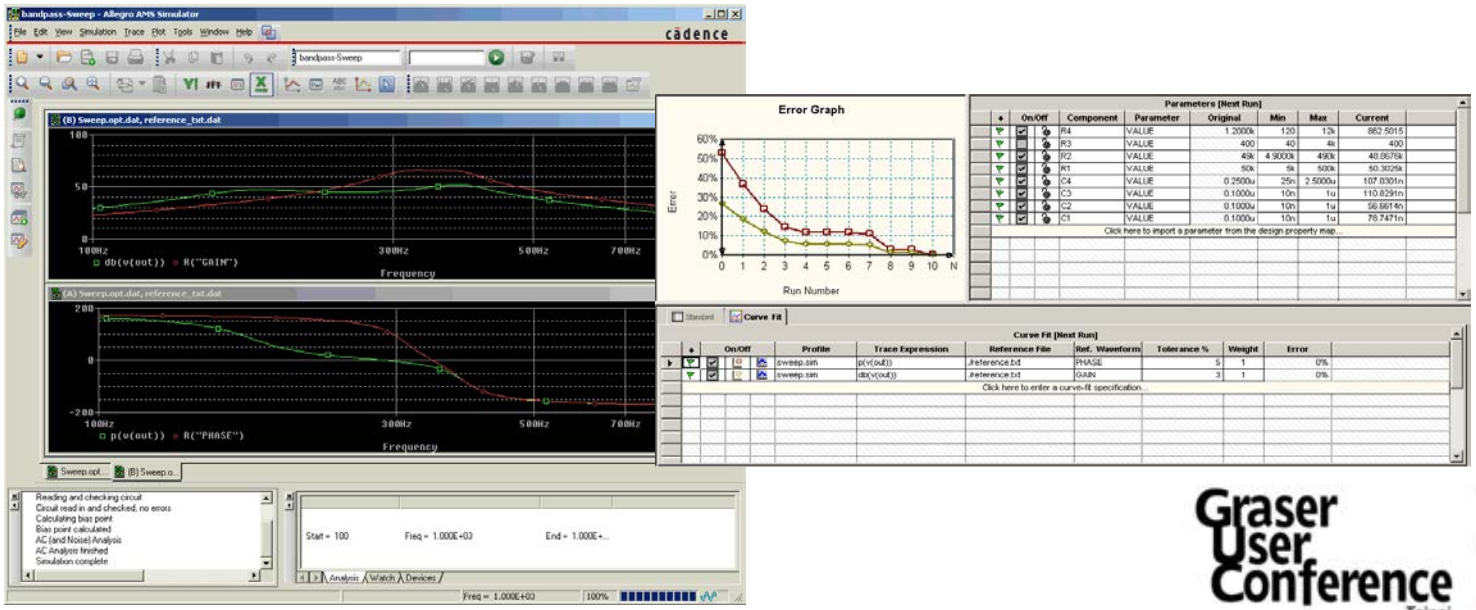
- Engineer specifies the Circuit Topology and the desired goals.
- Optimizer does the rest. It calculates the optimum component values to use to meet the desired goals.
  - One of the **most powerful** analysis tool.
  - Identify components impacting key circuit goals.
  - Identify designs goals and **optimize your design** to meet/beat these goals.
  - Design goals examples: Gain, BW, Overshoot, Pd, Ripple.
  - Design goals can be described as a waveform or Specification.



Graser User Conference Taipei

Optimizer

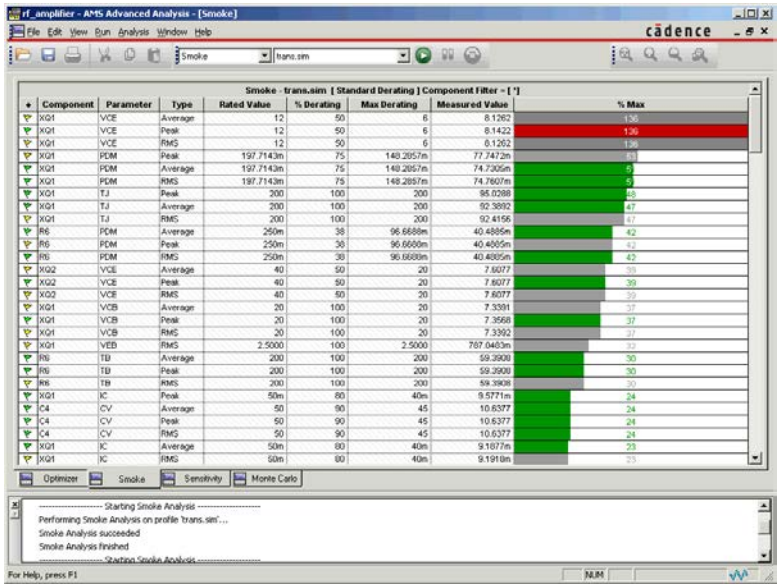
- Finds optimum combination of component values; automatically simulates, evaluates results, and adjusts component values to reach performance requirements.
- Can be used to retarget an existing design at new goals.
  - To meet requirements of newer circuits.
  - To meet regional requirements.
- Saves time for engineers by taking over the often “mindless” task of final optimization
- Can optimize to a set of goal functions and/or a set of curves.



Graser User Conference Taipei

Smoke

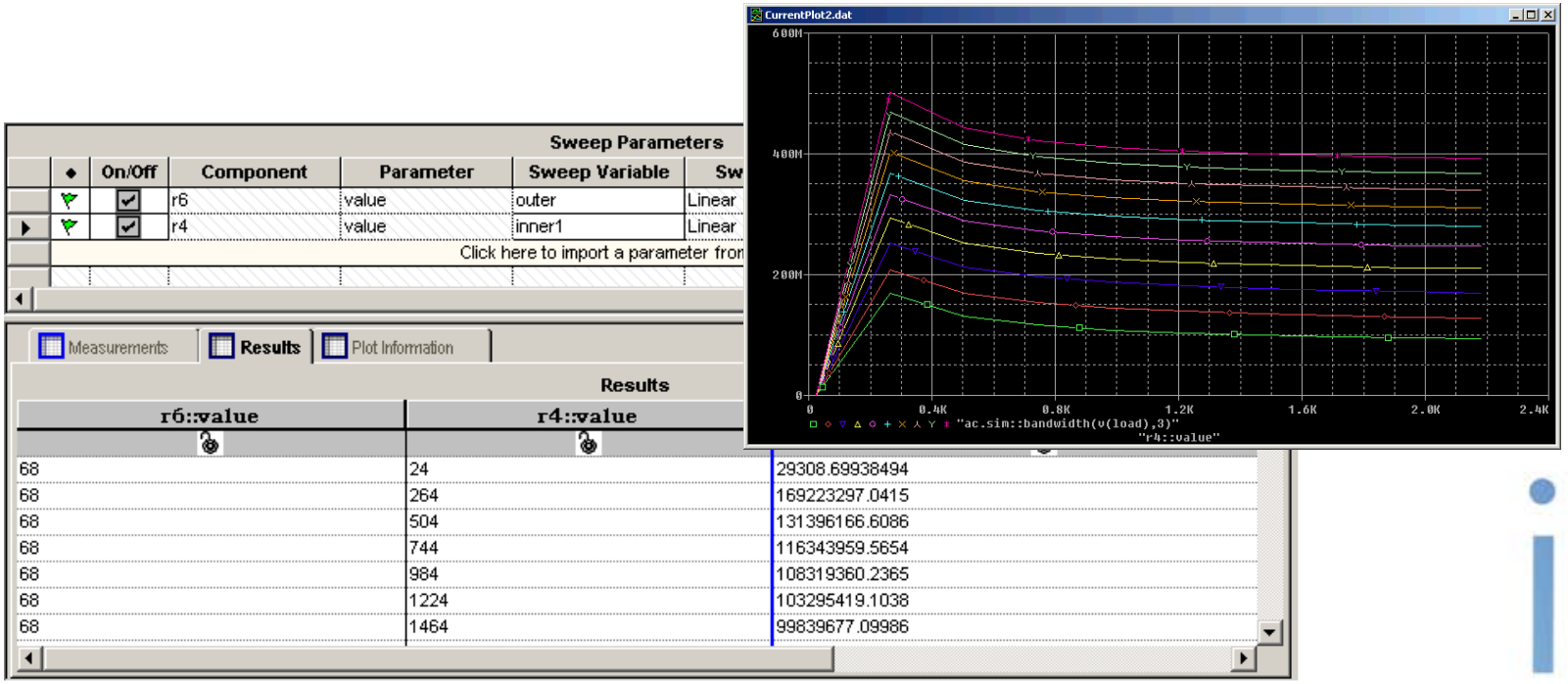
- Checks components to see if they are approaching or exceeding their recommended safe operating limits.
- Allows users to specify their own derating criteria to ensure components are not stressed, or exceed safe operating limits.
- Determines a part's performance under various stresses and environmental conditions.
- Helps in thermal design aspect of electronics circuit.
- Calculate Peak, Avg. and RMS power.
- Cross-probe to automatically find listed parts in the schematic.



Graser User Conference Taipei

Parametric

- Sweep multiple (nested) parameters.
- Quickly view results and create families of curves.
- Ensure there is no unusual circuit behavior while sweeping the component values.



Sweep Parameters

	On/Off	Component	Parameter	Sweep Variable	Sw
	☑	r6	value	outer	Linear
▶	☑	r4	value	inner1	Linear
		Click here to import a parameter fror			

Measurements | Results | Plot Information

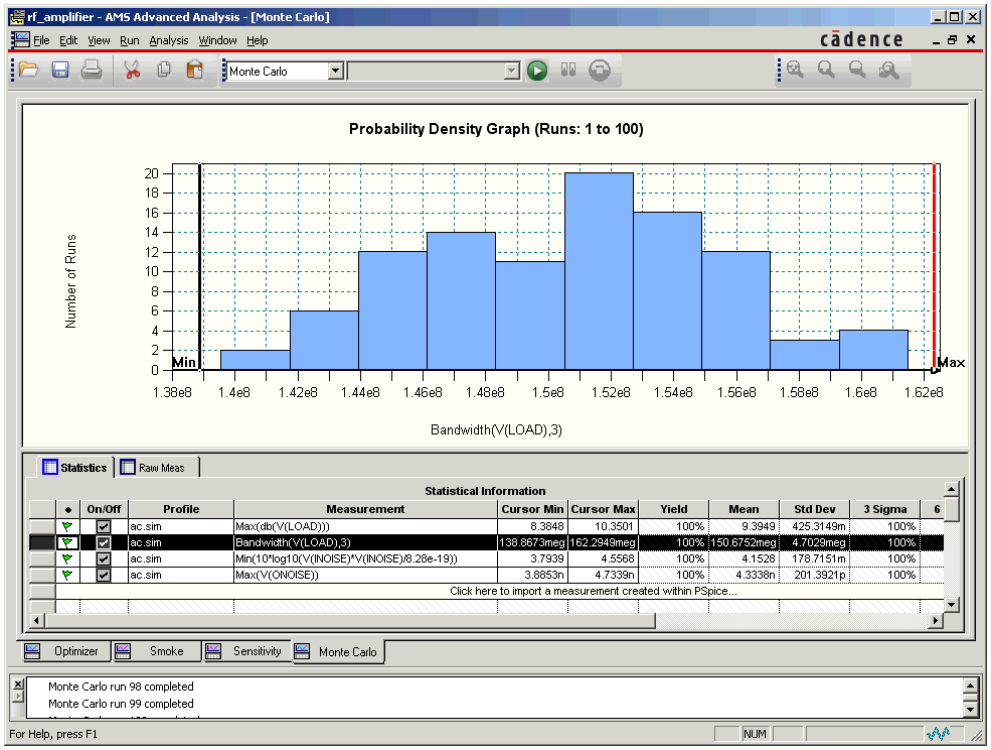
Results

r6::value	r4::value	
68	24	29308.69938494
68	264	169223297.0415
68	504	131396166.6086
68	744	116343959.5654
68	984	108319360.2365
68	1224	103295419.1038
68	1464	99839677.09986

Graser User Conference Taipei

Monte Carlo

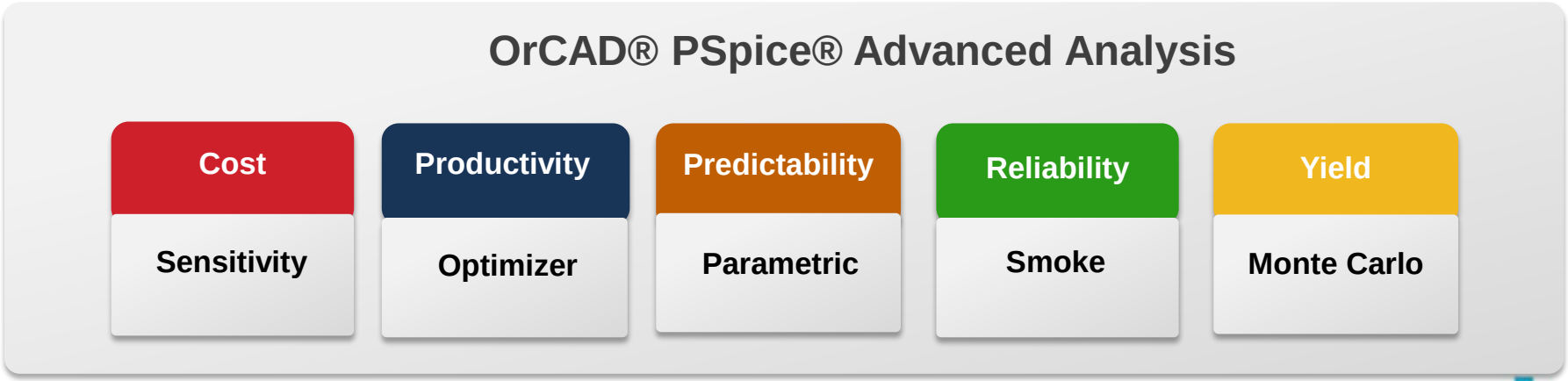
- Calculate Yield before going into manufacturing.
- Produce circuit performance statistics due to device variations.
- Set specification minimum and maximum, and estimate production yield before going to production.
- View graphical results as probability density histogram, or as cumulative distribution function.



Graser User Conference Taipei

OrCAD PSpice Advanced Analysis

- Help customers optimize their design while maintaining their cost and yield.



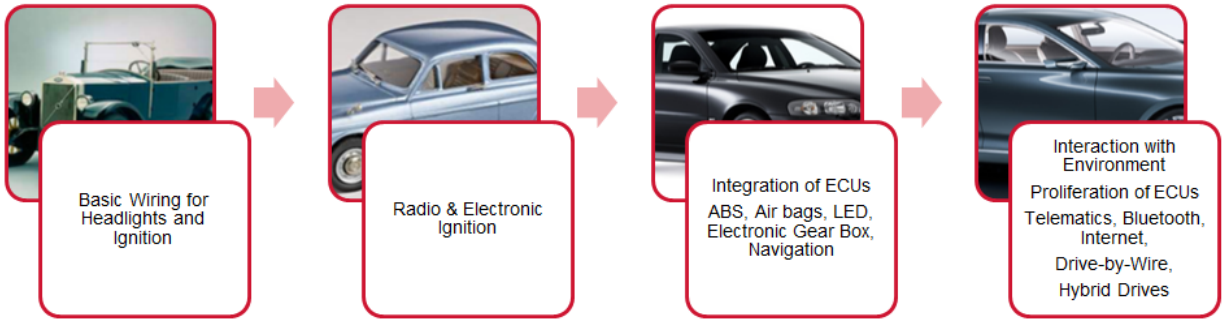
Graser User Conference Taipei

Graser User Conference Taipei

Industry Application

Automotive

- Automotive Engineering Design Challenges



Systems Modeling

ECU Logic Authoring

Power Electronics

Multi-Domain
Mixed Signal
Control Systems

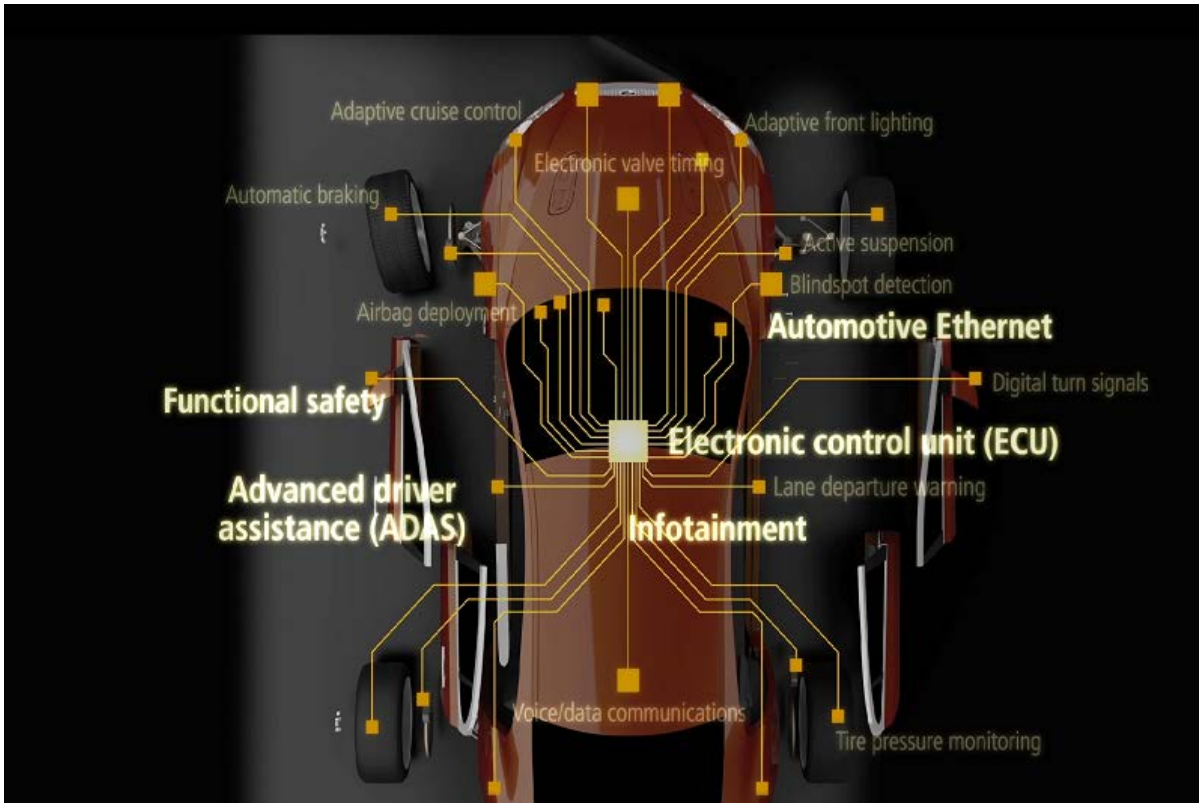
Sensors

Network Enabled

Embedded Software

Automotive

- Automotive Engineering Design Challenges



Systems Modeling

ECU Logic Authoring

Power Electronics

Multi-Domain Mixed Signal Control Systems

Sensors

Network Enabled

Embedded Software

Automotive

- Customer Key Challenges
  - Evaluate electronic interfaces for compatibility and robustness against various automotive conditions.
  - Identify and fix problem before Hardware Freeze.

- Adopt OrCAD® PSpice® Solution
  - OrCAD PSpice
  - OrCAD PSpice Advanced Analysis
  - OrCAD PSpice SLPS

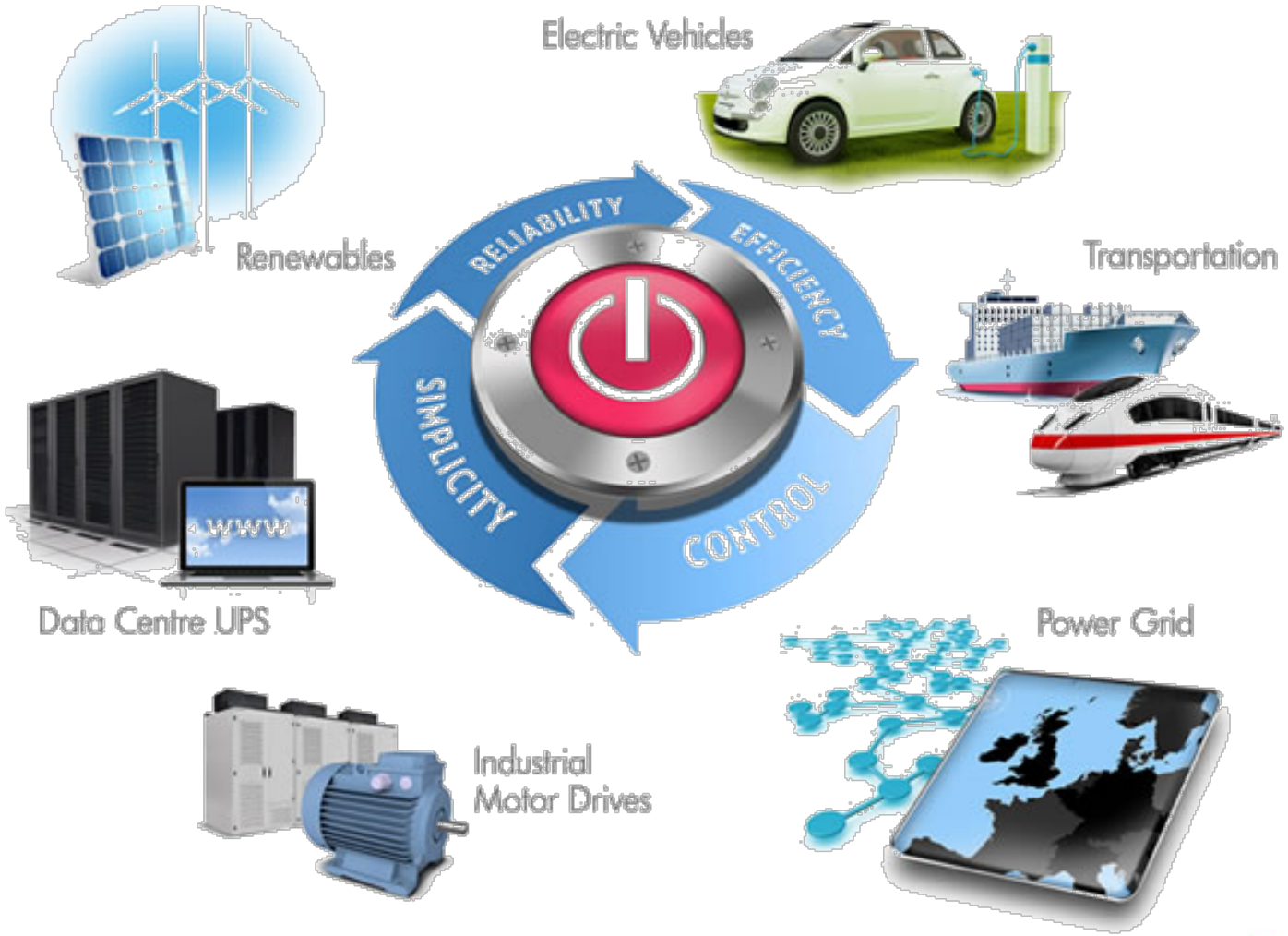
Automotive

- OrCAD® PSpice® Technology
  - Steady State Analysis
  - Transient Analysis
  - System Analysis/Subsystem Analysis
  - Simulate to identify below Failures
    - Short to battery
    - Short to ground
    - Jump start
    - Ground shift conditions
    - Reverse Battery condition
  - Worst Case Analysis
  - Monte Carlo Analysis
  - DC Sweep, Parametric Sweep, Temperature Sweep
  - Smoke Analysis

Automotive

- Results
  - Quickly identified stress on electronics components at different conditions.
  - Evaluated Power dissipation.
  - Simulated ECU subsystem with other automotive electronic subsystems.
  - Detected supplier design problems and vehicle interface error earlier on.
    - Saving time and money.
  - Met the rigorous quality and reliability standards.

Power Electronics



Graser User Conference Taipei

Power Electronics

- Customer Key Challenges
  - High power = hard to characterize accurately in the lab.
  - Requires special models for transformer(nonlinear magnetic), power semiconductors.
  - Component stress.
  - Power electronics designers that need PCB routing effects included in their simulation.

- Adopt Cadence® Solution
  - OrCAD® PSpice®
  - OrCAD PSpice Advanced Analysis
  - Cadence® Sigrity™ solution

Power Electronics

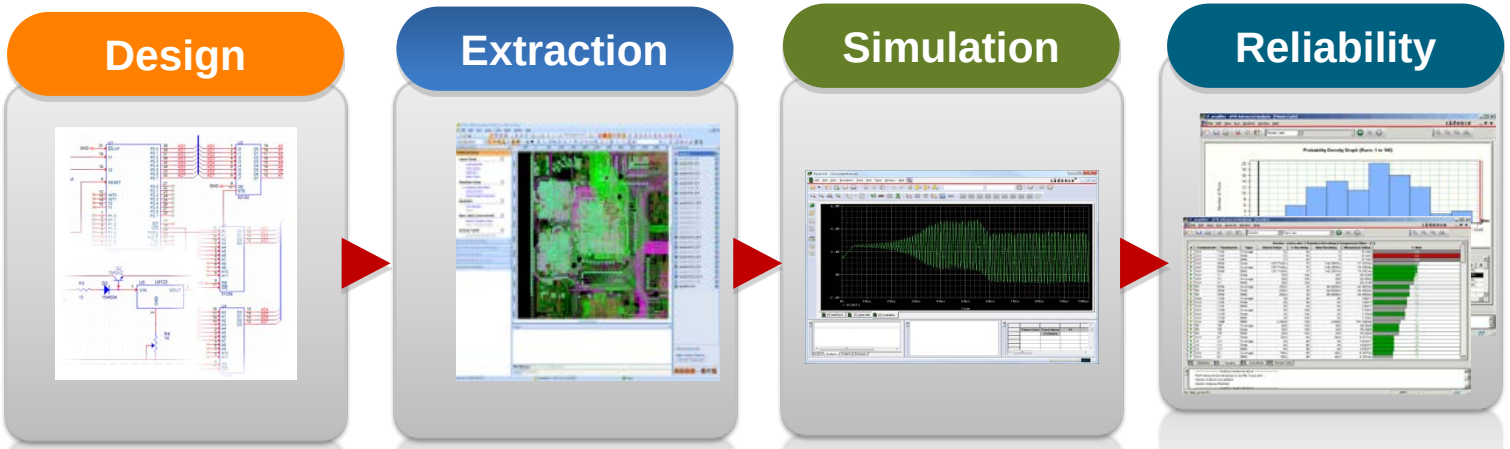
Cadence Solution

- OrCAD® PSpice® Technology
  - Allows mixed-signal simulation and system level analysis capabilities across different levels of abstraction.
  - Core loss calculator.
  - Ease of access to Industry's most comprehensive set of accurate data from manufacturers, which are included in more than 33,000 simulation-ready PSpice models.
  - PSpice Modeling Apps for Voltage Controlled Oscillators (VCOs), Piece-Wise Linear Sources (PWLs), PWMs, Transformers and several others simplify modeling
  - Smoke Analysis.
  - Monte Carlo Analysis

- Cadence® Sigrity™ Technology
  - Extract PCB routing effects.
    - Power electronics designers can simulate early validating design requirements are met.

Power Electronics

- Cadence® Solution
  - OrCAD® PSpice® and Cadence® Sigrity™



- Results
  - OrCAD PSpice verifies component stress.
  - Meet Time To Market targets.
    - Faster development process.
    - OrCAD PSpice delivers accurate results enabled First-Time-Right Design.

System Simulation Solution

- Summary
  - Increase Design Productivity by automating the design process using the PSpice Optimizer. Produce more optimized circuits than humans.
  - Reduce errors from re-entering the schematics when going from schematic to layout.
  - Increase reliability of the design by using the advanced analyses package. Reduce the number of costly Field Failures.
  - Excellent Model development capability Large analog/mixed level components in standard library.

Graser User Conference Taipei

Thank You