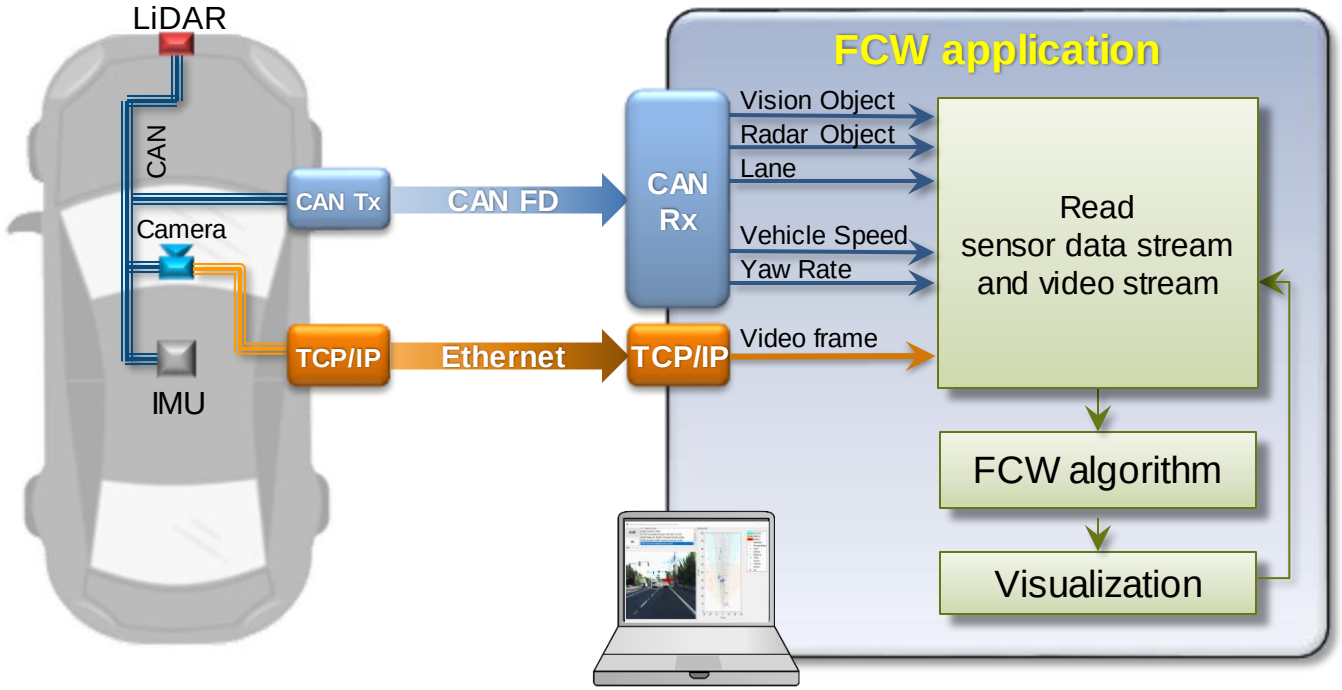
Leveraging Virtuoso/ MATLAB and PSpice/Simulink Integration for AMS Product Development

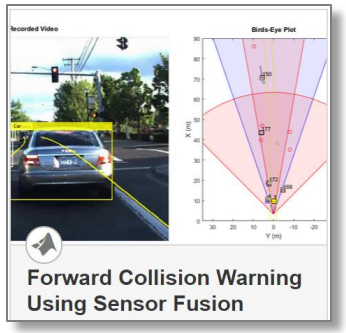
Rajesh Berigei, MathWorks

Kishore Karnane, Cadence

Complex Systems are Everywhere – Here is one

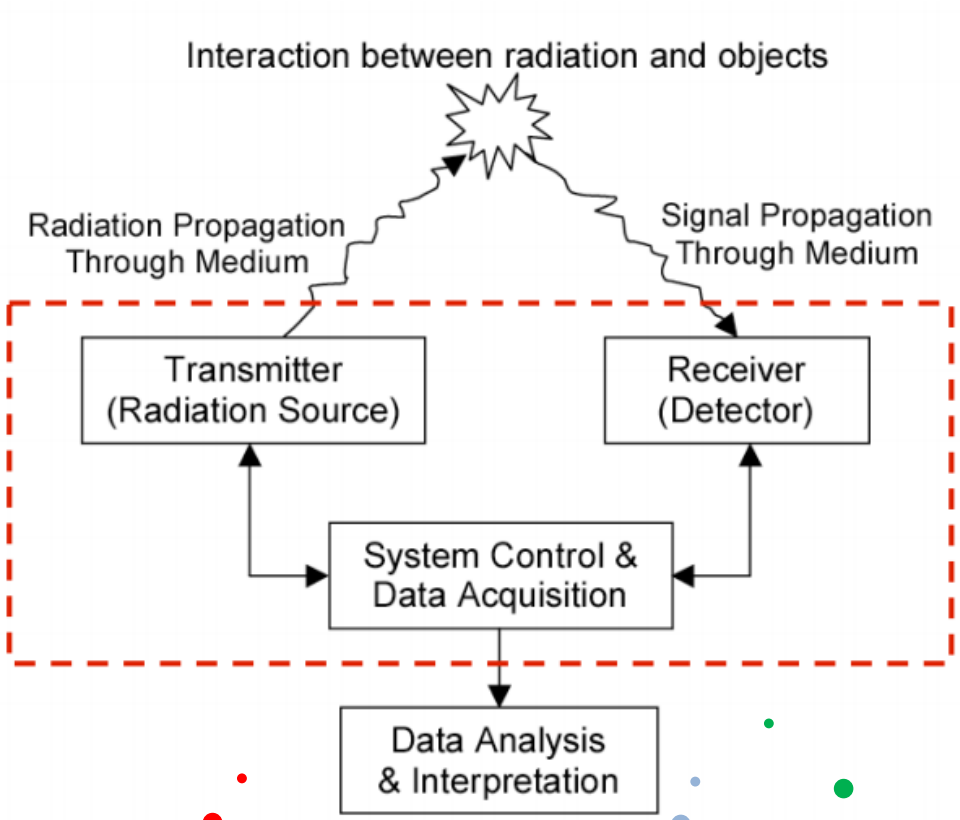
- Evaluate algorithm performance – machine learning, neural nets
- Understand sensor characteristics aligned with real-world situation
- Tune algorithm parameters while driving
- Connect system level to supply chain IC and board components



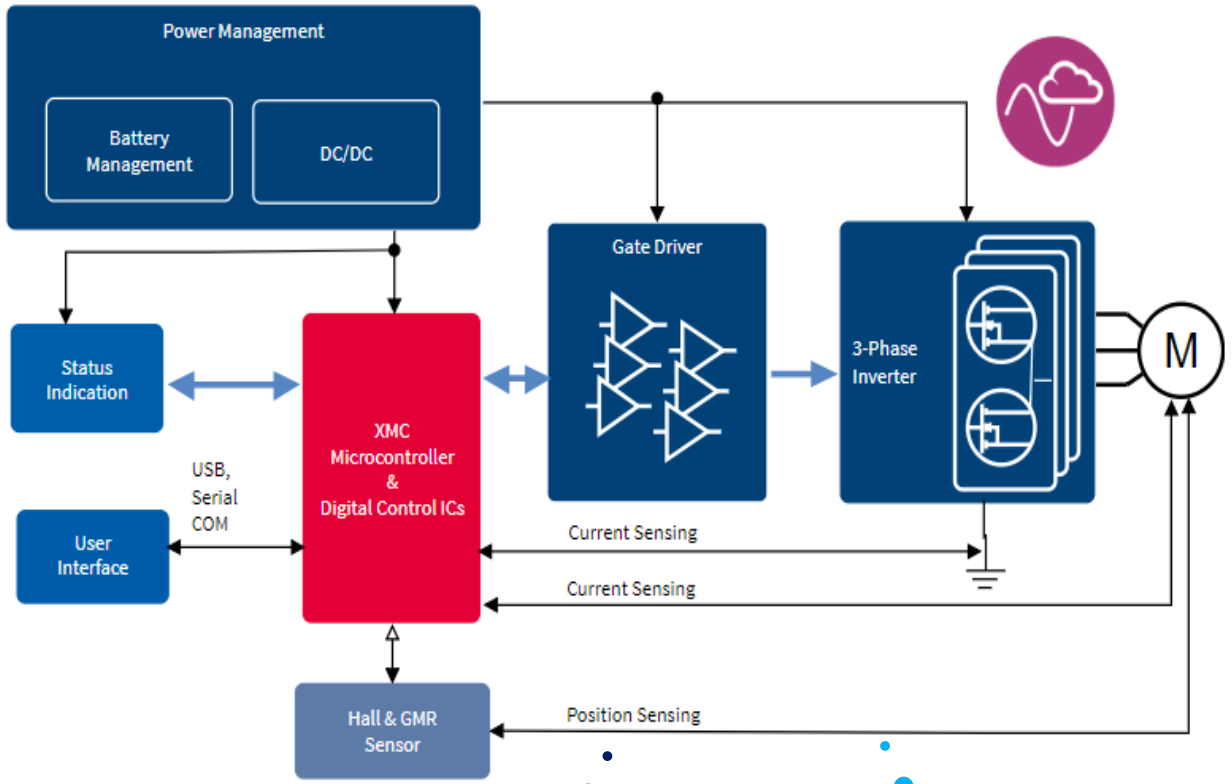


How to tame complexity?

LiDAR Tranceiver



Motor Drive Control



Volume Production On ASIC?

Embedded Software?

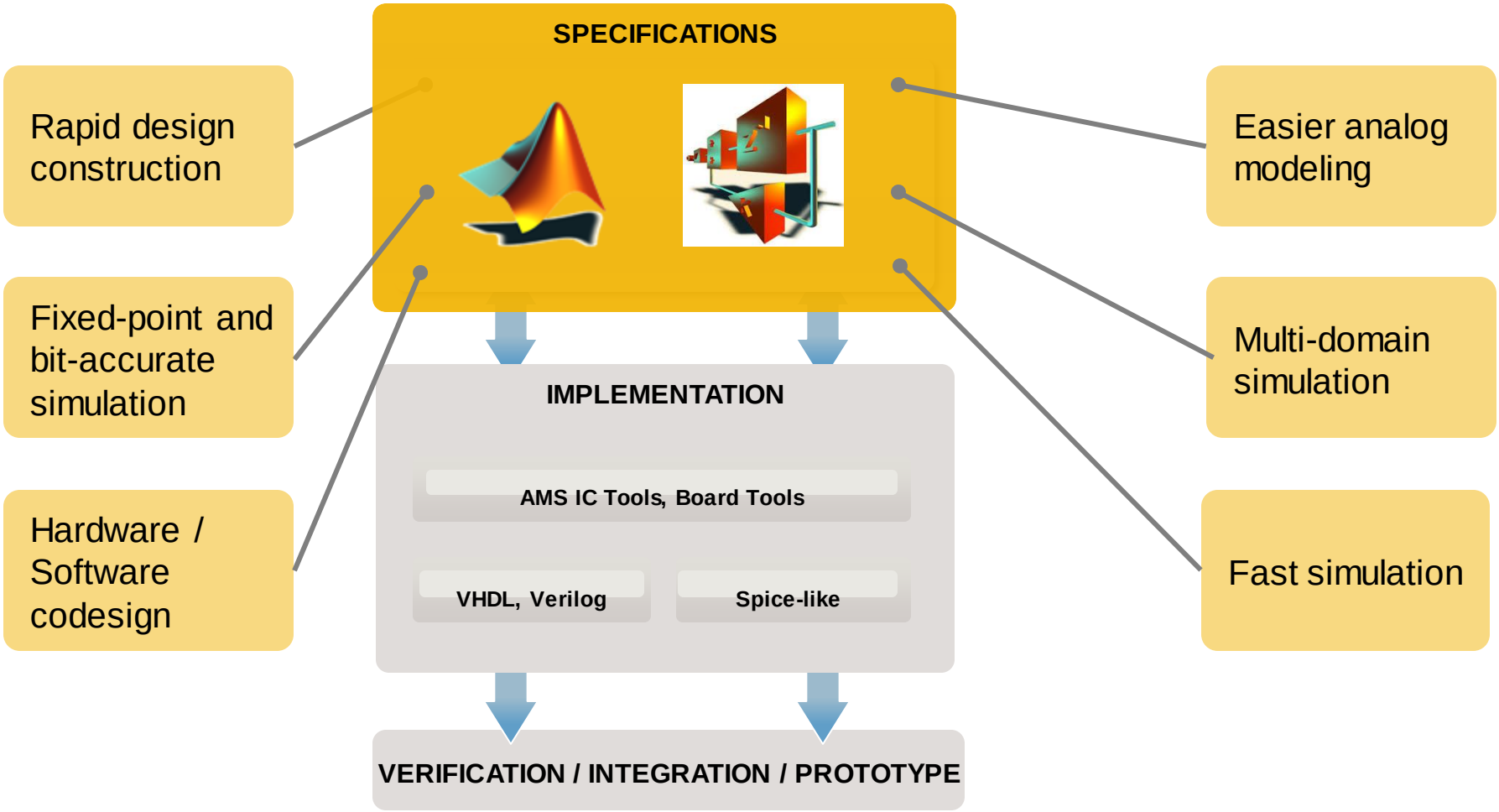
Prototype On FPGA?

Prototype On Board?

Volume Production On SOM?

Top-Down Design With MATLAB and Simulink

Focus on Simulation and Model Refinement at the System Level



Options to Integrate Workflow with Downstream IC and PCB Tools

- Cosimulation

Option 1

- Code Generation

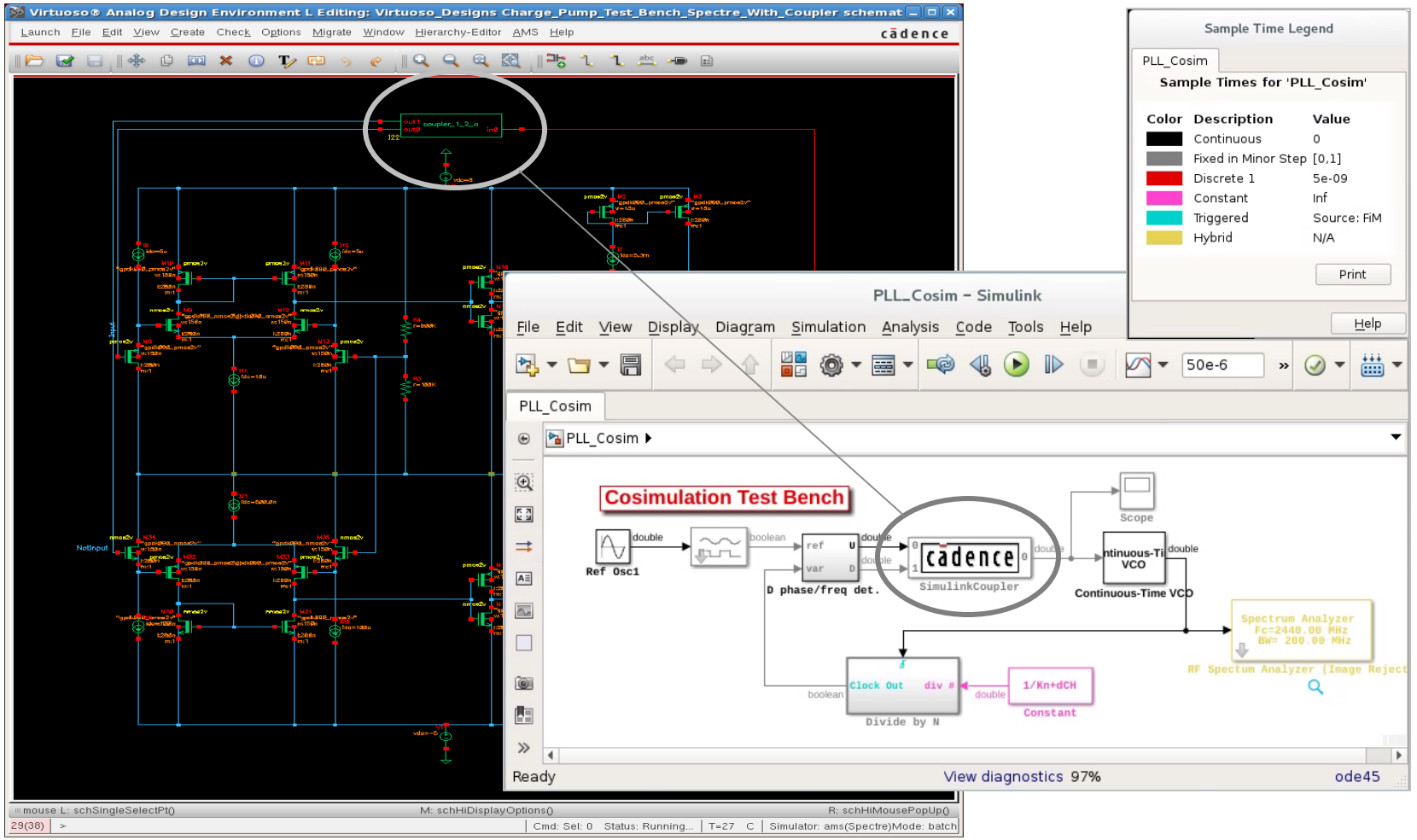
Option 2

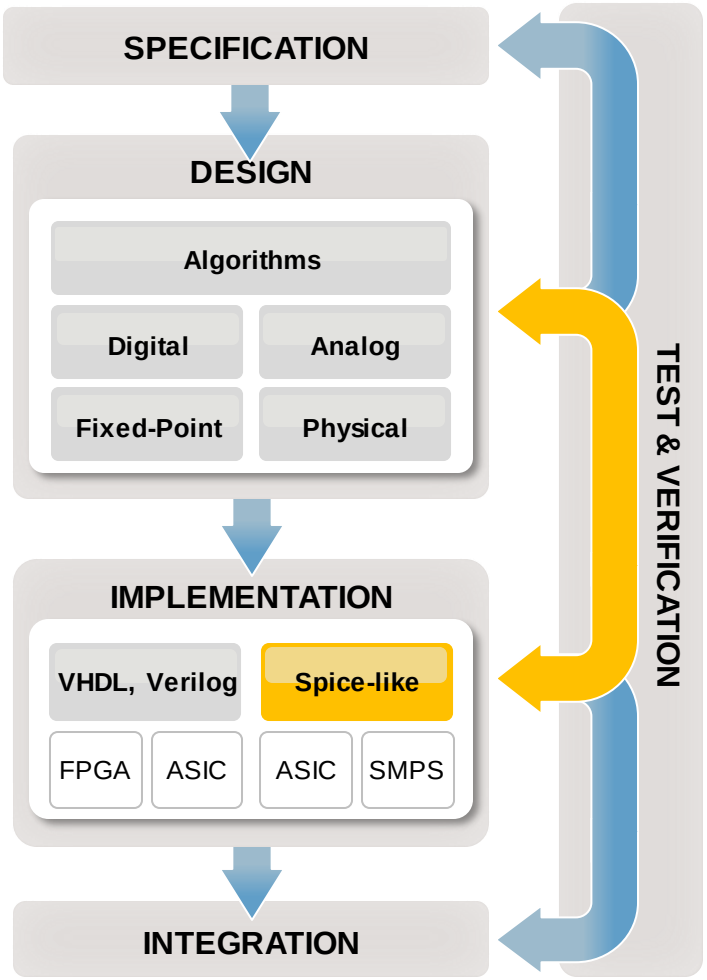
- Post-Processing

Option 3

Option 1: Cosimulation

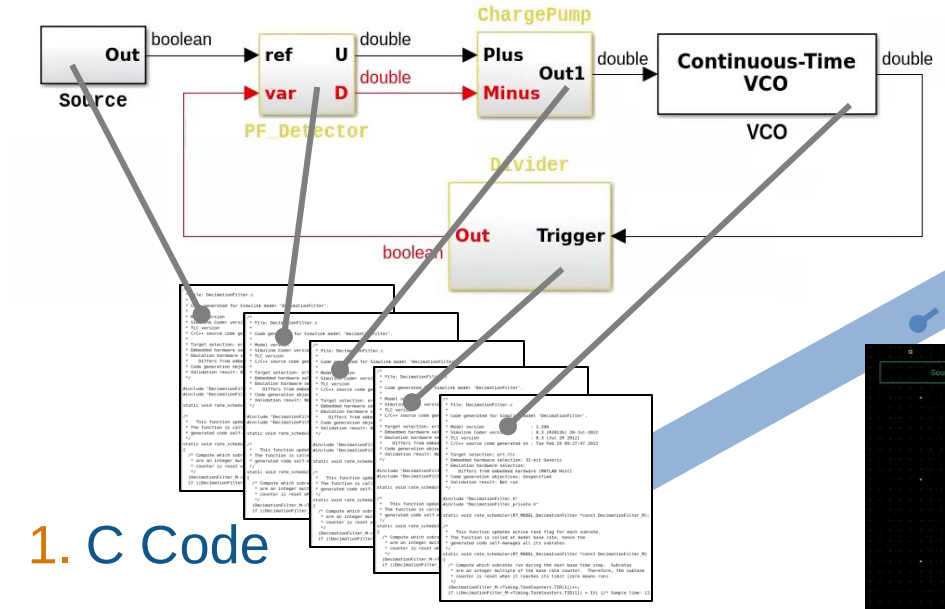
- Verify the transistor implementation against the executable specifications

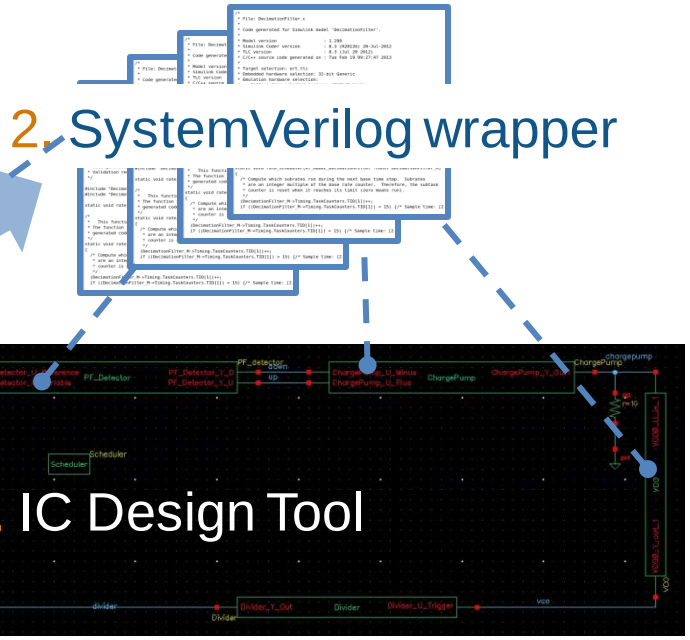




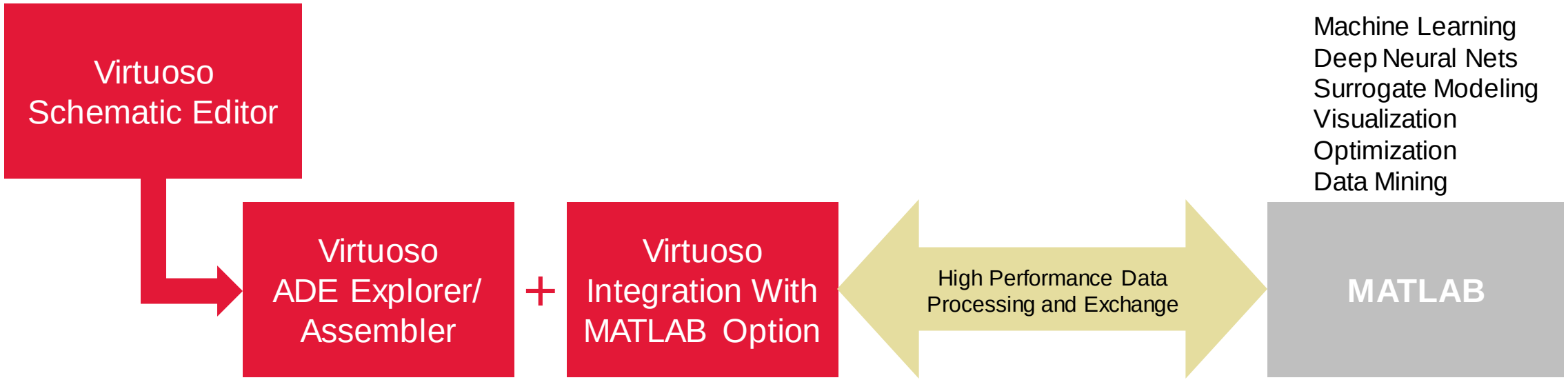
Option 2: DPI-C Compliant System Verilog Generation

1. Make the Simulink model / MATLAB code compliant with C code generation
2. Generate C code
3. Automatically wrap the C code using the DPI-C interface
4. Import, build and simulate an equivalent behavioral SystemVerilog model in your IC design tool





Option 3: Simulation Data Post Processing in MATLAB

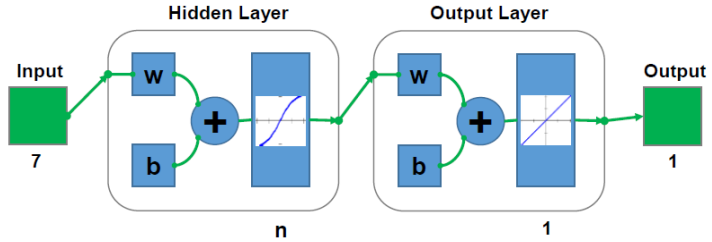


Standard design input methods including the creation of design tests inside Virtuoso ADE Explorer/Assembler/Verifier. These tests can include MATLAB expressions or make calls to MATLAB scripts for post-processing.

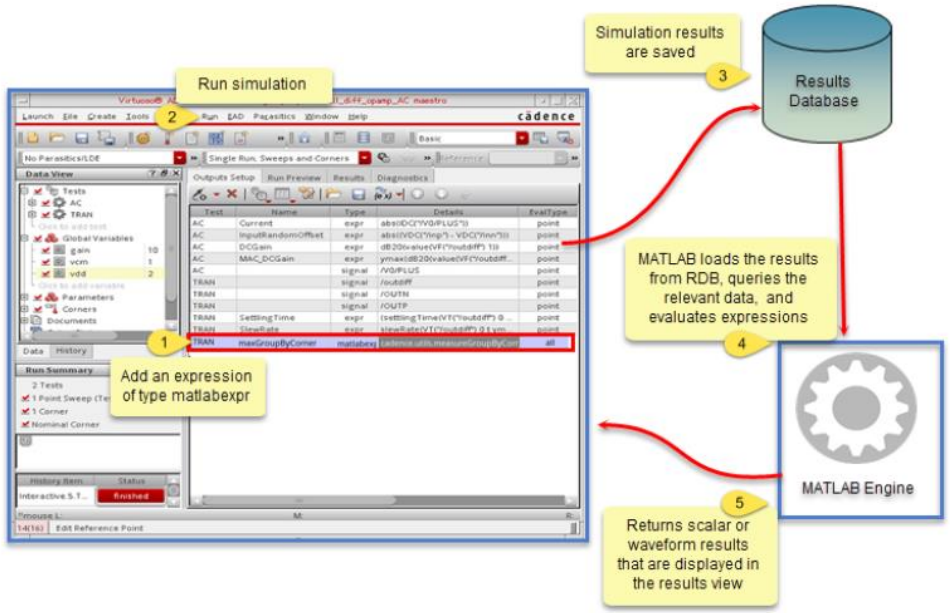
MATLAB can read and produce the PSF XL database for ADE and ViVA. MATLAB can be launched in a real time mode from within ADE for on the fly data-processing.

Option 3: Workflow Using Neural Net Based Design Optimization

Spec for Auto-zero Linear Amplifier
Variables within given range
pm > 40
DC Gain > 20
Minimize Offset Voltage (vos)
Maximize Unit Gain Bandwidth (ugb)



$$f_{obj}(\vec{x}) = 3 * |vos(\vec{x})| - ugb(\vec{x}) + 5 * power(\vec{x}) + area(\vec{x})$$



	Phase Margin	DC Gain	Voltage Offset	Unit Gain Bandwidth	Bias Current	Area
LH Sampling	74.9	35.2	514u	28.9M	0.56u	145
Downhill	76.3	34.3	523u	25.3M	0.5u	122
Spectre	77.5	34.1	522u	21.6M	0.5u	122

Texas Instruments: "A Surrogate Model Optimization Flow for Analog IC Sizing"

Mixed-Signal Example Library

Download from: https://www.mathworks.com/campaigns/products/offer/mixed-signal.html

PLL

PLL Tutorial
PLL Behavioral Model with Impairments
Voltage Controlled Oscillator including Phase Noise
PLL 2.4GHz including Cadence Virtuoso AMS Designer Analog Cosimulation
PLL 50x including different Measurements
PLL with Dual Modulus Prescaler
Fractional N PLL

ADC

ADC Tutorial including Cadence Incisive Digital Cos
ADC Behavioral Model with Impairments and Meas
Interleaved ADC
Subranging ADC
Successive Approximation ADC
3rd Order Sigma-Delta ADC including Circuit Level
4th Order Sigma-Delta ADC

SerDes

SerDes Tutorial
Backplane Modeling Workflow and App
64b/66b Coding
64b/67b Coding
8b/10b Coding
Tunable Equalizer and Bathtub Curve Generation with Statistical Approach and Parallel Simulation
Clock Recovery
SerDes 10 Gbps
SerDes 2 Gbps with Circuit-Level CTLE

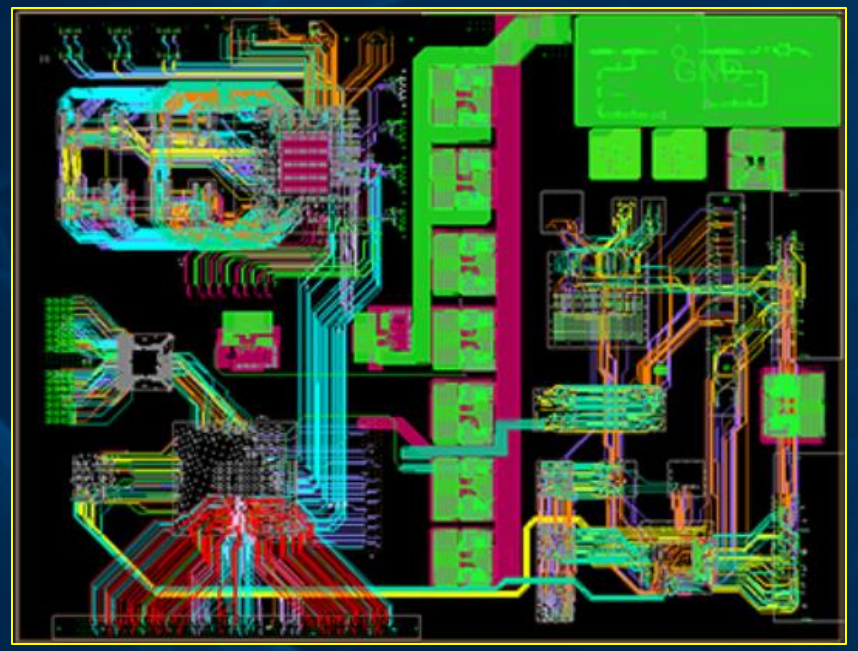
SMPS

Switched Mode Power Supply Tutorial
Boost
Buck
Flyback
SEPIC

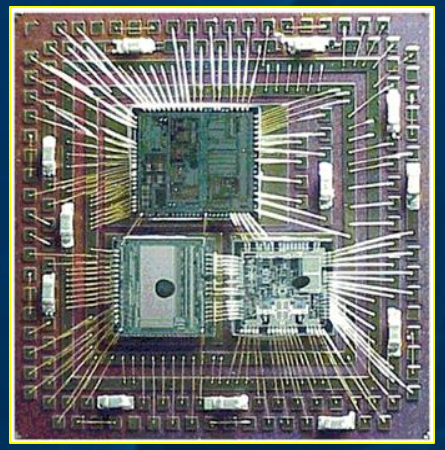
Cadence System Design Environment

Integrating IP, IC, package, PCB, and analysis

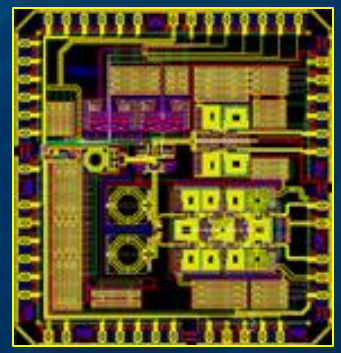
- Our software helps engineers move between various stages of electronic design so that your favorite electronic gadget is ready for the holiday rush!



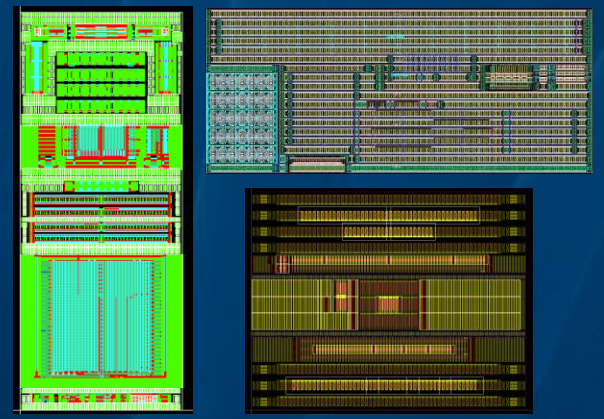
Printed Circuit Board (PCB)



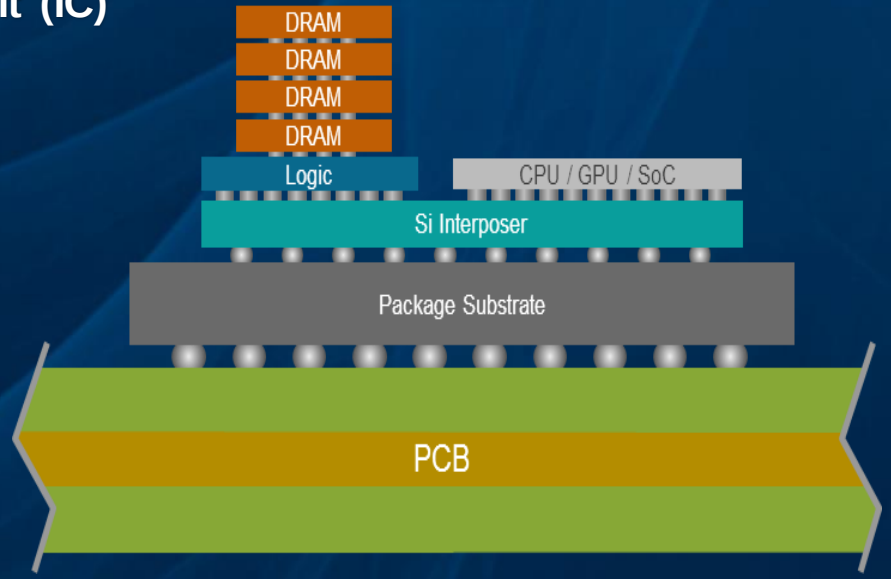
Package



Integrated Circuit (IC)



Intellectual Property (IP)



cadence

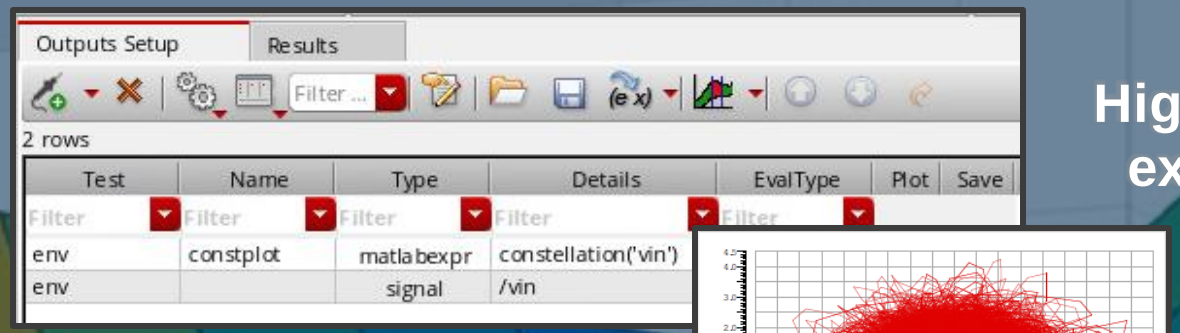
Bridging the Divide Between ICs and Systems

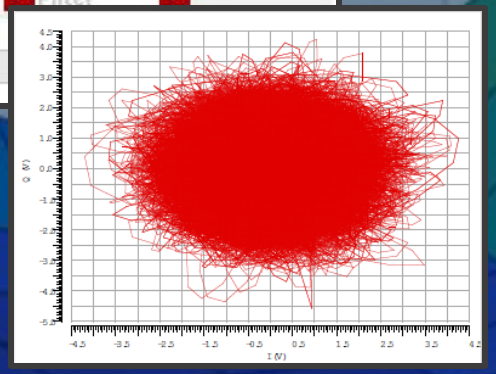
MathWorks system design capabilities integrated with Cadence solutions

MathWorks

+

cadence





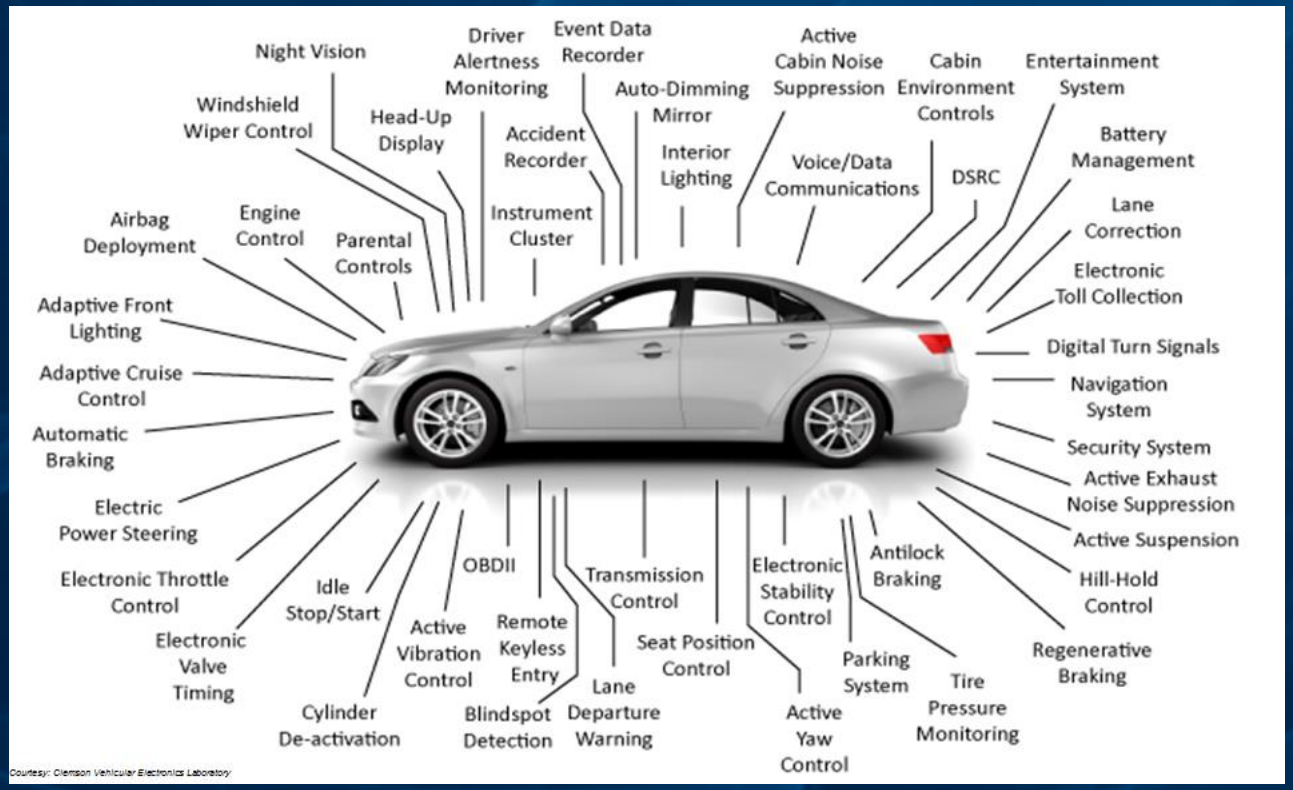
High-performance IC data exchange and analysis

PSpice® <> Simulink

System-level simulation solutions for IoT and automotive applications

cadence

Electro-Mechanical Simulations in Automotive



Systems Modeling

ECU Logic Authoring

Power Electronics

Multi-Domain Mixed Signal Control Systems

Sensors

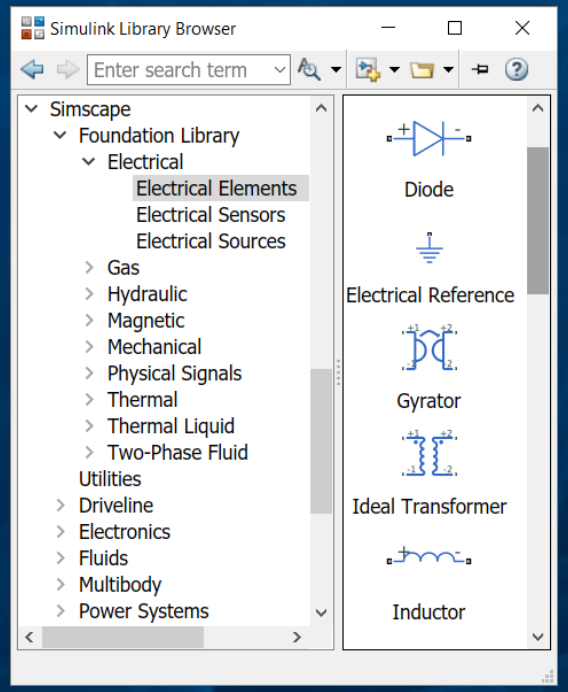
Network Enabled

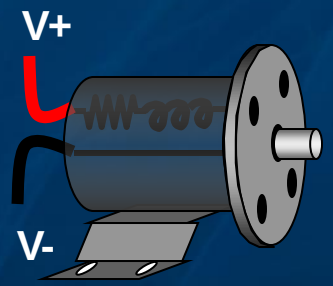
Embedded Software

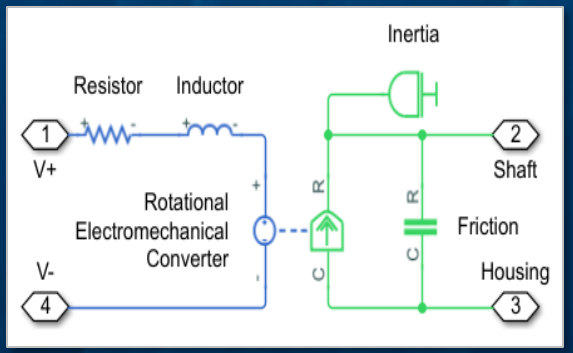
cadence

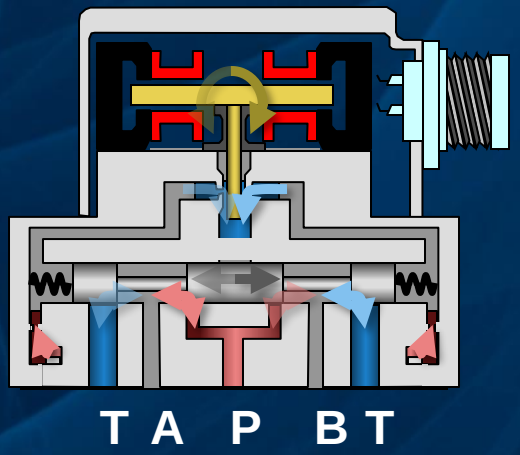
Easily Integrate MATLAB Models for Mechanical Components

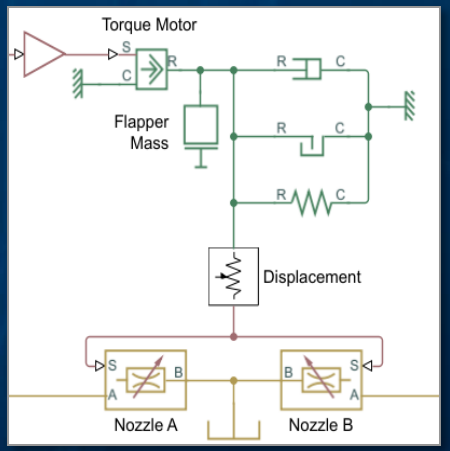
- Eases process of modeling Physical Systems
  - Build models that reflect structure of physical system
  - Leverage MATLAB to create reusable models
- An electrohydraulic servo-valve example
  - Shows multidomain modeling, with electrical, mechanical, and hydraulic components









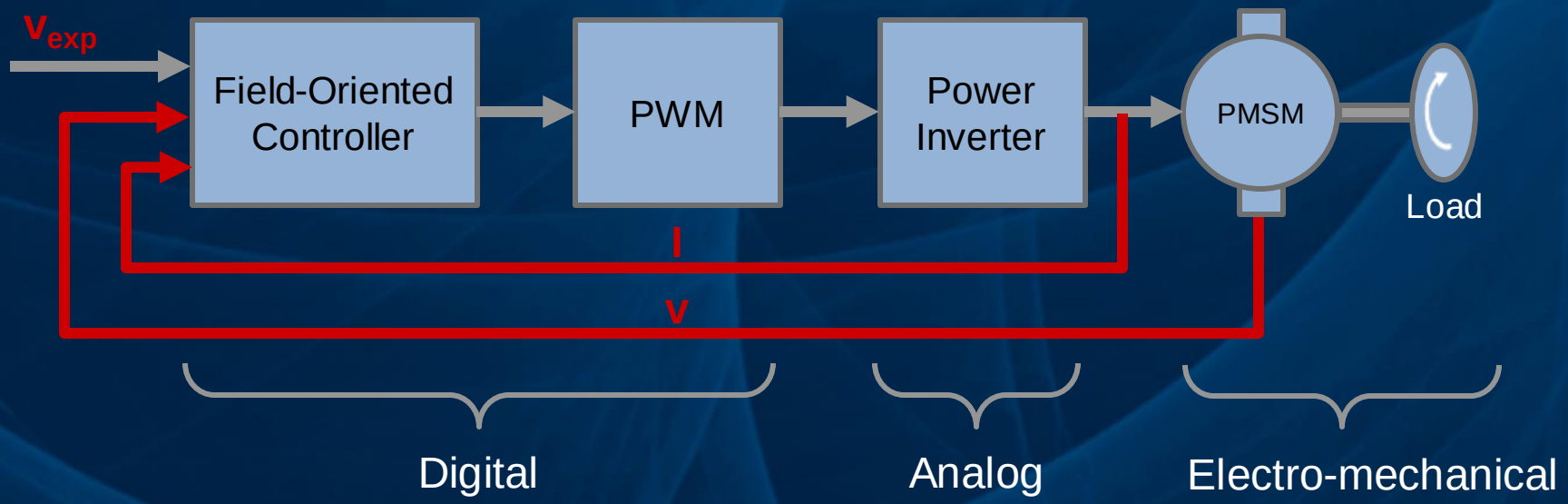


cadence

Permanent Magnet Synchronous Motor Drives

- Field-Oriented Control of a PMSM Drive
- Commonly used in hybrid electric vehicles, manufacturing machinery, and industrial automation

Analog/Mixed-Signal Design

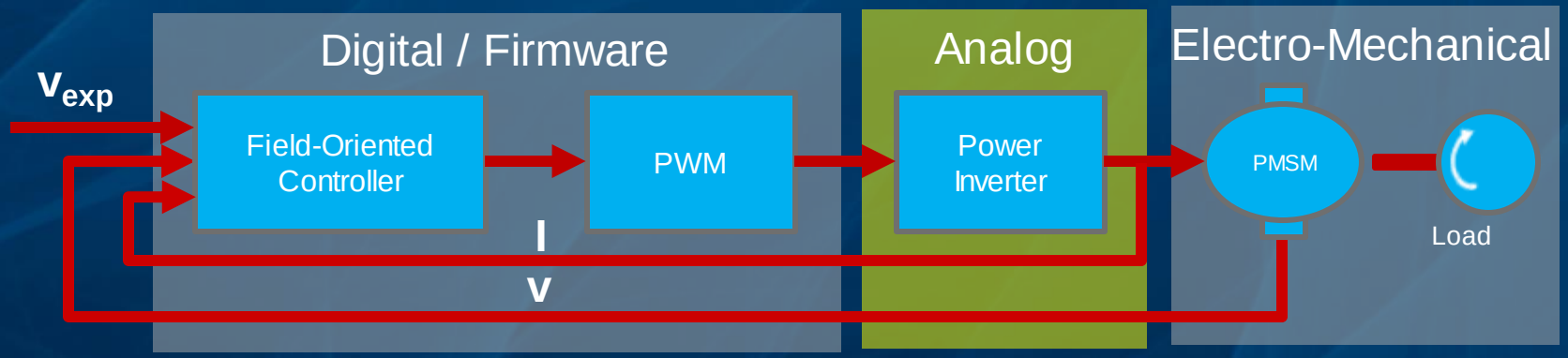


cadence

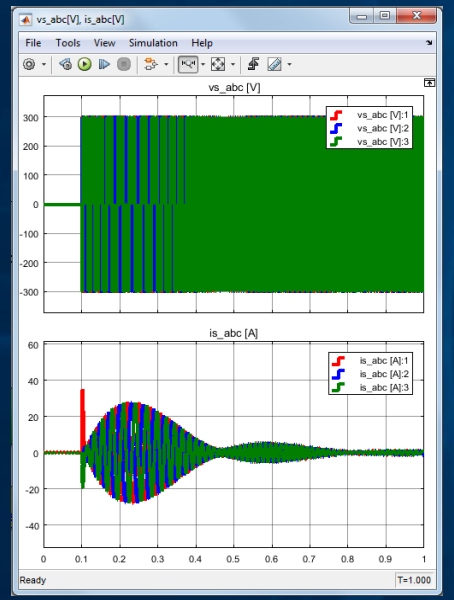
Automotive System Design for Electric Vehicles

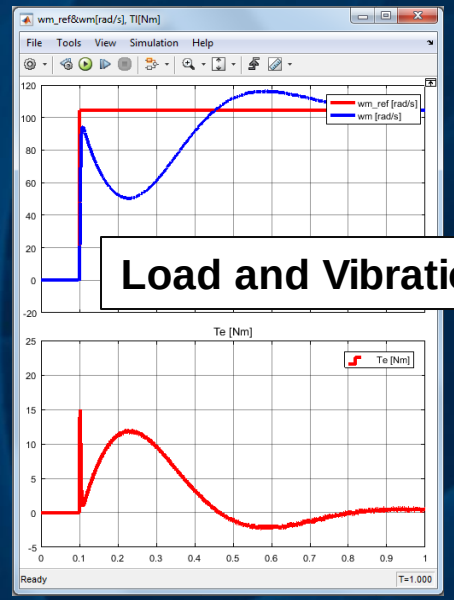
MATLAB / Simulink / PSpice integration

- From actuators to electric vehicle motors
- Acceleration of 0-60mph in 2.7 secs

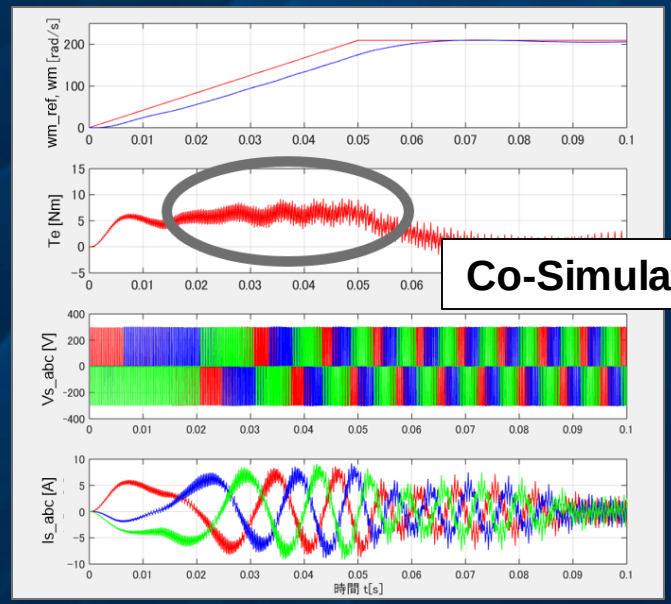


- Example control of a permanent-magnet-synchrone-machine for motor powertrains





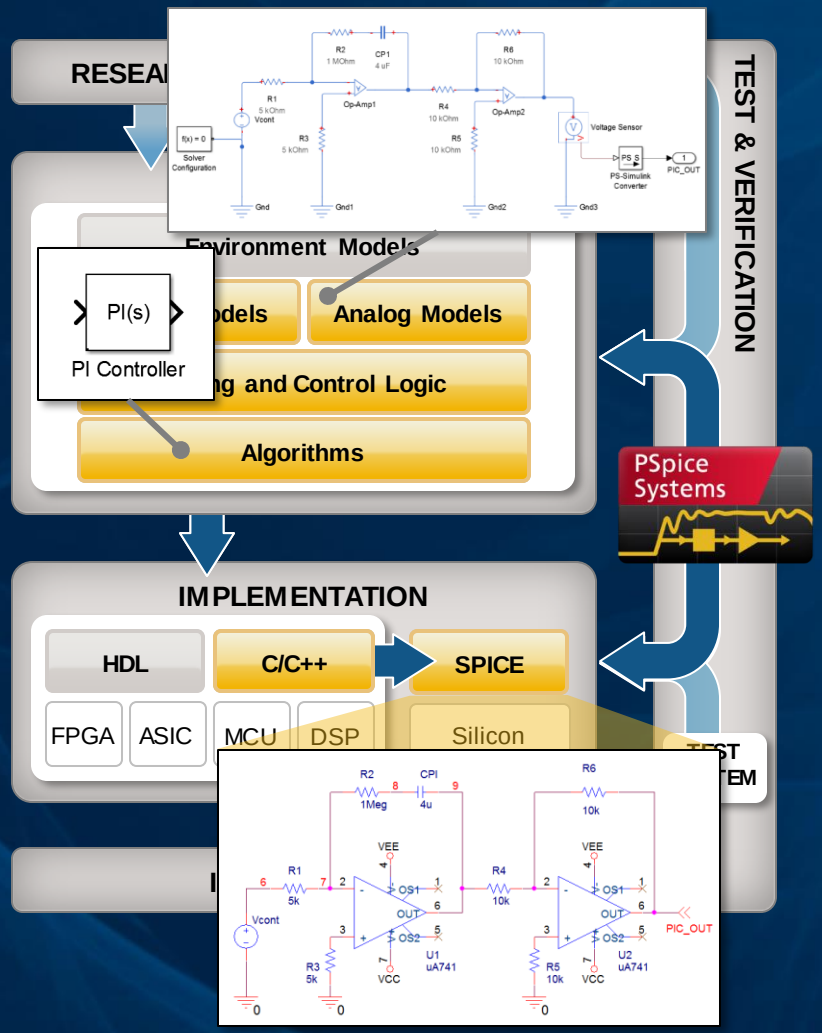
Load and Vibration



Co-Simulation

cadence

Model-Based Design for PCB



- Top-Down Workflow
  - **Starting point:**
    - Mathematical Model
    - Physical Model
  - **Needs**
    - Simulation speed (proof of concept)
    - Reuse of existing testbench
    - Sign-off Transistor-level simulation
  - **Solution**
    - Co-simulation with Simulink and PSpice using PSpice Systems Option
    - Model integration through automatic C code generation and PSpice DMI

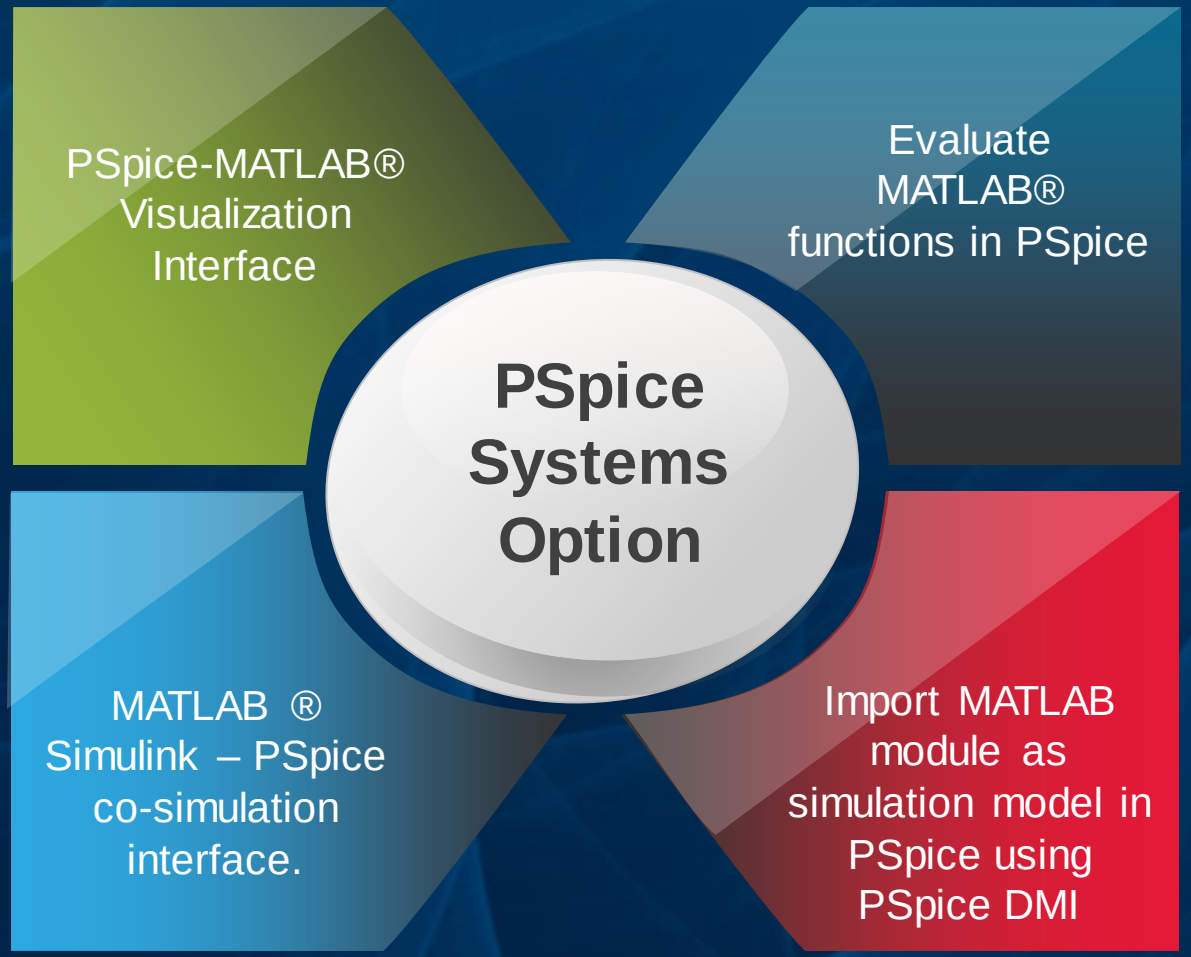
cadence

PSpice Simulink Co-Simulation - Benefits

- Co-simulate electrical, mechanical, and systems
- Simulate with ideal models for faster simulation
- Simulate with actual electrical designs using PSpice models
- Electrical simulations with PSpice models exhibit non-linearities, delay, and other real-world effects
- Full access to PSpice and MATLAB environments for in-depth design and debugging and visualizing data

cadence

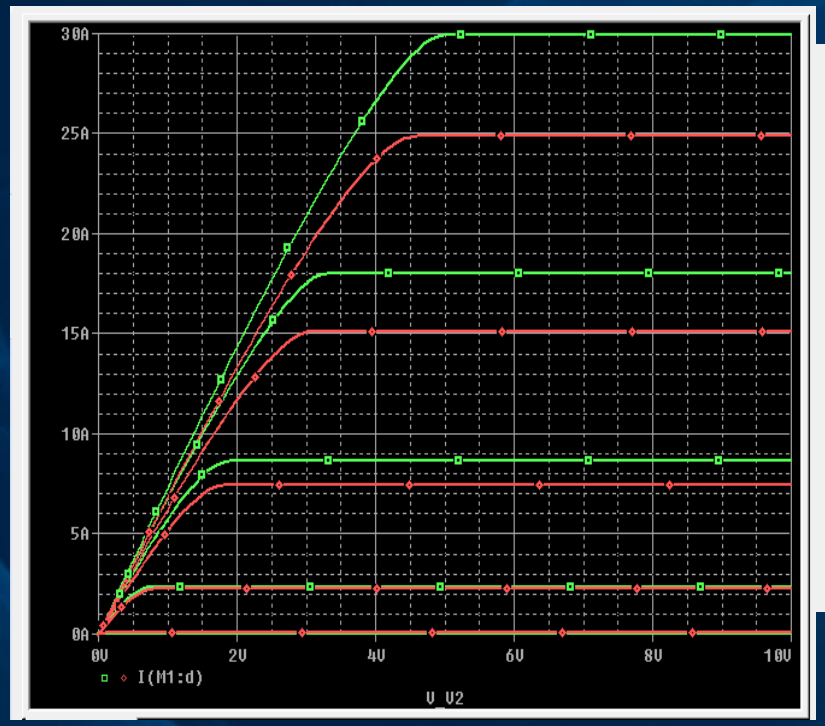
PSpice Systems Option

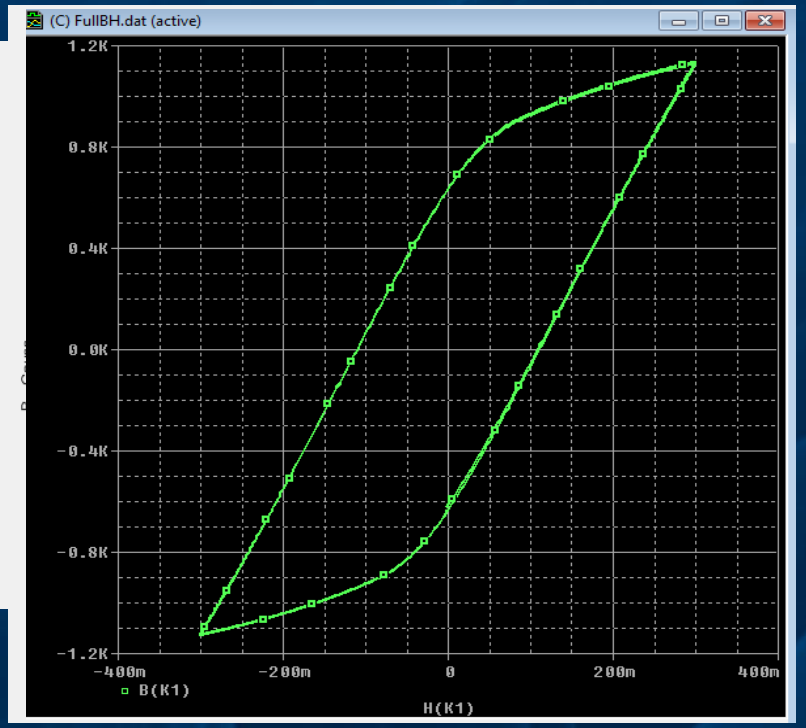


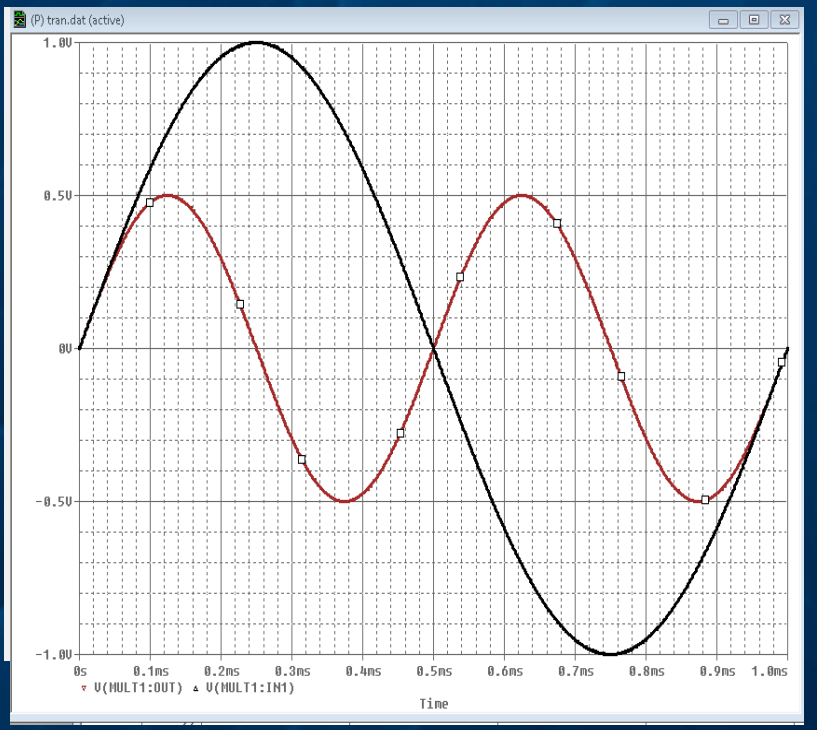
cadence®

Examples: PSpice – MATLAB Visualization Interface

- DC Sweep at Multiple Temperature
- Plot multiple B-H loops
- Polar Plots on AC Analysis

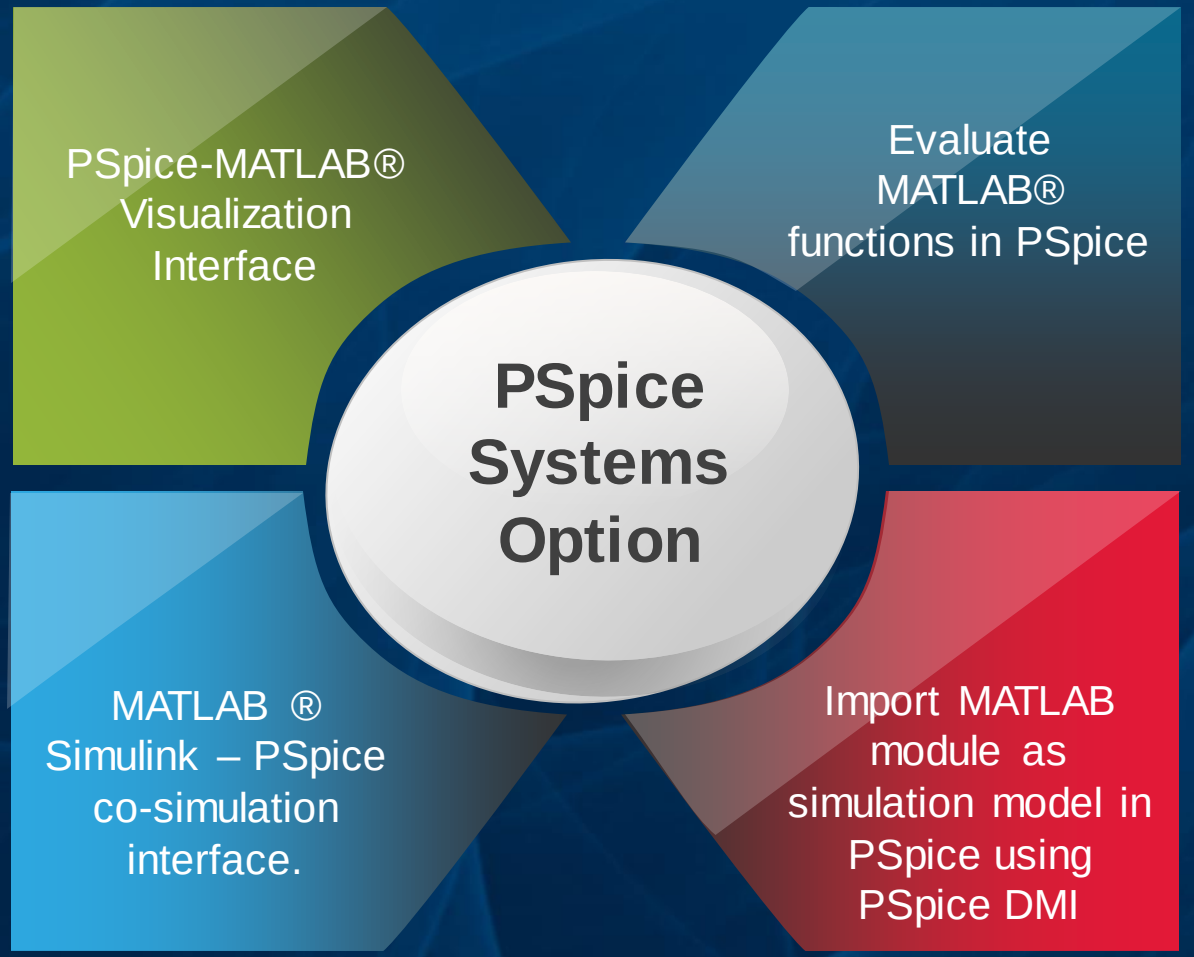






cadence

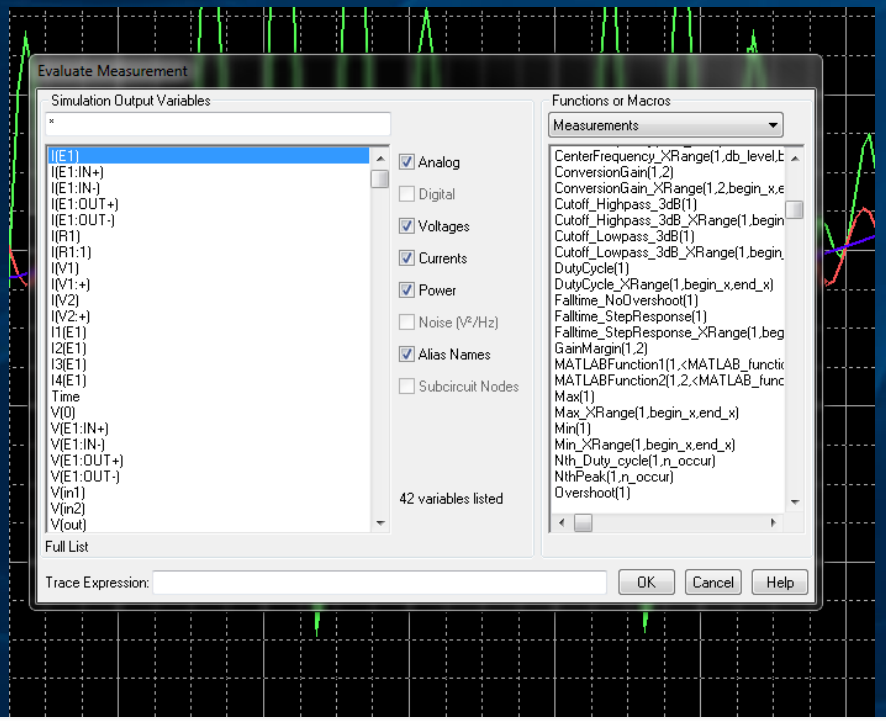
PSpice Systems Option



cadence®

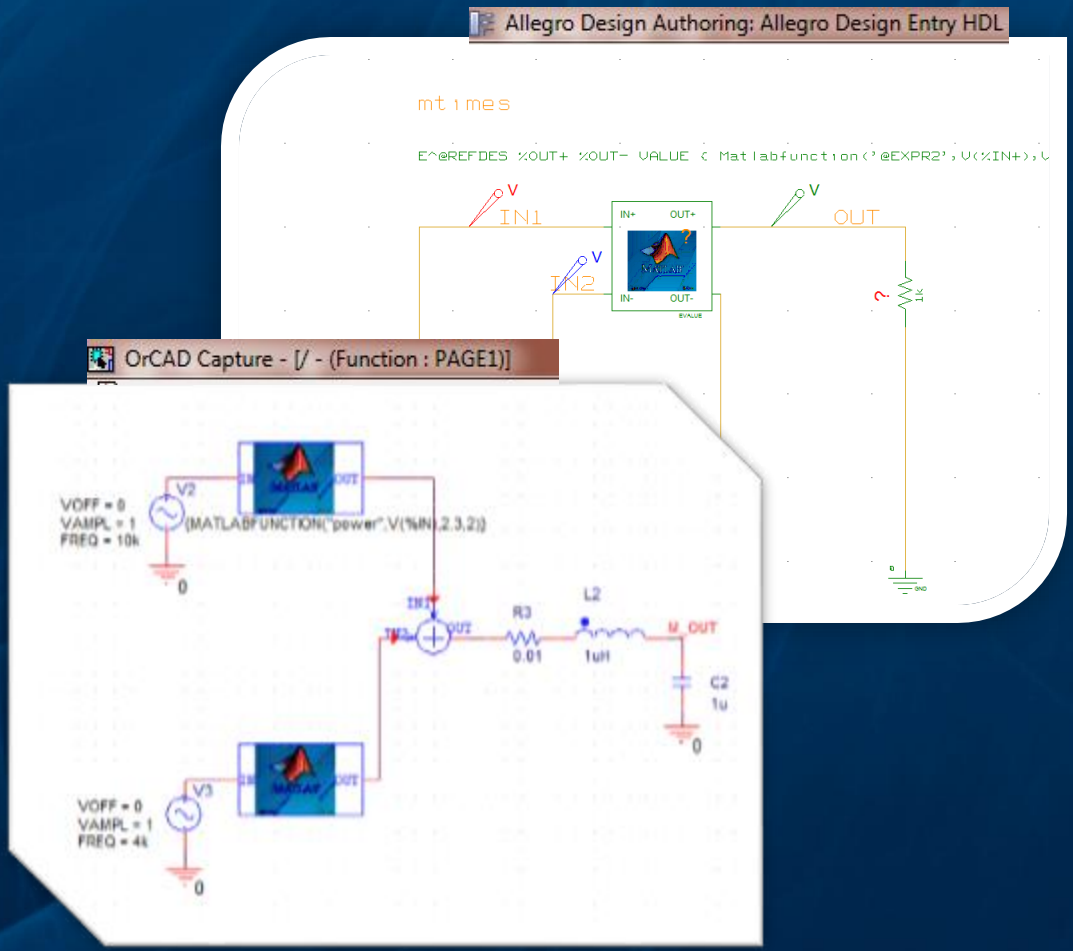
Evaluate MATLAB functions in PSpice

Include MATLAB functions for measurements



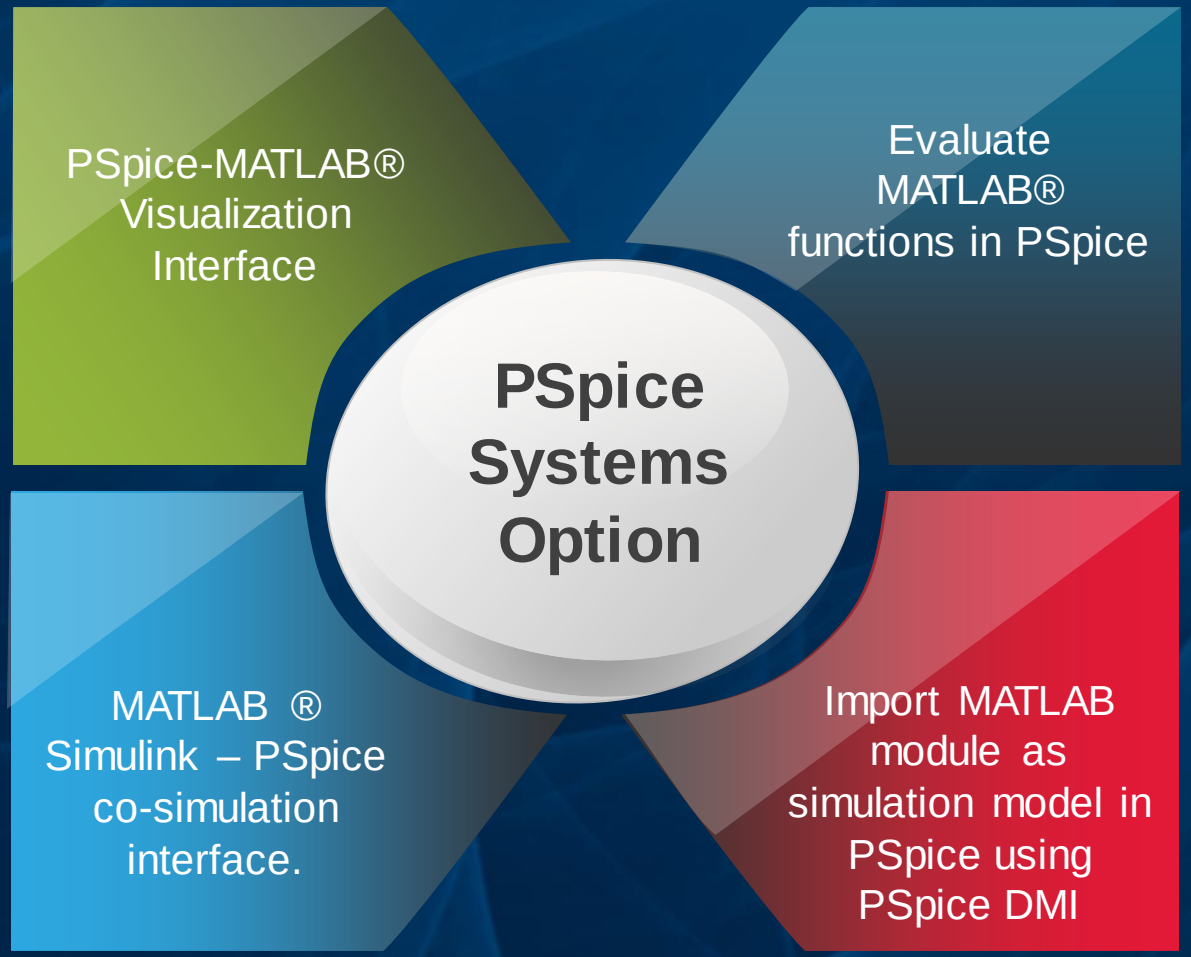
Evaluate	Measurement	Value
☑	matlabFunction1(V(RLC),risetime)	8.95455
☑	matlabFunction1(V(ABM3:IN),peak2rms)	1.35111
☑	matlabFunction1(V(R2:2),peak2peak)	71.99985m

Use MATLAB functions in simulation



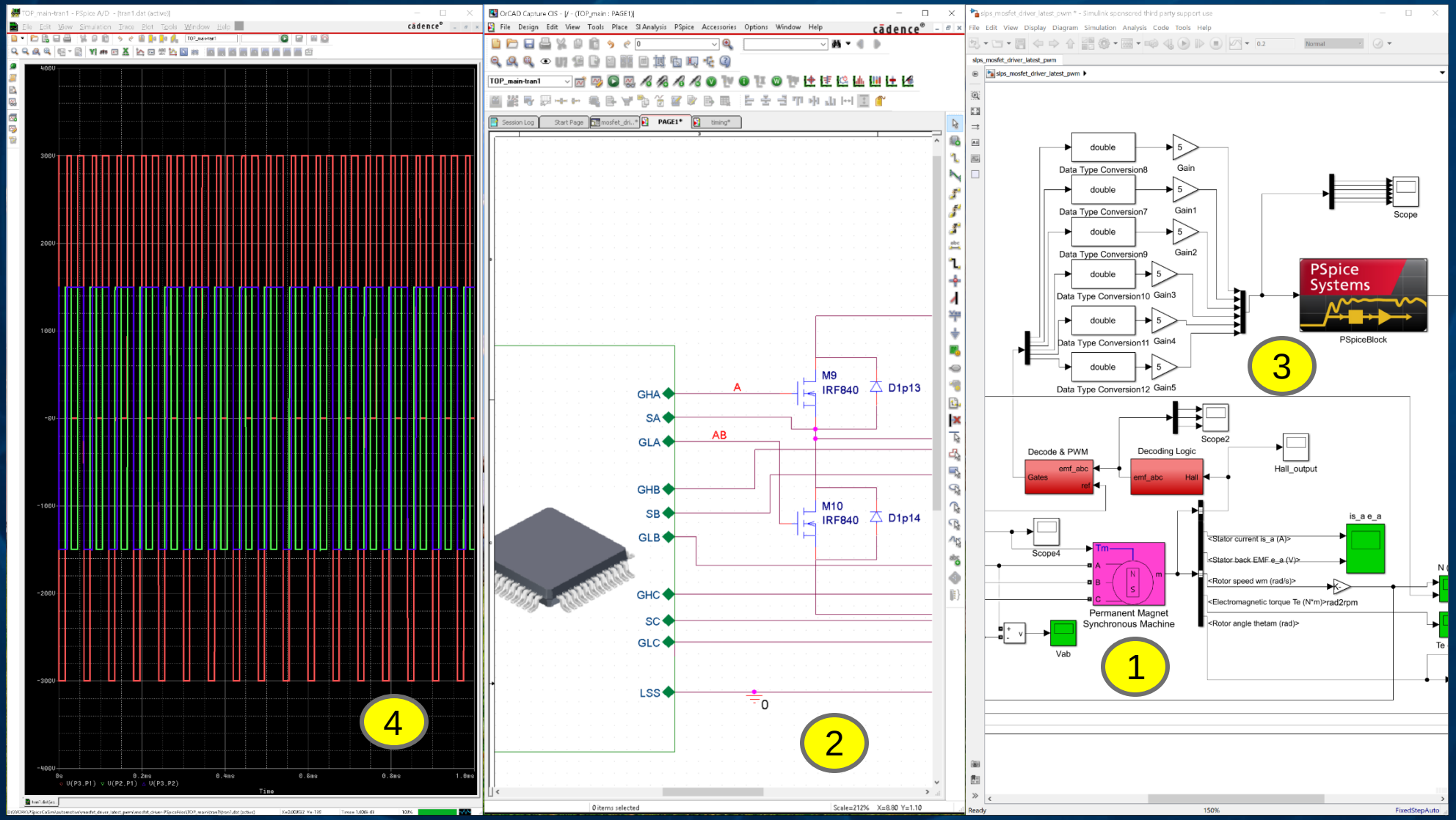
cadence

PSpice Systems Option



cadence®

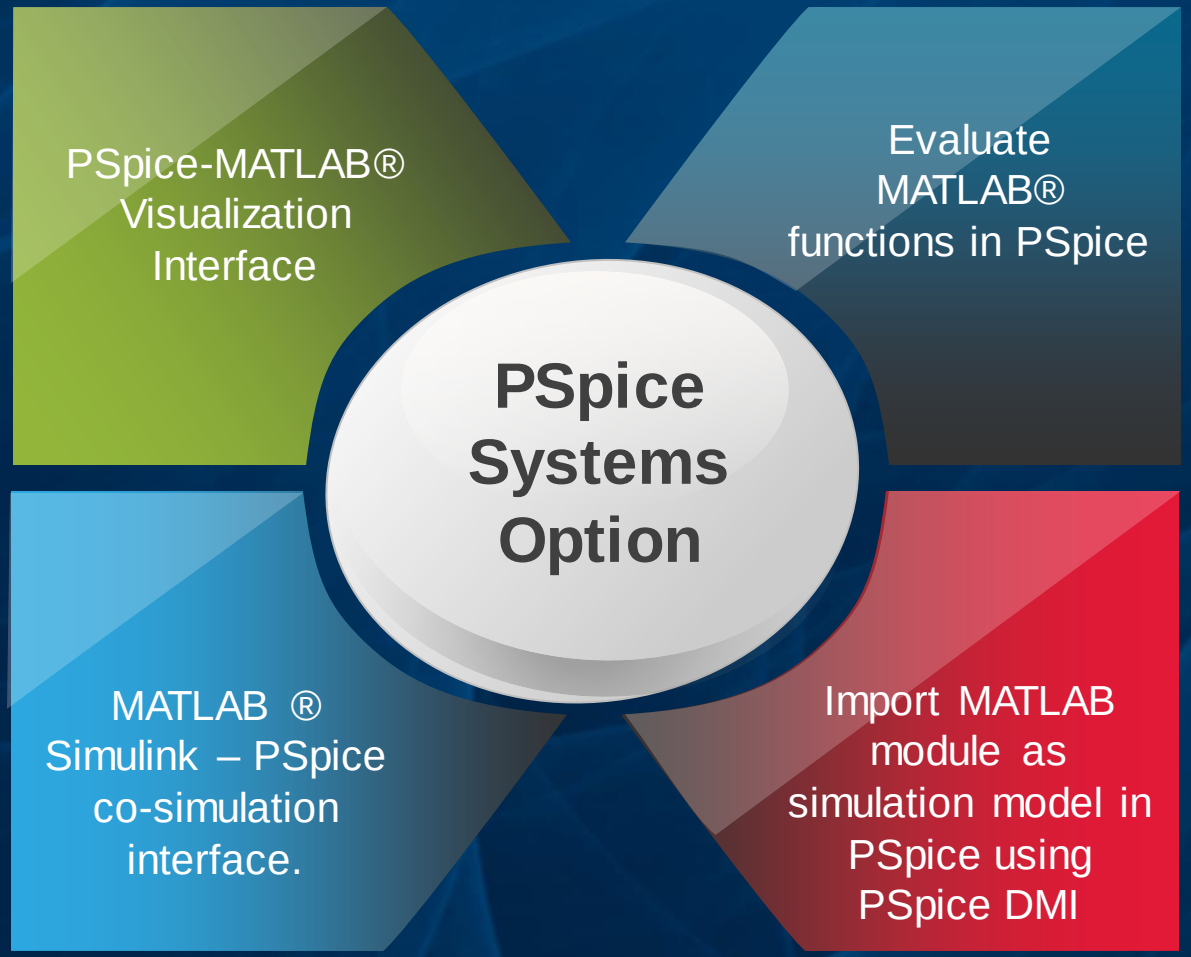
PSpice Simulink Co-Simulation– High Level User Flow



1. Initial block level implementation in Simulink
2. Implement circuit level design with PSpice
3. Integrate Block and Circuit level together using PSpice CoSim
4. Fine tune design for various operating conditions

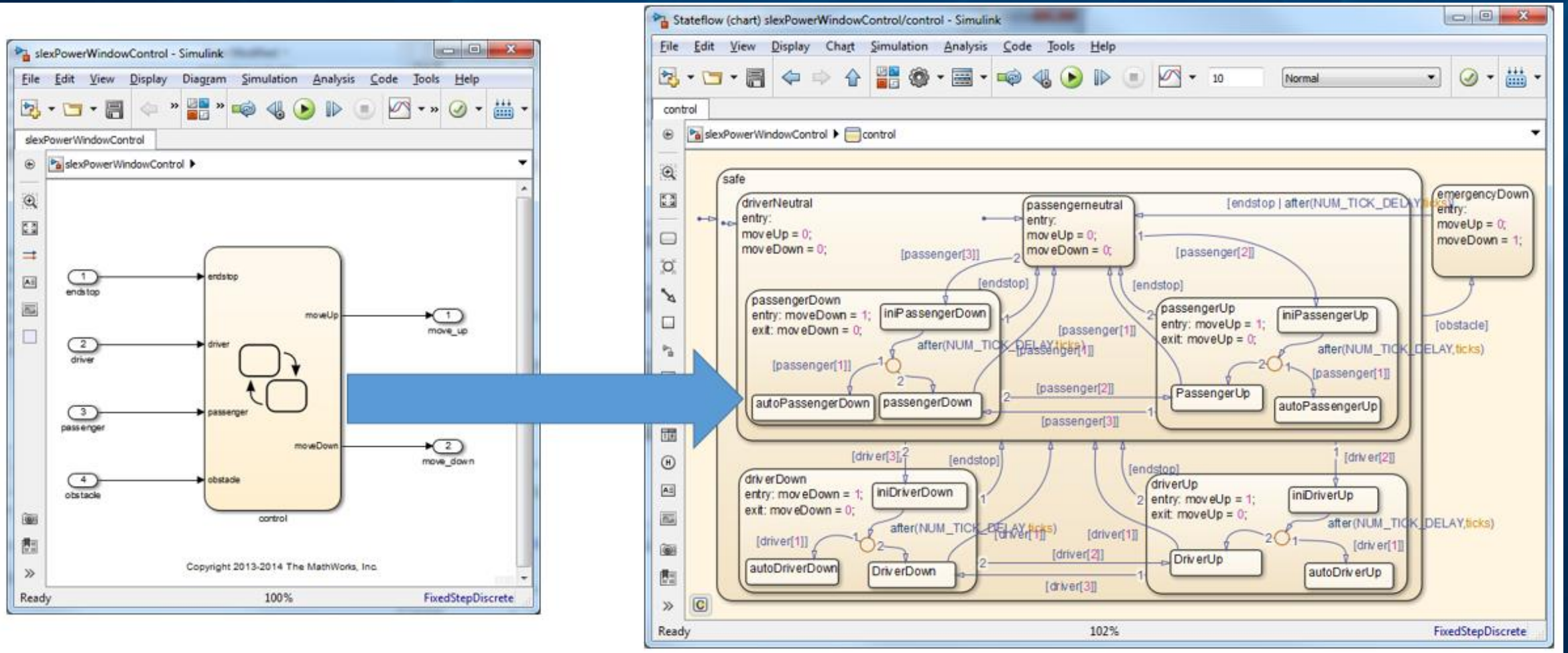
cadence

PSpice Systems Option



cadence®

Import MATLAB module as simulation model in PSpice



cadence

Summary

- Cadence and MathWorks:
  - Provide powerful tools to mine information and visualize results from simulation data
  - Allow you to “shift left” and make correct architecture decisions and reduce long, costly design iterations
  - Enable you to bring system-level considerations into your IC and PCB design and verification flows

- Next Steps:
  - Come visit Cadence Booth in the MATLAB Expo Exhibition area

cadence

Contacts

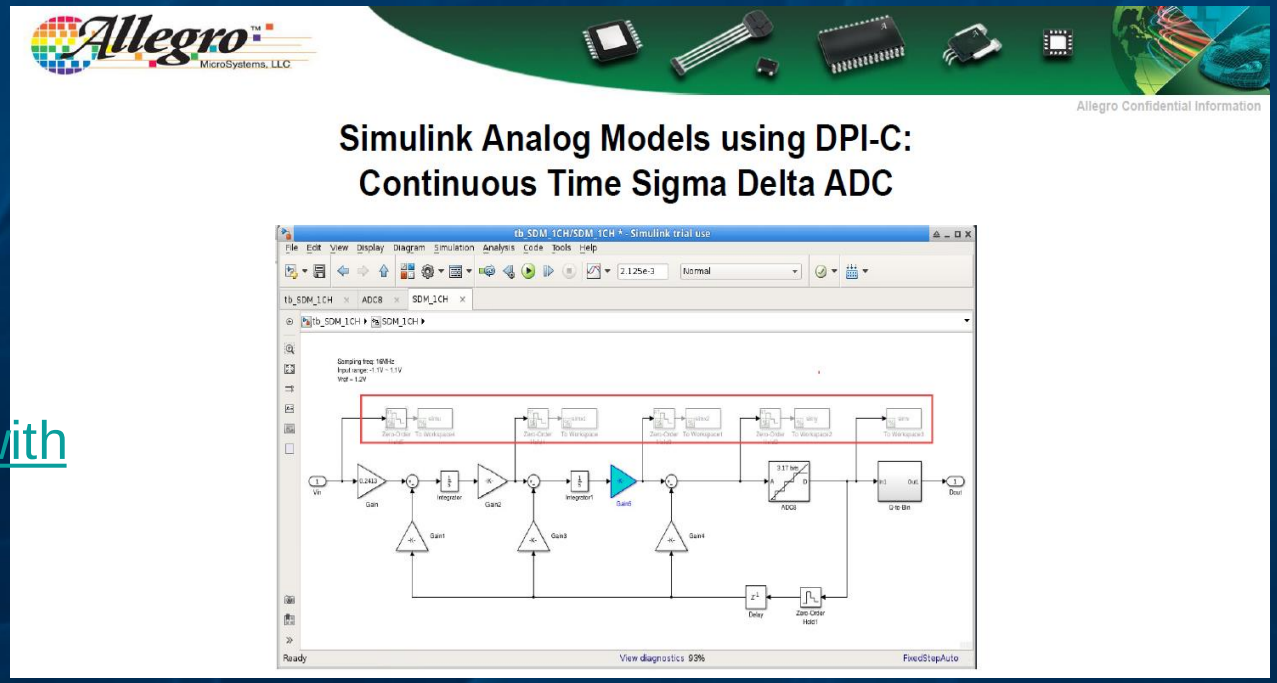
- MathWorks Contact:
  - Rajesh Berigei Rajesh.Berigei@mathworks.com

- Cadence Contacts :
  - Kishore Karnane karnane@cadence.com
  - Steve Lewis nycsteve@cadence.com

cadence

Customer References and Collateral

- Automotive ASIC Model Based Design
  Jamie Haas - Allegro Microsystems
  MATLAB Expo 2017 in San Jose



Marketing Collateral

- More Info: http://www.orcad.com/pspice-and-simulink-integration
- Webinar: Combining MATLAB and Simulink with PSpice to Streamline PCB Design
- Video: Extending the Power of MathWorks MATLAB Inside the Virtuoso ADE Suite
- Webinar: MathWorks and Cadence Design Flow for Analog/Mixed-Signal IC Development

cadence