

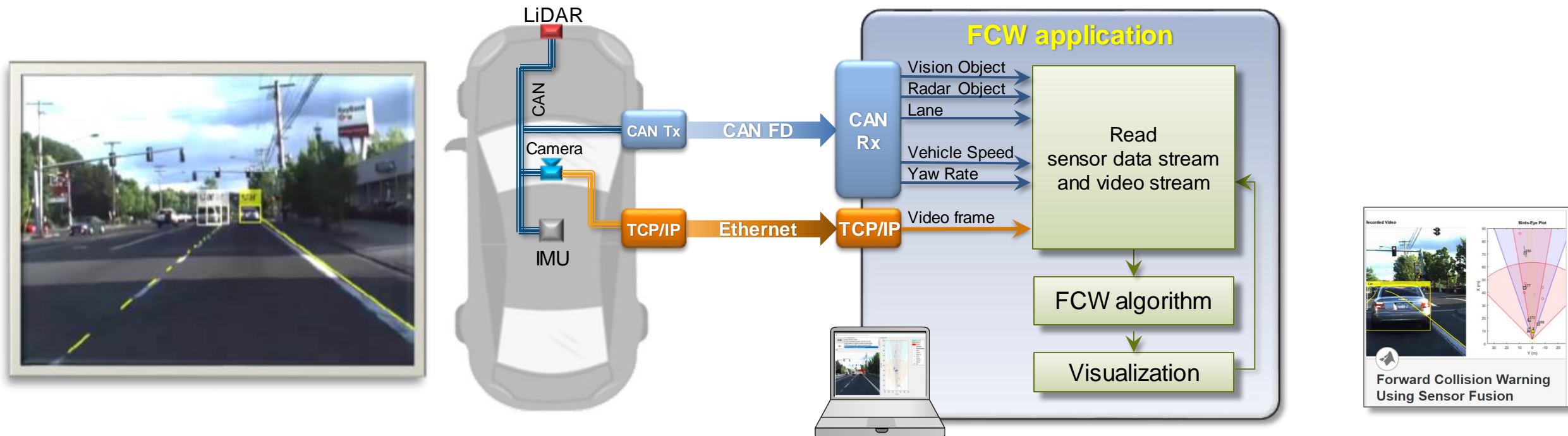
Leveraging Virtuoso/ MATLAB and PSpice/Simulink Integration for AMS Product Development

Rajesh Berigei, MathWorks

Kishore Karnane, Cadence

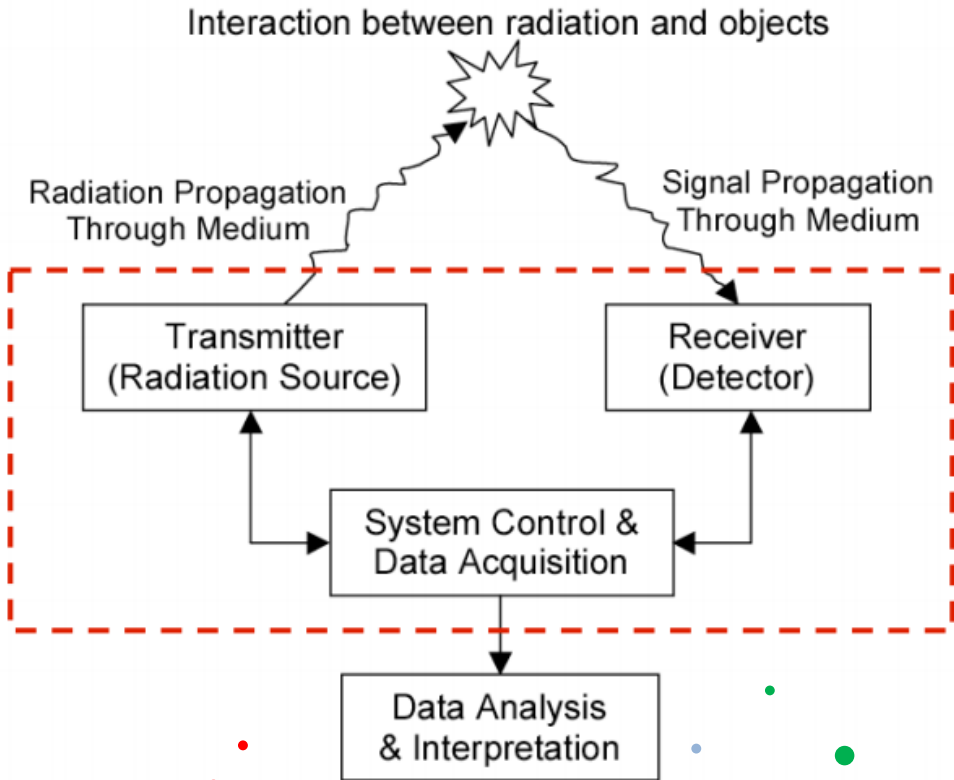
Complex Systems are Everywhere – Here is one

- 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

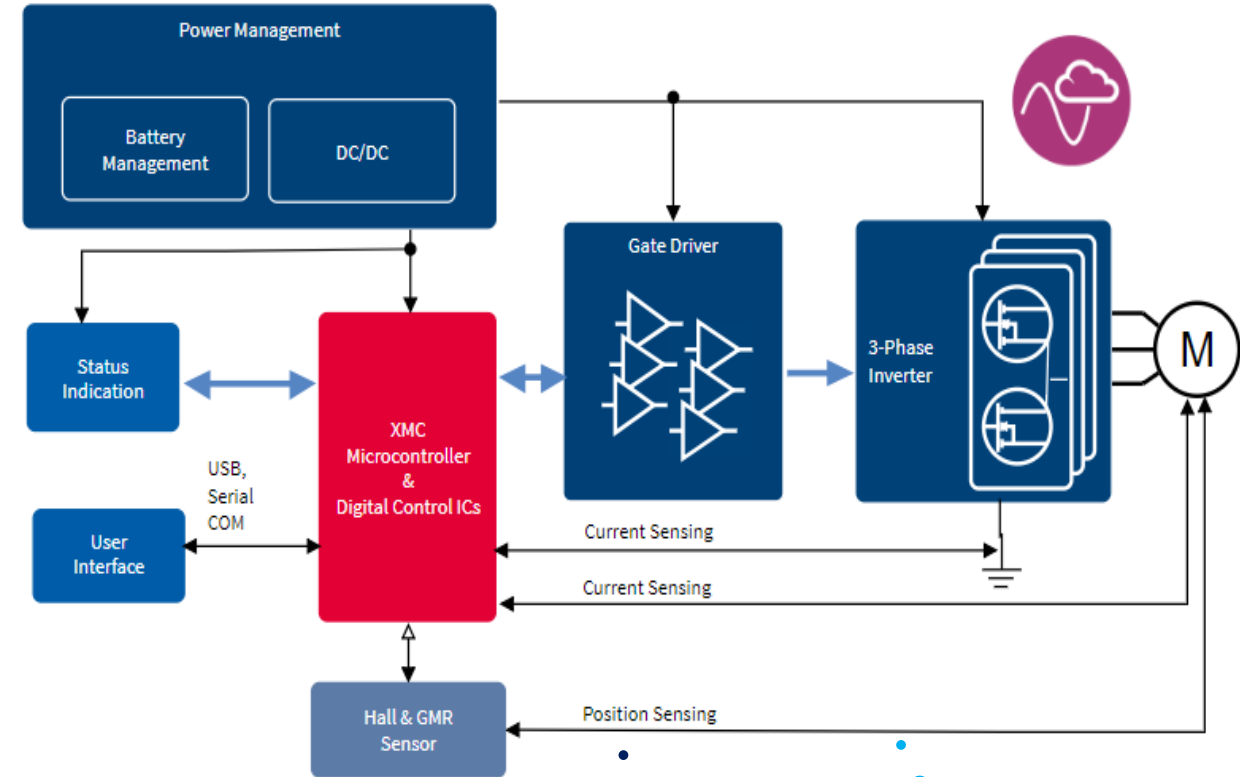


How to tame complexity?

LiDAR Tranceiver



Motor Drive Control



volume
Production
On ASIC?

Embedded
Software?

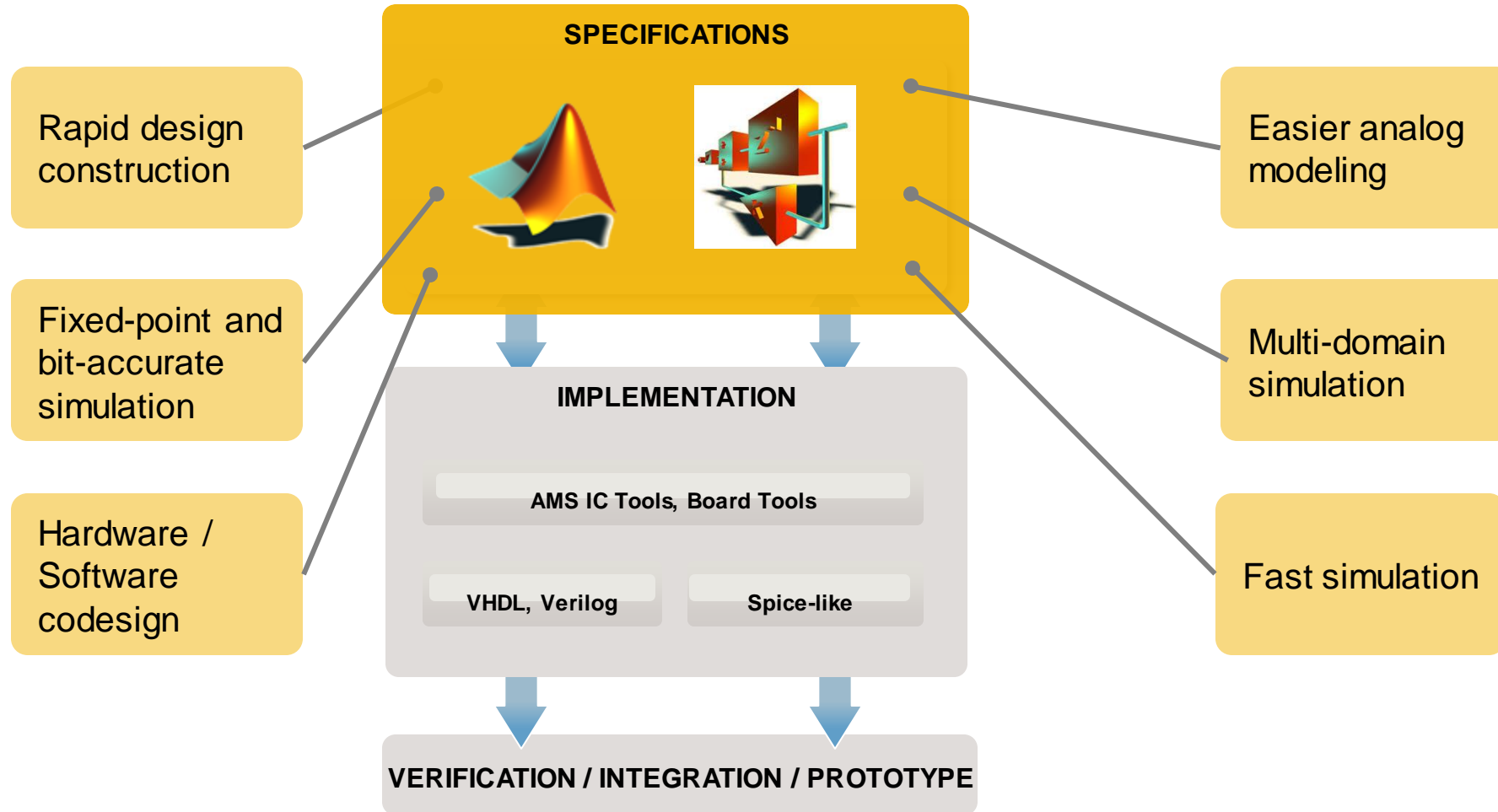
Prototype
On FPGA?

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

Sample Time Legend

Color	Description	Value
Black	Continuous	0
Grey	Fixed in Minor Step	[0,1]
Red	Discrete 1	5e-09
Magenta	Constant	Inf
Cyan	Triggered	Source: FIM
Yellow	Hybrid	N/A

PLL_Cosim - Simulink

File Edit View Display Diagram Simulation Analysis Code Tools Help

PLL_Cosim

Cosimulation Test Bench

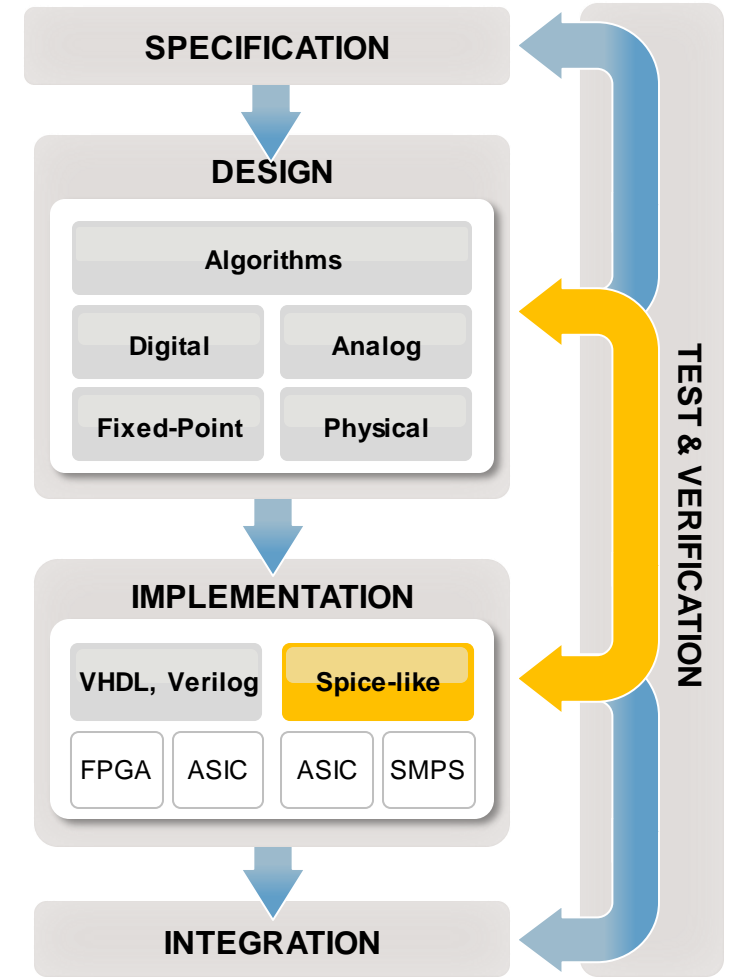
Ref Osc1 → double → boolean → ref U → var D → cadence (SimulinkCoupler) → double → Continuous-Time VCO → double → Scope

boolean → D phase/freq det. → double → 1/Kn+dCH Constant → double → Divide by N → boolean → Clock Out div #

Spectrum Analyzer: Fc=2440.00 MHz, Bw= 200.00 MHz

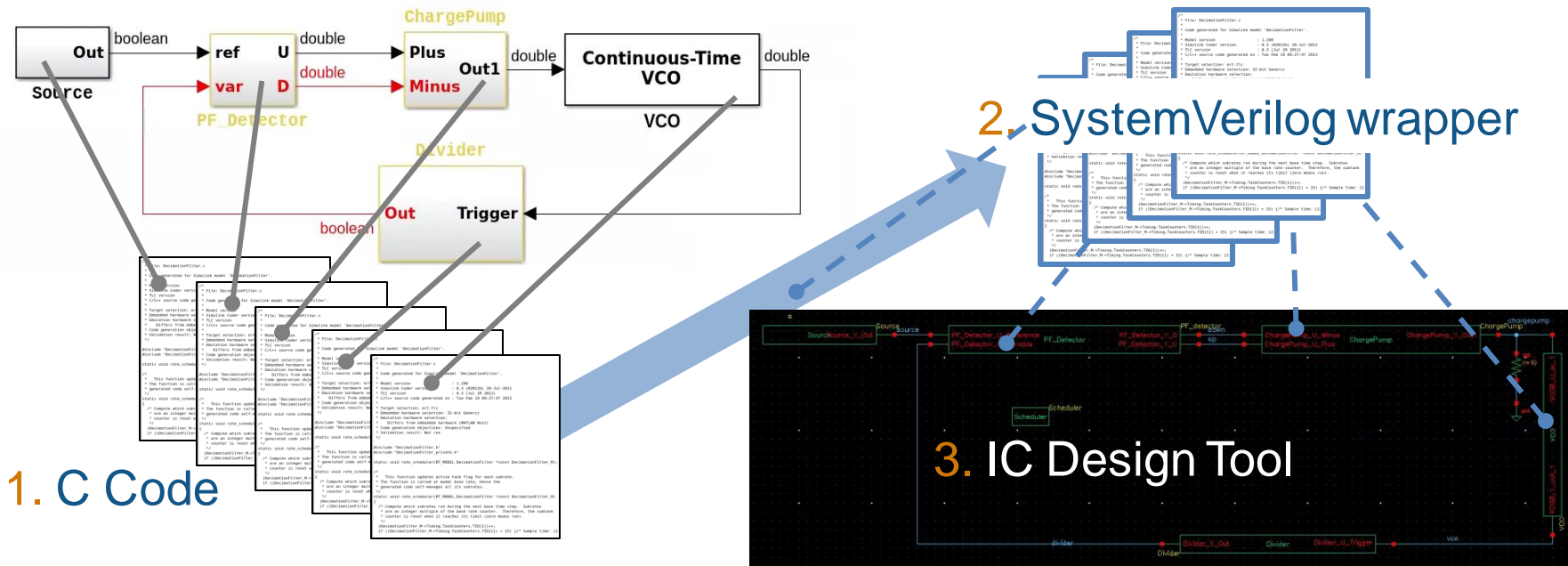
RF Spectrum Analyzer (Image Reject)

View diagnostics 97% ode45

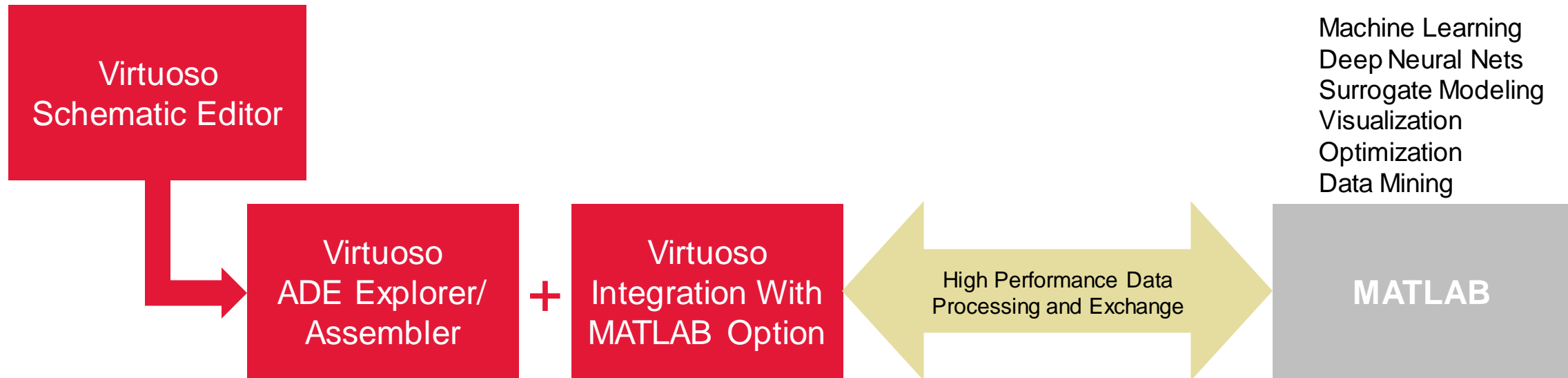


Option 2: DPI-C Compliant System Verilog Generation

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



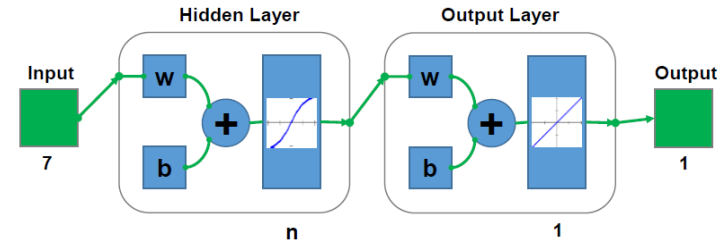
Standard design input methods including the creation of design tests inside Virtuoso ADE Explorer/Assembler/Verifier. These tests can include MATLAB expressions or make calls to MATLAB scripts for post-processing.

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.

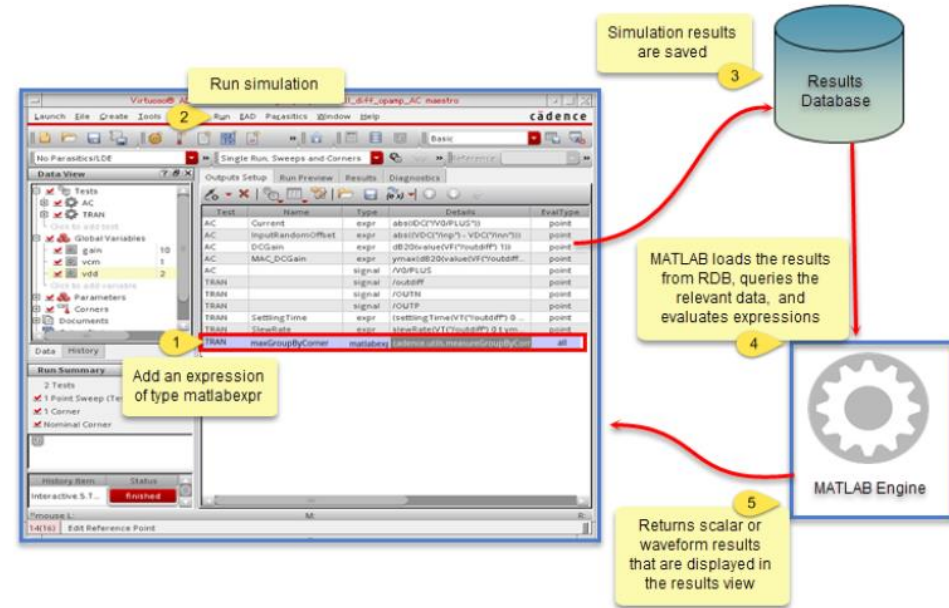


Option 3: Workflow Using Neural Net Based Design Optimization

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



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



	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	21.8M	0.5u	122

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](#)

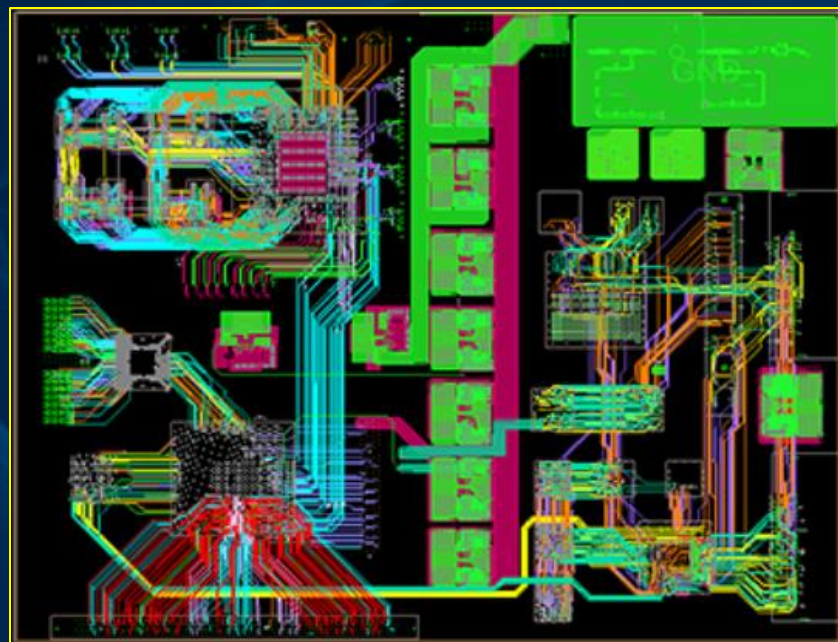
[Flyback](#)

[SEPIC](#)

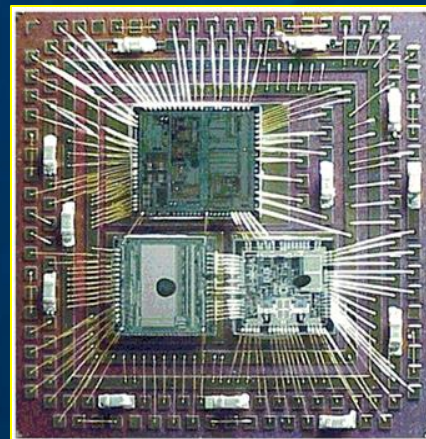
Cadence System Design Environment

Integrating IP, IC, package, PCB, and analysis

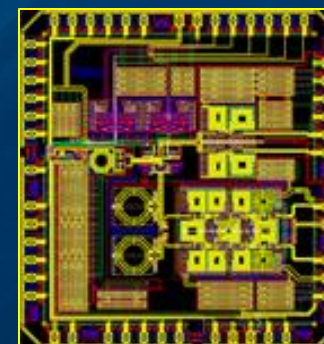
- Our software helps engineers move between various stages of electronic design so that your favorite electronic gadget is ready for the holiday rush!



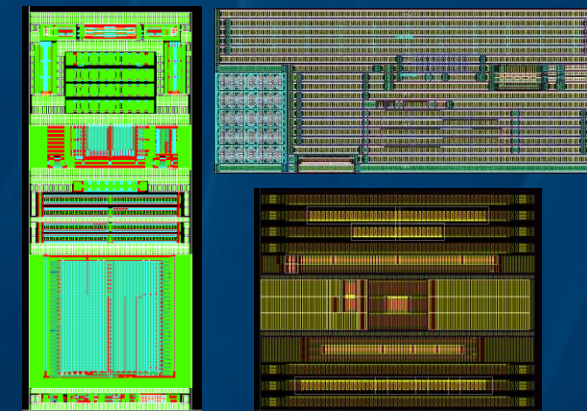
Printed Circuit Board (PCB)



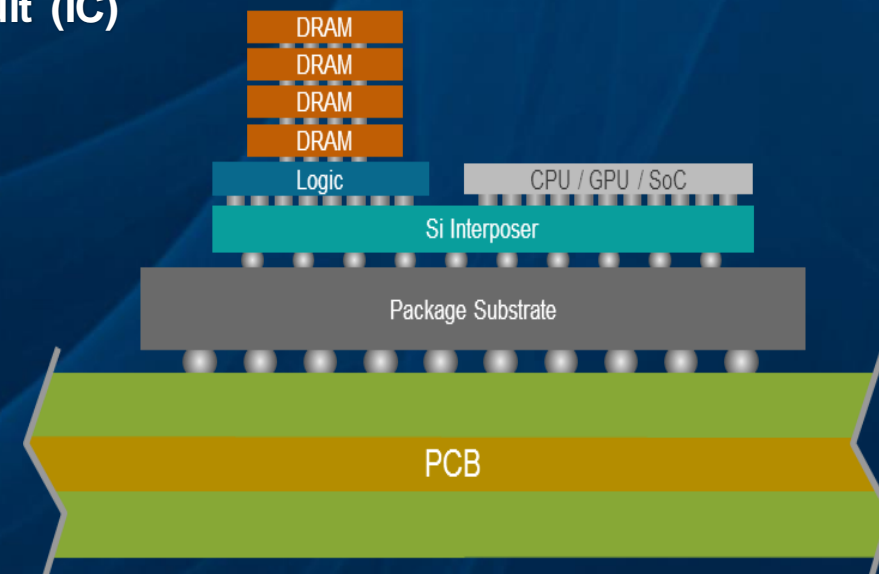
Package



Integrated Circuit (IC)



Intellectual Property (IP)



Bridging the Divide Between ICs and Systems

MathWorks system design capabilities integrated with Cadence solutions

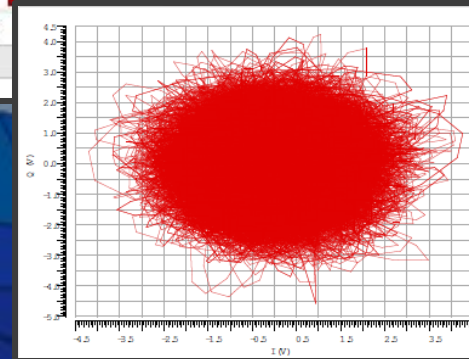


+

cādence®

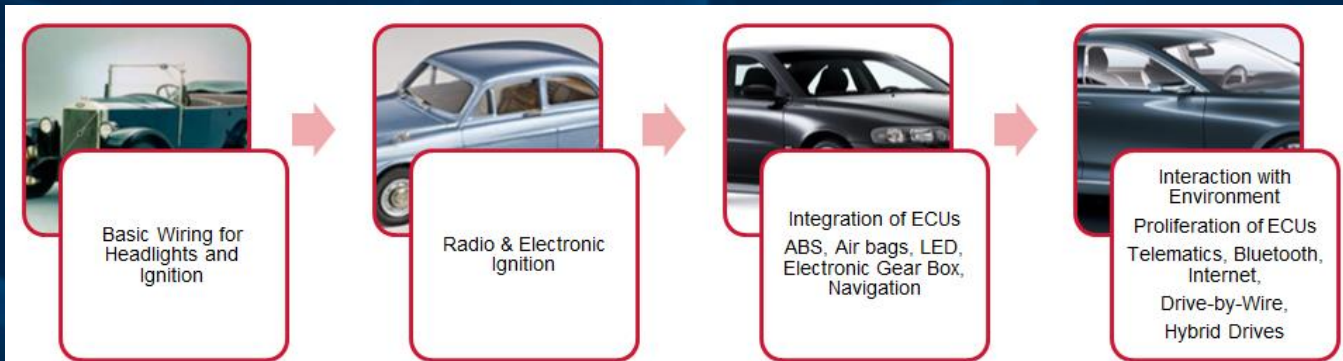
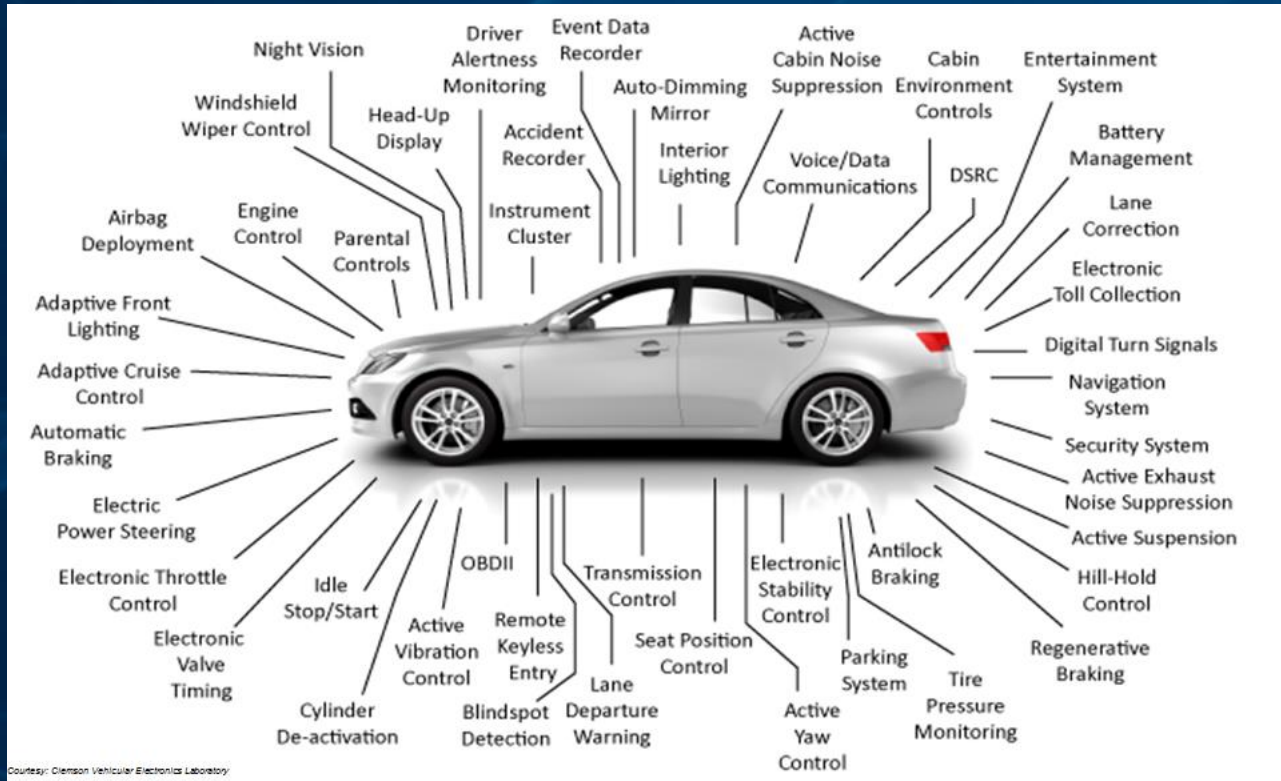
Test	Name	Type	Details	EvalType	Plot	Save
Filter	Filter	Filter	Filter	Filter	Filter	Filter
env	constplot	matlabexpr	constellation('vin')	env	env	env

High-performance IC data exchange and analysis



System-level simulation solutions for IoT and automotive applications

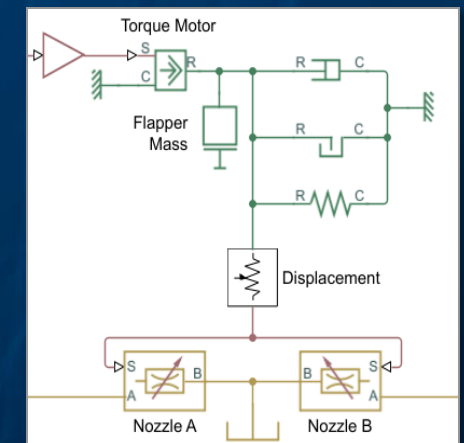
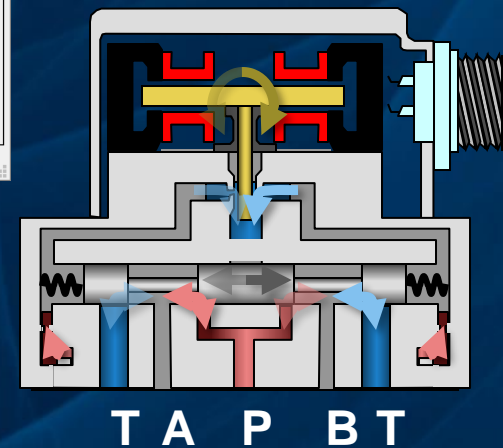
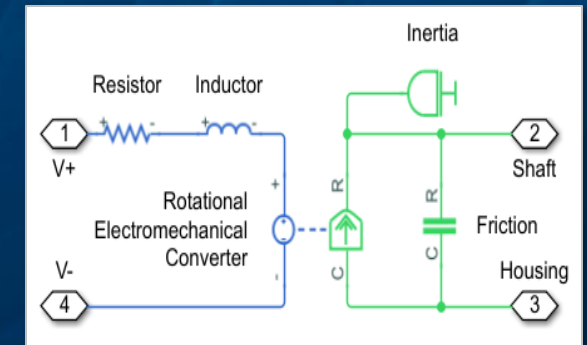
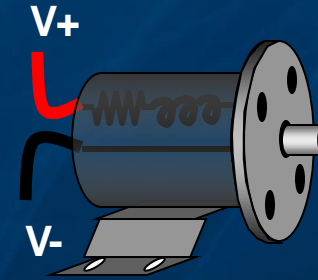
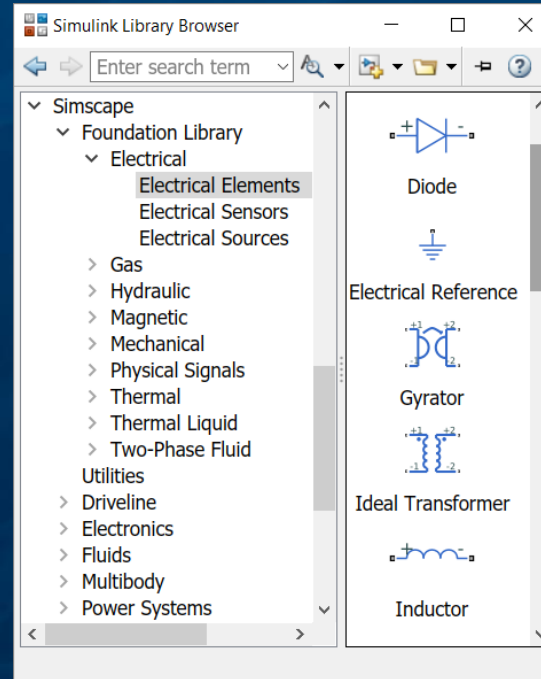
Electro-Mechanical Simulations in Automotive



- Systems Modeling
- ECU Logic Authoring
- Power Electronics
- Multi-Domain Mixed Signal Control Systems
- Sensors
- Network Enabled
- Embedded Software

Easily Integrate MATLAB Models for Mechanical Components

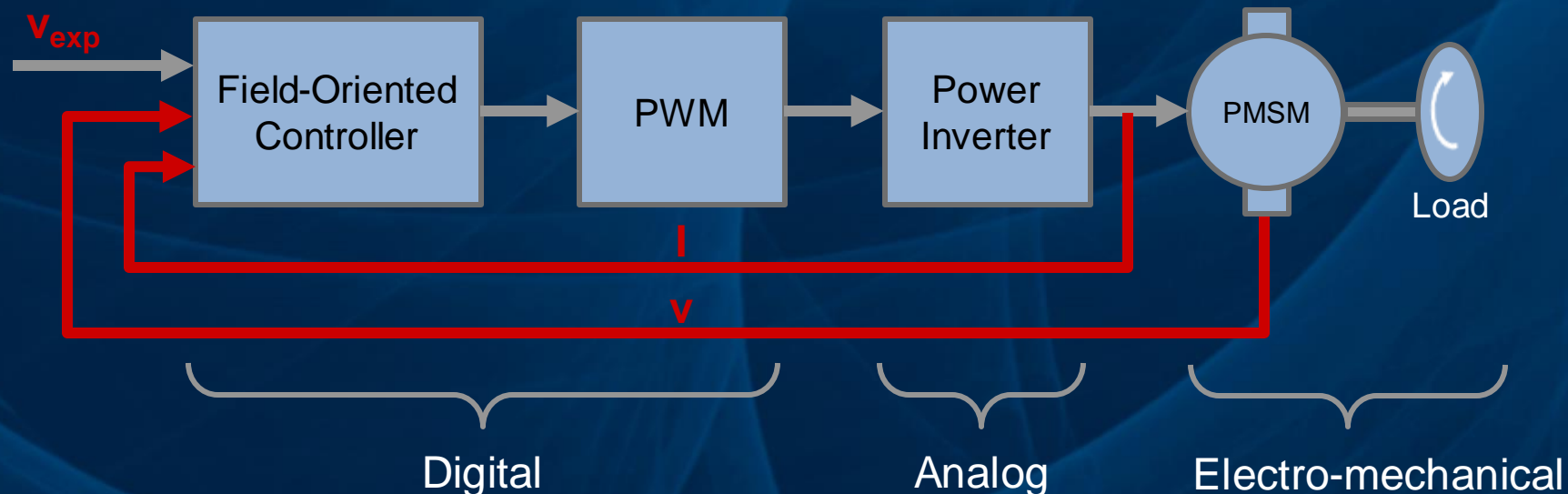
- Eases process of modeling Physical Systems
 - Build models that reflect structure of physical system
 - Leverage MATLAB to create reusable models
- An electrohydraulic servo-valve example
 - Shows multidomain modeling, with electrical, mechanical, and hydraulic components



Permanent Magnet Synchronous Motor Drives

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

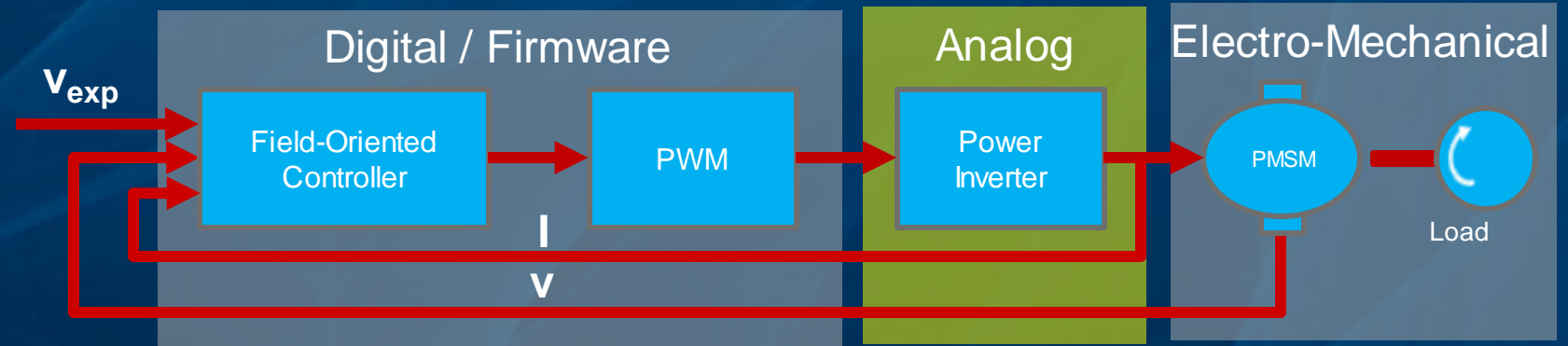
Analog/Mixed-Signal Design



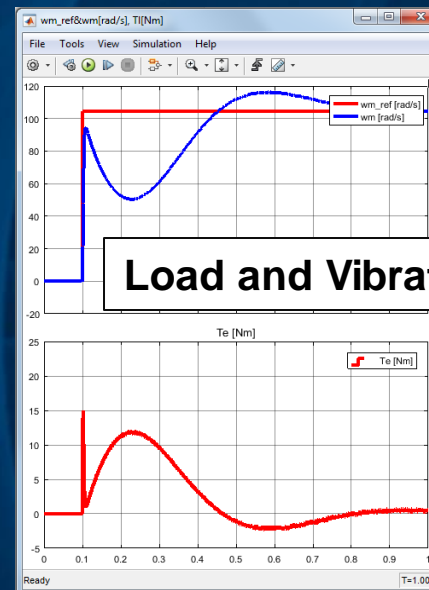
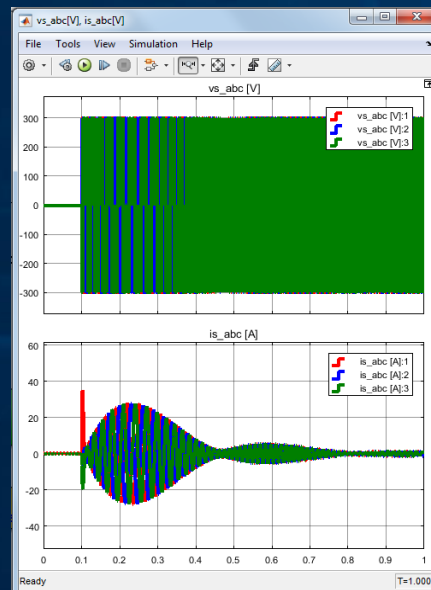
Automotive System Design for Electric Vehicles

MATLAB / Simulink / PSpice integration

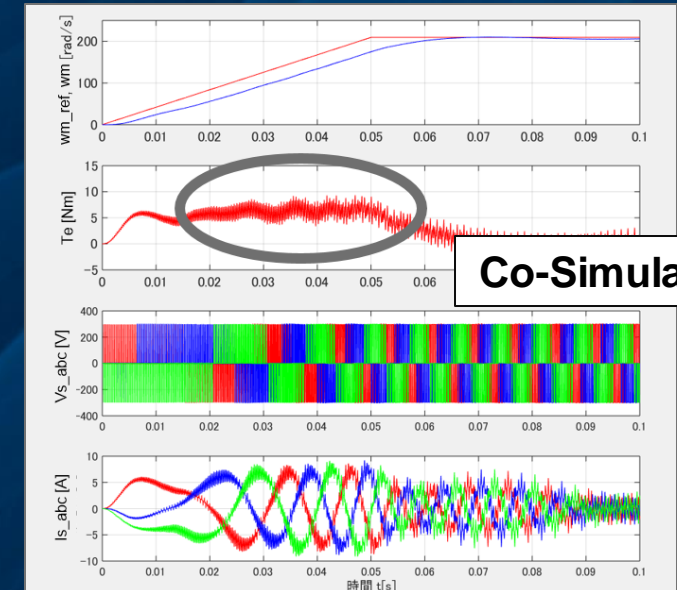
- From actuators to electric vehicle motors
- Acceleration of 0-60mph in 2.7 secs



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

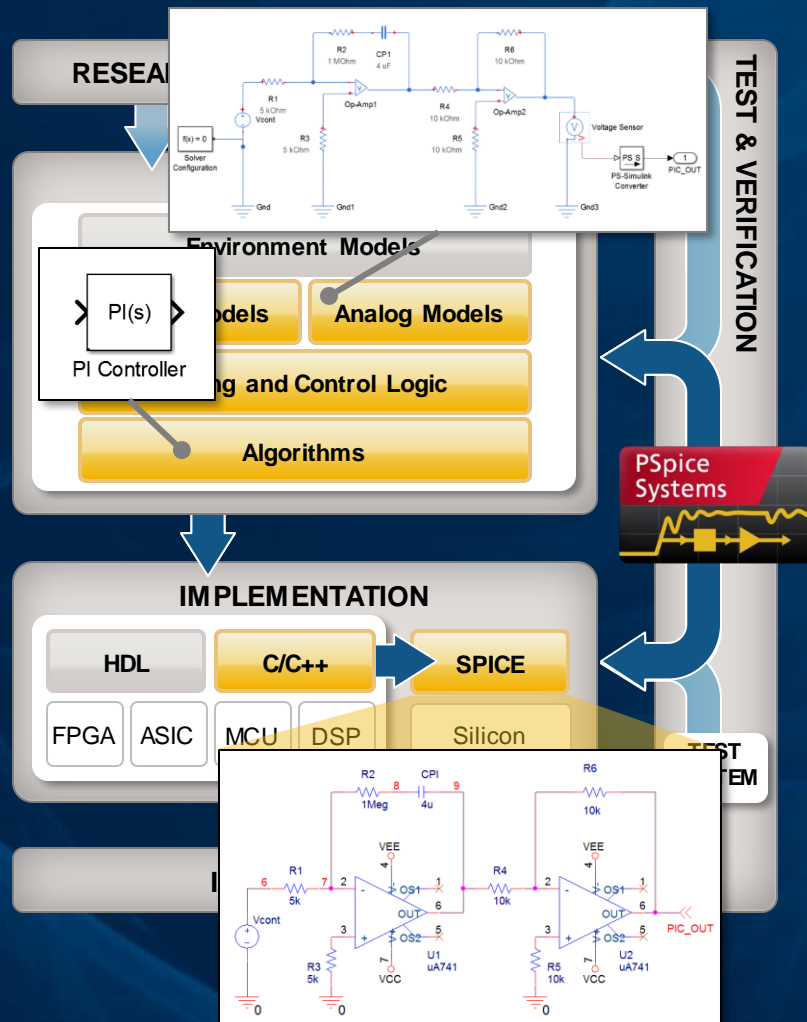


Load and Vibration



Co-Simulation

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

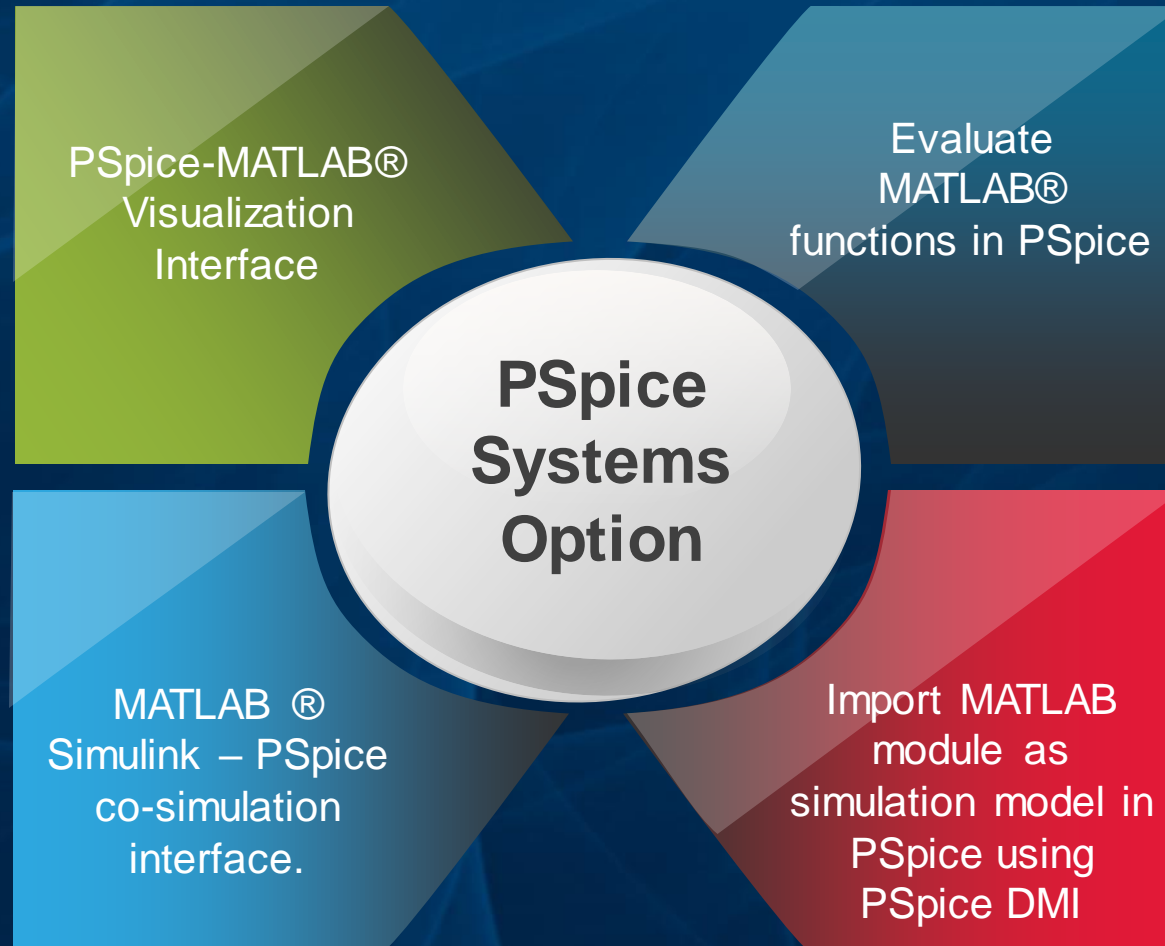
PSpice Simulink Co-Simulation - Benefits

- 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

MATLAB & Simulink

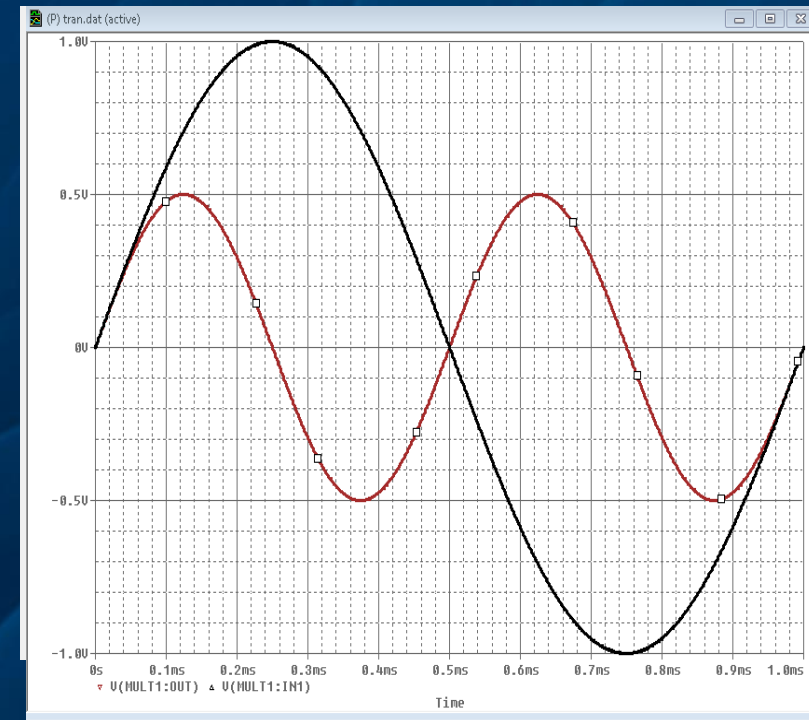
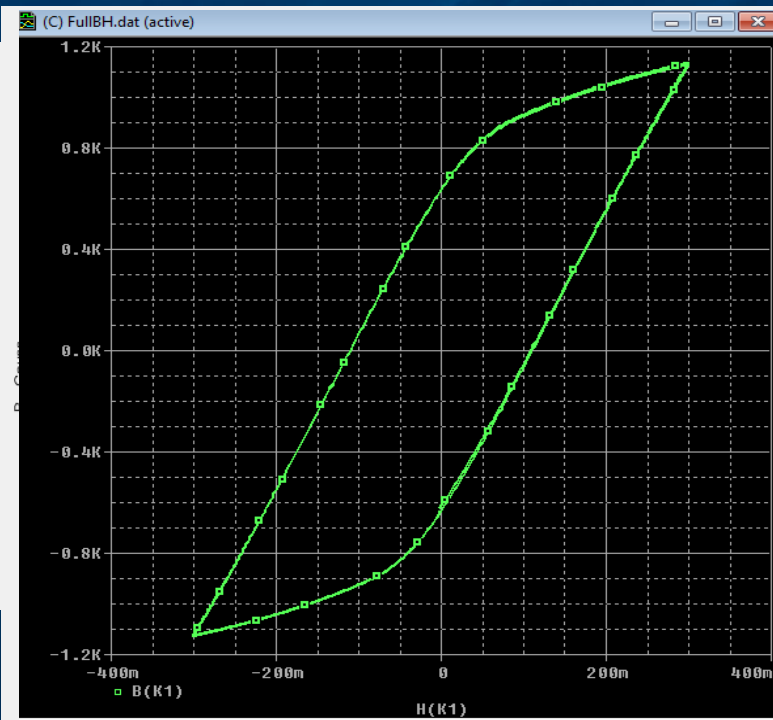
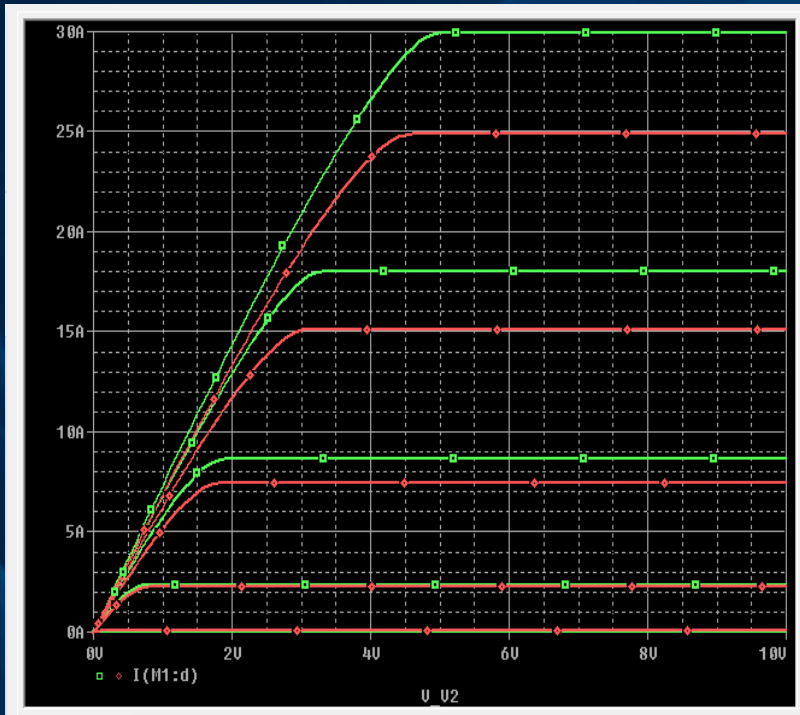


PSpice Systems Option

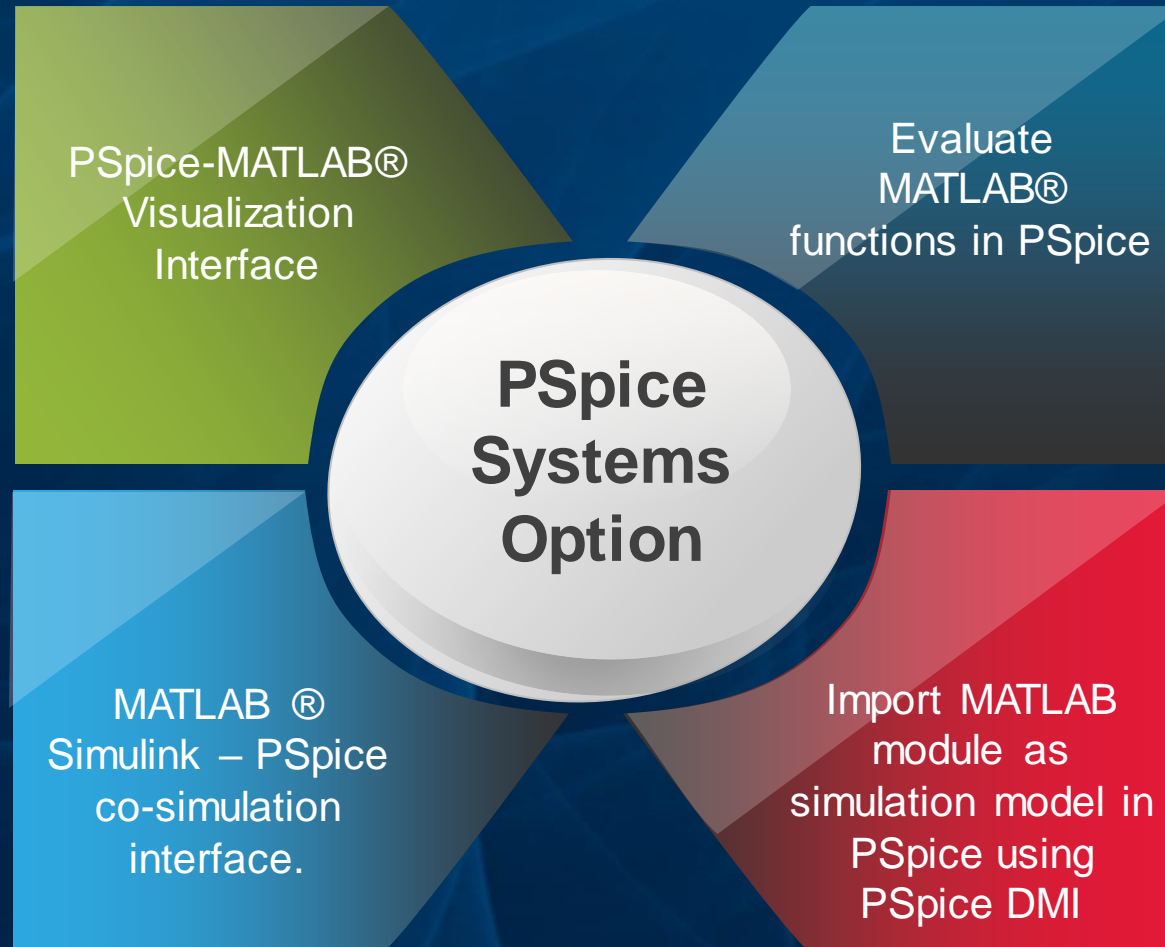


Examples: PSpice – MATLAB Visualization Interface

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



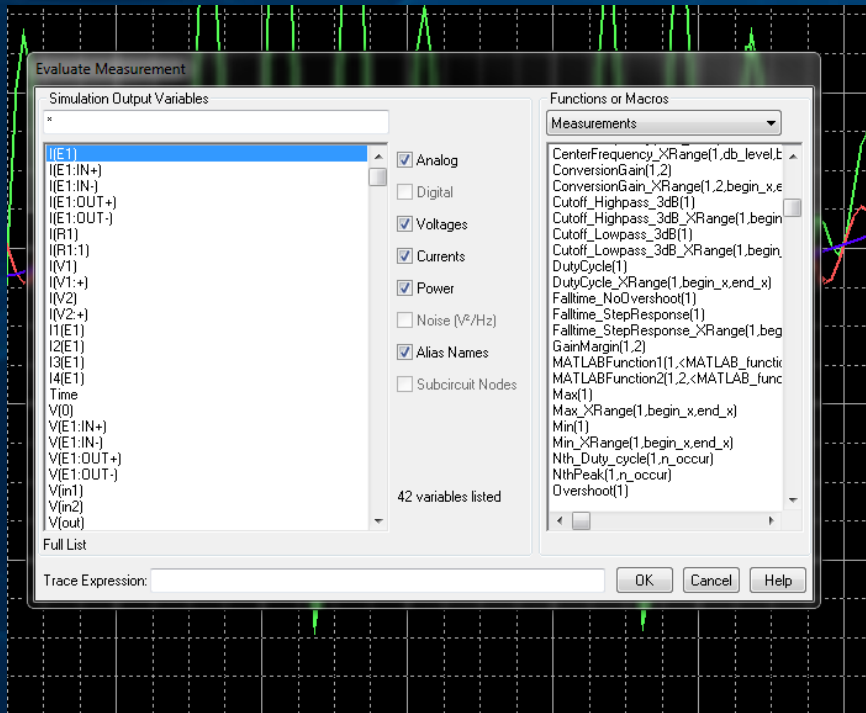
PSpice Systems Option



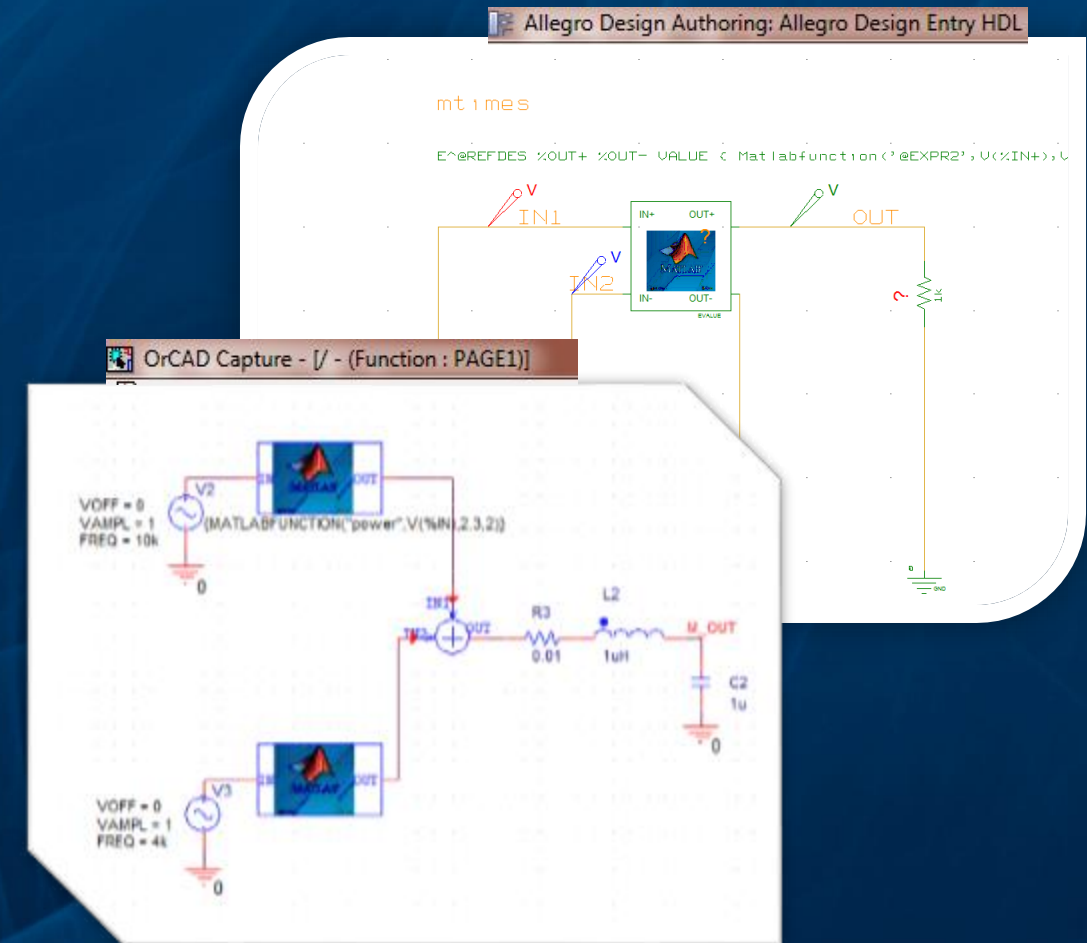
Evaluate MATLAB functions in PSpice

Include MATLAB functions for measurements

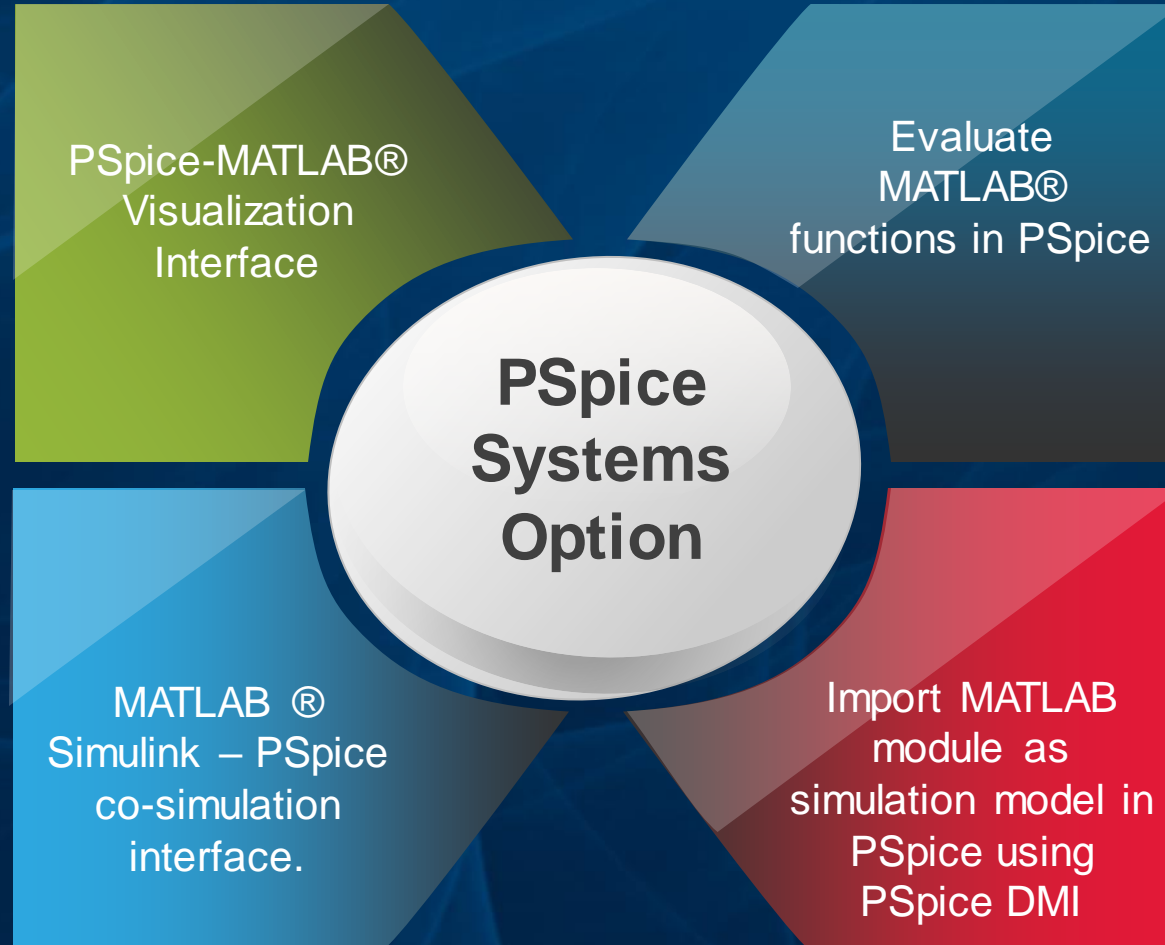
Use MATLAB functions in simulation



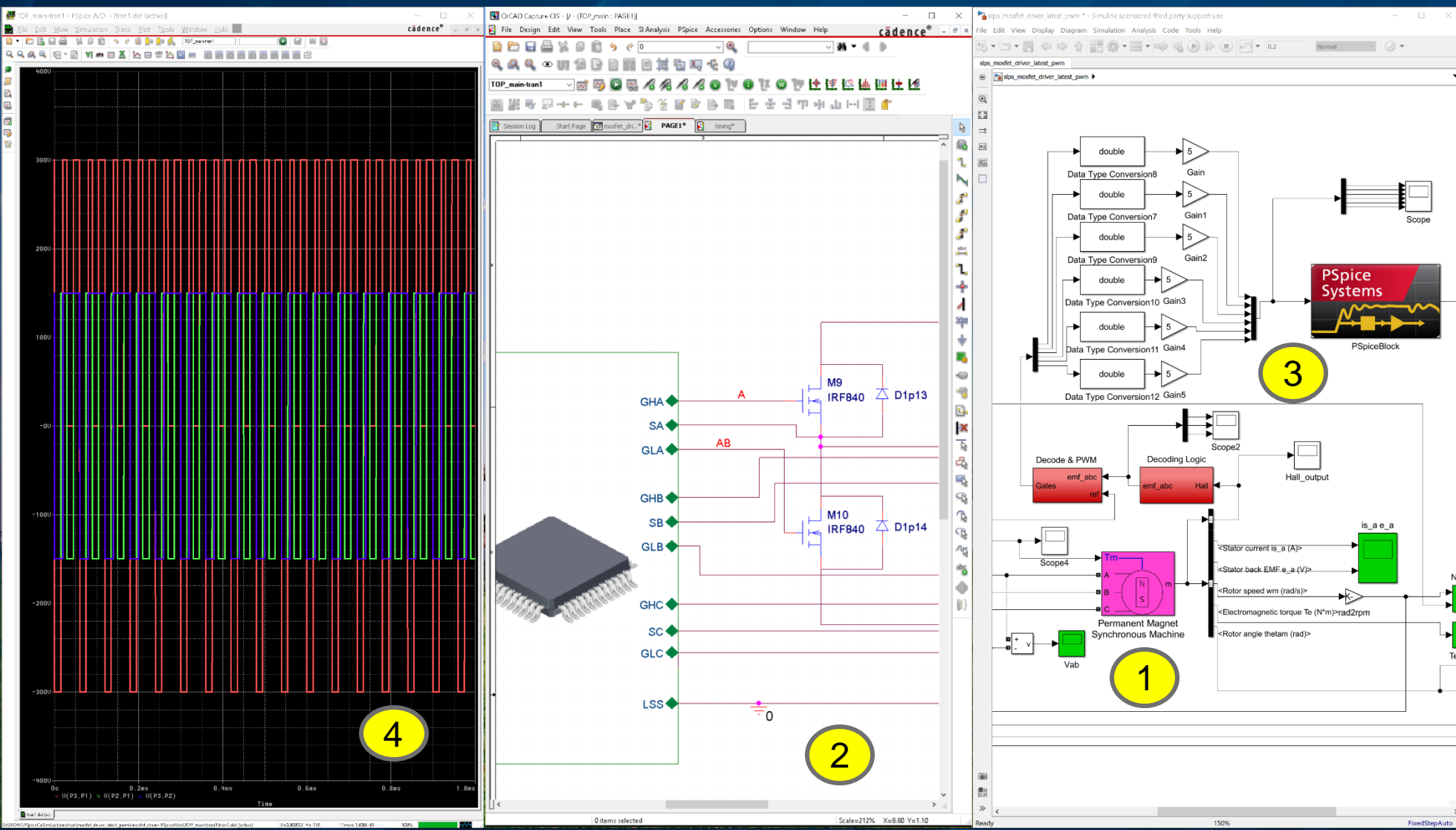
	Evaluate	Measurement	Value
▶	<input checked="" type="checkbox"/>	matlabFunction1(V(RLC),risetime)	8.95455
	<input checked="" type="checkbox"/>	matlabFunction1(V(ABM3:IN),peak2rms)	1.35111
	<input checked="" type="checkbox"/>	matlabFunction1(V(R2:2),peak2peak)	71.99985m



PSpice Systems Option



PSpice Simulink Co-Simulation– High Level User Flow



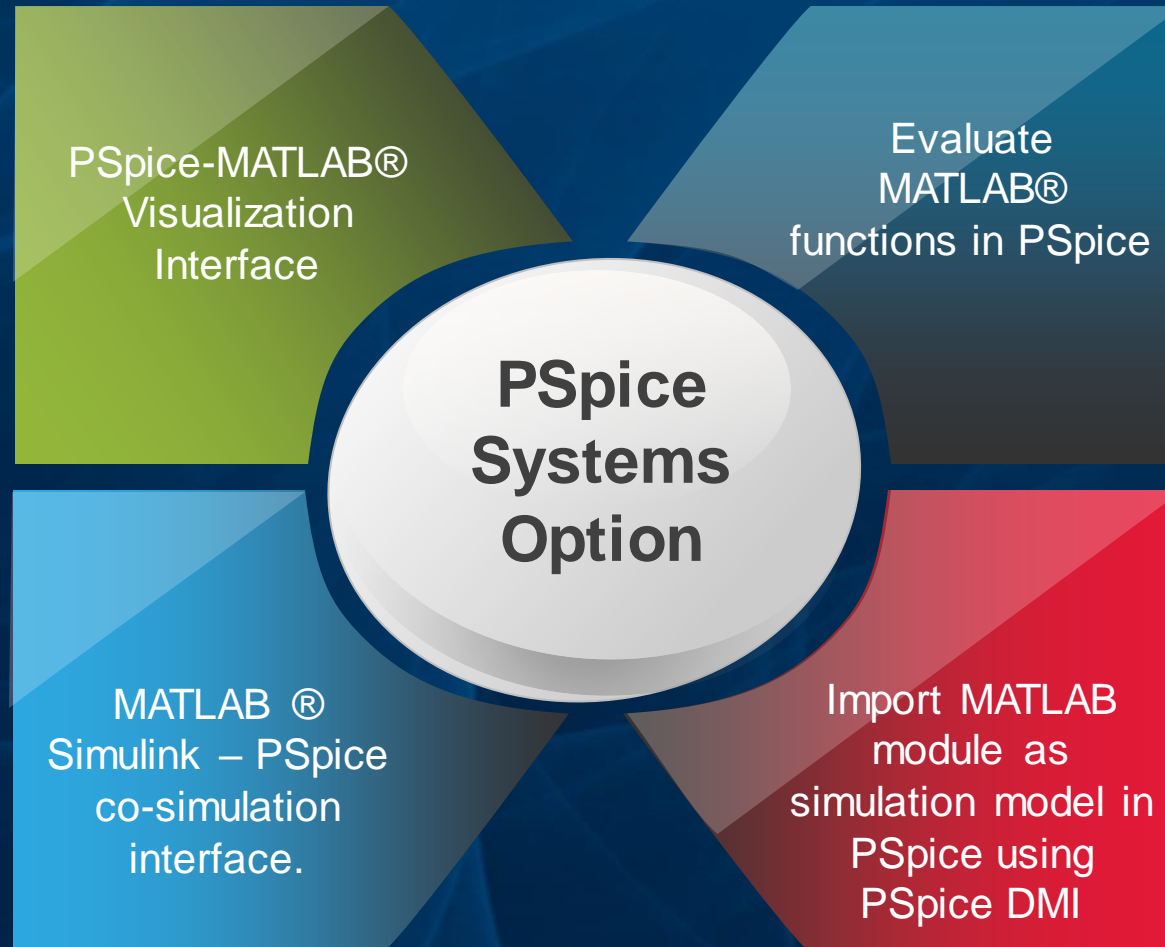
Initial block level implementation in Simulink 1

Implement circuit level design with PSpice 2

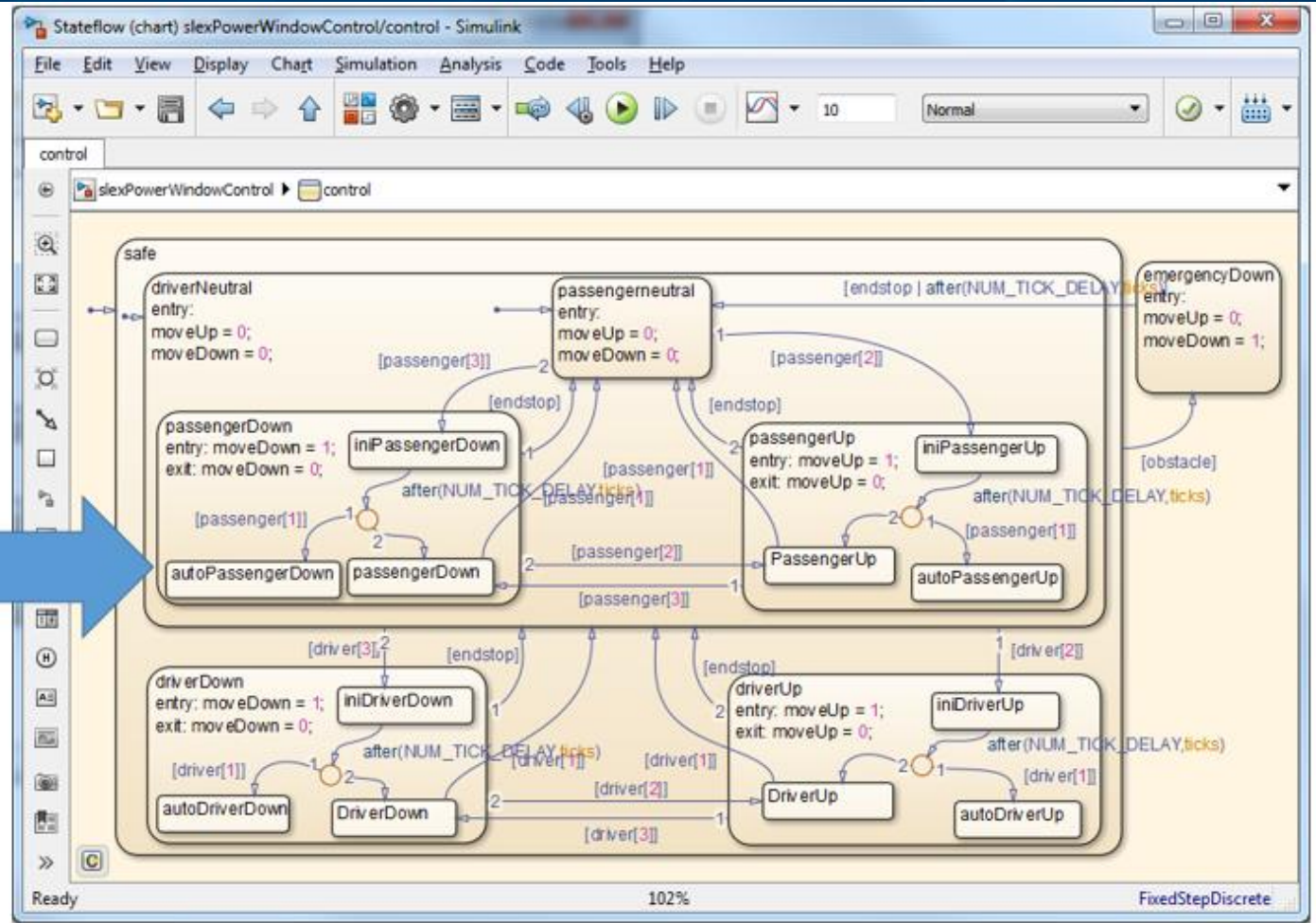
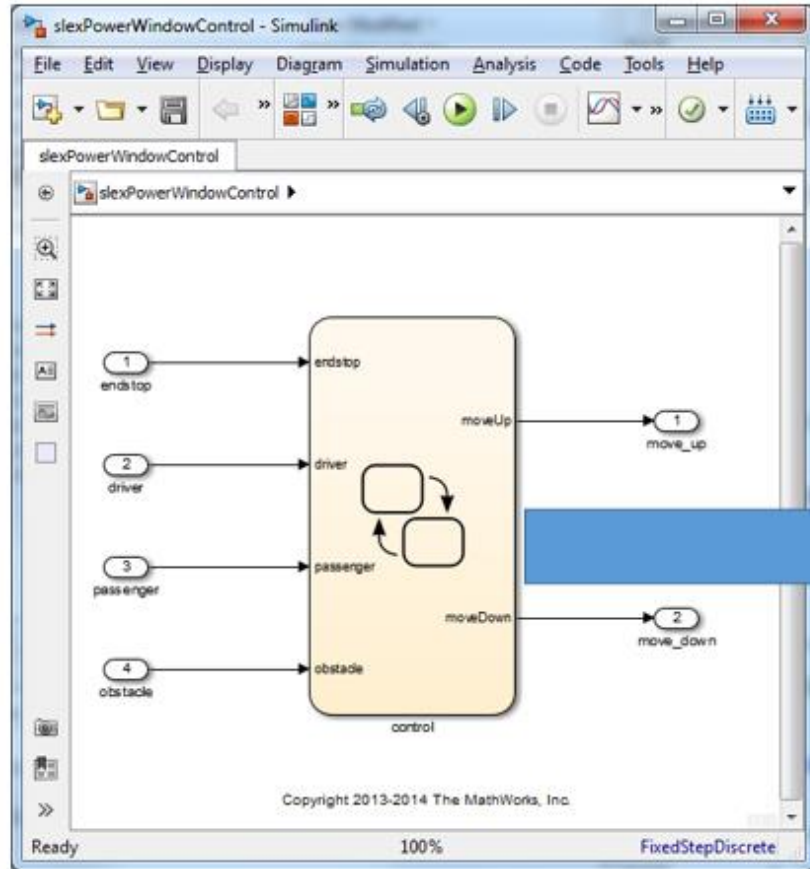
Integrate Block and Circuit level together using PSpice CoSim 3

Fine tune design for various operating conditions 4

PSpice Systems Option



Import MATLAB module as simulation model in PSpice



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

Contacts

- MathWorks Contact:
 - Rajesh Berigei Rajesh.Berigei@mathworks.com
- Cadence Contacts :
 - Kishore Karnane karnane@cadence.com
 - Steve Lewis nycsteve@cadence.com

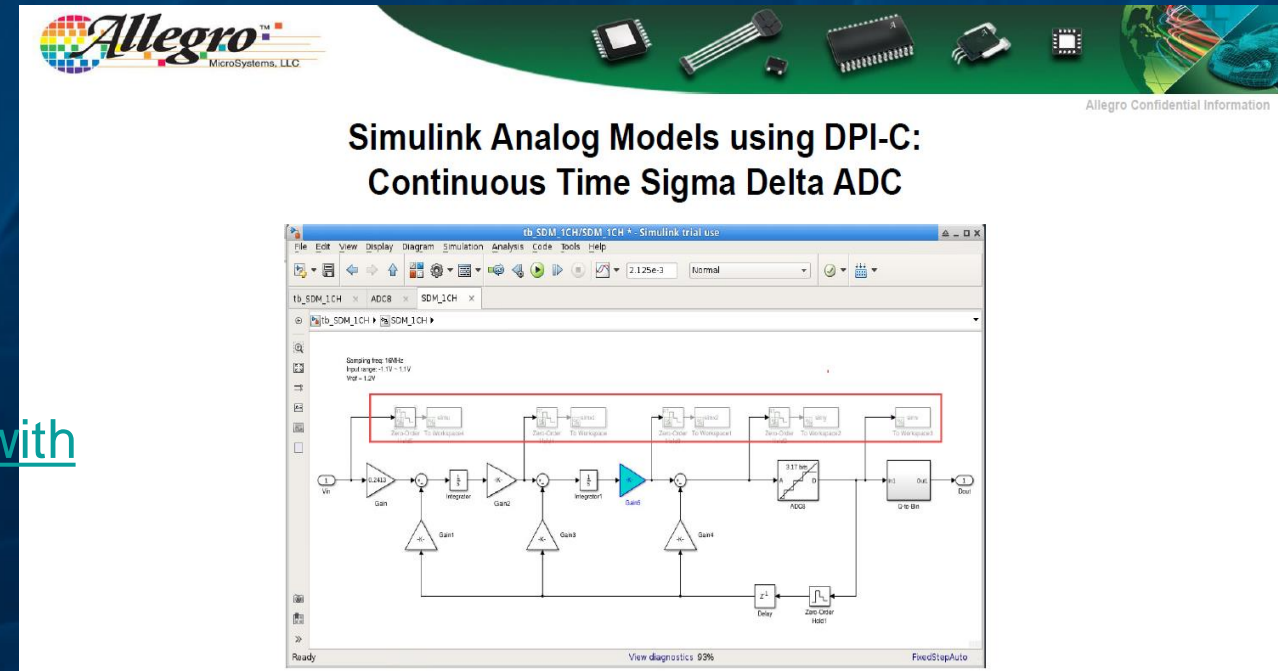


Customer References and Collateral

- [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](#)



cādence®

©2018 Cadence Design Systems, Inc. All rights reserved worldwide. Cadence, the Cadence logo, and the other Cadence marks found at www.cadence.com/go/trademarks are trademarks or registered trademarks of Cadence Design Systems, Inc. All other trademarks are the property of their respective owners.