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

Interaction between radiation and objects

Radiation Propagation Through Medium

Signal Propagation Through Medium

Transmitter (Radiation Source)

Receiver (Detector)

System Control & Data Acquisition

Data Analysis & Interpretation

Power Management

Battery Management

DC/DC

Gate Driver

3-Phase Inverter

XMC Microcontroller & Digital Control ICs

Hall & GMR Sensor

Current Sensing

Status Indication

USB, Serial COM

User Interface

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
- Code Generation
- Post-Processing

Option 1
Option 2
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. 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.

- Machine Learning
- Deep Neural Nets
- Surrogate Modeling
- Visualization
- Optimization
- Data Mining
Option 3: Workflow Using Neural Net Based Design Optimization

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 Prezcaler
Fractional N PLL

ADC

ADC Tutorial including Cadence Incisive Digital Co:
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!
Bridging the Divide Between ICs and Systems
MathWorks system design capabilities integrated with Cadence solutions

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

Basic Wiring for Headlights and Ignition
Radio & Electronic Ignition
Integration of ECUs ABS, Air bags, LED, Electronic Gear Box, Navigation
Interaction with Environment Proliferation of ECUs Telematics, Bluetooth, Internet, Drive-by-Wire, Hybrid Drives
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**

![Diagram of Field-Oriented Controller, PWM, Power Inverter, PMSM, and Load with Digital, Analog, and Electro-mechanical components]
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-synchro machine for motor powertrains

Digital / Firmware
Field-Oriented Controller → PWM

Analog
Power Inverter

Electro-Mechanical
PMSM → Load

Load and Vibration
Co-Simulation

©2018 Cadence Design Systems, Inc. All rights reserved.
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
PSpice Systems Option

- PSpice-MATLAB® Visualization Interface
- Evaluate MATLAB® functions in PSpice
- MATLAB® Simulink – PSpice co-simulation interface.
- Import MATLAB module as simulation model in PSpice using PSpice DMI
Examples: PSpice – MATLAB Visualization Interface

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

- PSpice-MATLAB® Visualization Interface
- Evaluate MATLAB® functions in PSpice
- MATLAB ® Simulink – PSpice co-simulation interface.
- Import MATLAB module as simulation model in PSpice using PSpice DMI
Evaluate MATLAB functions in PSpice

Include MATLAB functions for measurements

Use MATLAB functions in simulation
PSpice Systems Option

- PSpice-MATLAB® Visualization Interface
- Evaluate MATLAB® functions in PSpice
- MATLAB ® Simulink – PSpice co-simulation interface.
- Import MATLAB module as simulation model in PSpice using PSpice DMI
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
PSpice Systems Option

PSpice-MATLAB® Visualization Interface

Evaluate MATLAB® functions in PSpice

MATLAB® Simulink – PSpice co-simulation interface.

Import MATLAB module as simulation model in PSpice using PSpice DMI
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
Marketing Collateral

- 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