

Designing for the Internet of Things with Cadence PSpice A/D Technology

By Alok Tripathi, Software Architect, Cadence

The Cadence® PSpice® A/D release 17.2-2016 offers a comprehensive feature set to address design challenges in IoT sensors, controllers, and actuators. With system simulation and modeling technology that enables a unified design environment for mixed-signal design, PSpice A/D can help you deliver a high-quality product within your time-to-market window.

Contents

Introduction	1
Key Challenges in IoT Design	1
Simulating IoT-Enabled Devices	2
Sensors.....	2
Controllers.....	3
Actuators	4
Summary.....	5

Introduction

In almost every domain, there is an urgent need for devices to connect to some form of a network, or, in other words, to make these devices internet enabled. There is an explosion in the number of devices getting connected to the internet every day. These devices are known as the “Internet of Things” (IoT), and their omnipresence is increasing with every passing day. This evolution is not limited to one or a few types of application segments, instead, it impacts almost all classes of designs and applications. This change touches every aspect of our day-to-day life, ranging from wearable devices, home appliances, and security systems, to sensors monitoring industrial processes. Due to the very nature of devices and the need for them to operate in diverse operating conditions, these devices are exposed to unprecedented, highly varying external factors, such as wide thermal fluctuations, electrical noise, fluids, moisture, vibrations, and shock. Ensuring reliable operation of these systems for their entire life cycle is not only crucial but critical for survival in a highly competitive market space.

IoT presents unprecedented opportunities across industry sectors. Ride this wave with Cadence PSpice A/D Release 17.2-2016!

Key Challenges in IoT Design

What do these challenges mean for a designer, and how do they impact the traditional design and simulation flow? Let’s look at an IoT device from a system perspective, and at its key components. At a high level, any IoT device can be divided into the following three key blocks from a design and simulation perspective:

- Sensor
- Controller
- Actuator



Above these three key functional blocks, there are standard communication layers, such as ZigBee, RFID, Bluetooth, BACnet, and CoAP, that enable communication, or establish a connection, to the internet. In some cases, you may have sensors, controllers, and actuators all housed in single body, and in some cases, these could be part of different modules, but connected to each other.

Simulating IoT-Enabled Devices

From a design-simulation perspective, designs of IoT-enabled devices require the ability to:

- Design all three blocks of IoT devices
- Simulate the complete system

Traditionally, designers rely on SPICE simulation for designing a system for reliable and predictable operation. A persistent question in today's times for designers is: Can we still rely on SPICE simulation for designing the newer generation IoT-enabled devices? And the answer is, yes. The Cadence PSpice A/D release 17.2-2016 enables designers to effectively overcome IoT design challenges. The system aggregators typically simulate the system by aggregating available models of off-the-self components that are readily available in the market. For several cases, you might not have a model available, or the available model may not be a traditional SPICE model. The PSpice A/D release 17.2-2016 offers you additional modeling capabilities to meet this specific need. The next section presents an overview of the unique capabilities PSpice A/D offers to meet design and simulation challenges for these IoT-enabled devices.

Sensors

The purpose of these devices is to sense physical or electrical parameters, such as voltage, current, power, pressure, light, motion, position, proximity, occupancy, weight, and temperature, which can be processed further.

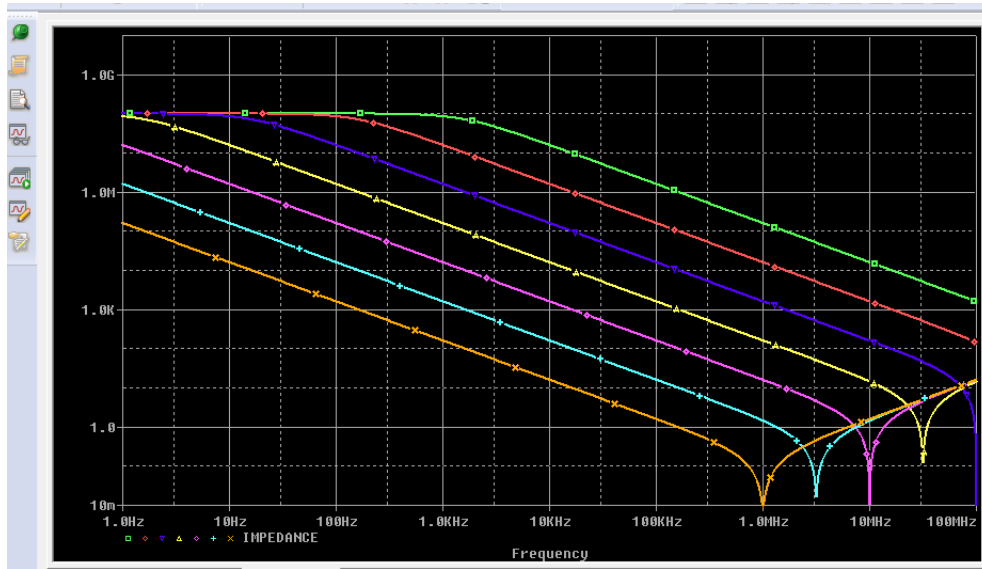


Figure 1: Simulating frequency response of a sensor input impedance

These sensors act as the eyes and ears of the vast interconnected system. Sensors, as their name suggests, are the sensitive elements in the design flow, and require careful design for reliable circuit implementation and functioning. Almost all of these sensors are the front-end analog type, and based on precision electronics. Some of the common design challenges for sensors are high-precision amplifiers, mixed-signal converters, and digital processing. Any unwanted parasitic elements in the signal chain can render the sensor data unreliable, or unusable. Therefore, maintaining the best signal-to-noise ratio is critical in IoT devices. PSpice A/D is the tool of choice for the simulation of analog and precision electronics circuits. With its accurate time-tested simulator and vast built-in library of sensors and precision electronics devices, it provides a jump start for front-end analog sensor design and simulation.

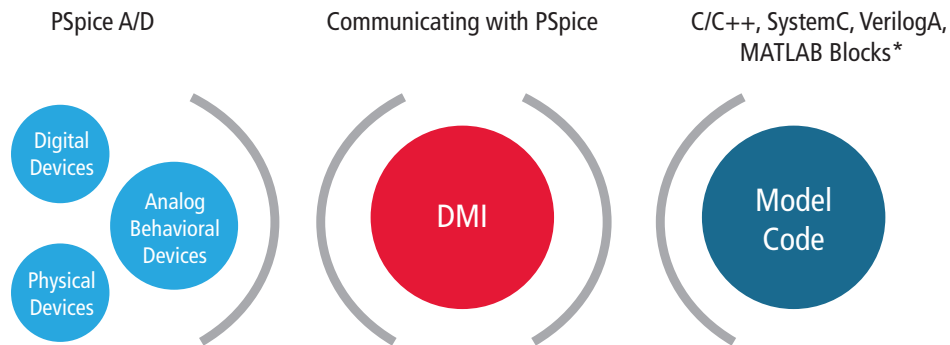


Figure 2: New PSpice Device Modeling Interface

PSpice A/D offers several different modeling tools and analysis techniques to accurately model and simulate these diverse types of sensors to meet design challenges. One can model a component using various levels of abstraction, as per the design need. These abstraction levels include architectural, functional, behavioral, gate level, circuit level, and physical implementation.

Controllers

In general, controllers fall into two categories, simple on-off controllers and advanced feedback-loop-type controllers with complex algorithms implemented in high-level languages, such as C/C++.

* Functionality available in PSpice AA or PSpice SLPS Option

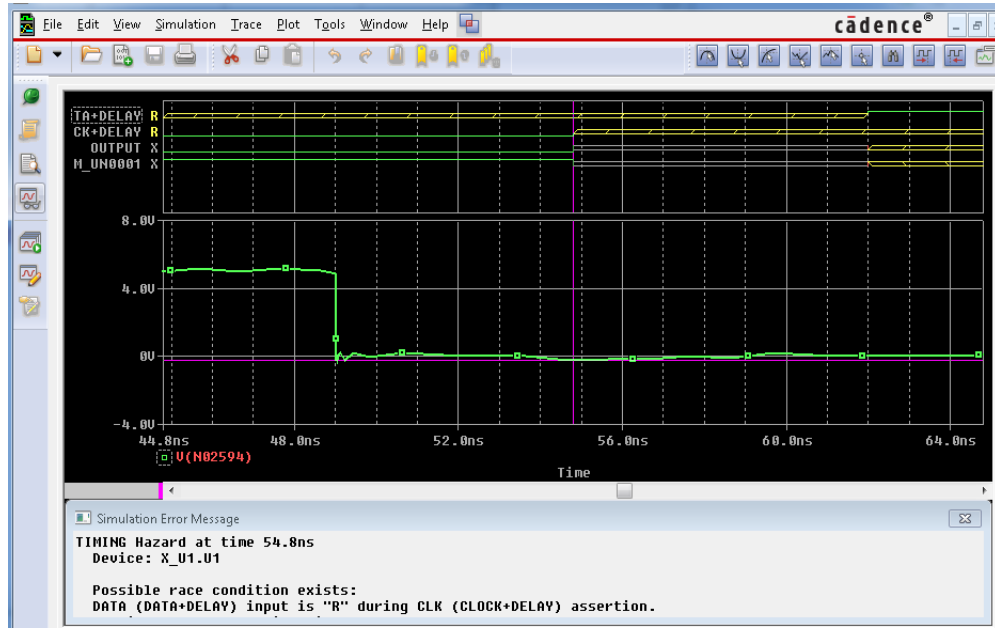


Figure 3: Digital worst-case simulation and best-in-class mixed-signal waveform analysis in PSpice A/D

Depending upon their application, IoT devices require one of these controllers. On-off controllers are simpler to implement but pose additional design challenges in the specific context of IoT devices. Some of these challenges are propagation delays and appropriate voltage levels at interfaces. These challenges arise due to the differing interconnects, digital logic families, and classes of devices. PSpice A/D offers a true mixed-signal design environment with an integrated analog and event-driven simulator with a fully automated analog-to-digital convertor (ADC) and digital-to-analog convertor (DAC) interface generator. This environment enables fast design intent capture, simpler modeling techniques, and accurate results. With its unique digital worst-case analysis and multilevel digital signal support, designers can easily simulate integration of controllers in true real-life operating conditions.

For modeling a controller with feedback loop, PSpice A/D offers comprehensive design modeling and simulation techniques. One can use conventional “spice” modeling techniques, such as precision electronics devices model and analog behavior modeling techniques, to model analog controllers. Designers can also use the new PSpice Device Model Interface* to model digital, SystemC®, or MATLAB-based systems to model digital controllers.

Actuators

One can argue that actuators are the least impacted module of this IoT design evolution. However, this may not be true, or be true only to a limited extent—miniaturization of devices and the need to conserve energy offer significant challenges for actuator designers, and the users of these actuators in systems. For example some of these devices may be expected to operate for very long durations on standard AA batteries.

* Functionality available in PSpice AA or PSpice SLPS Option

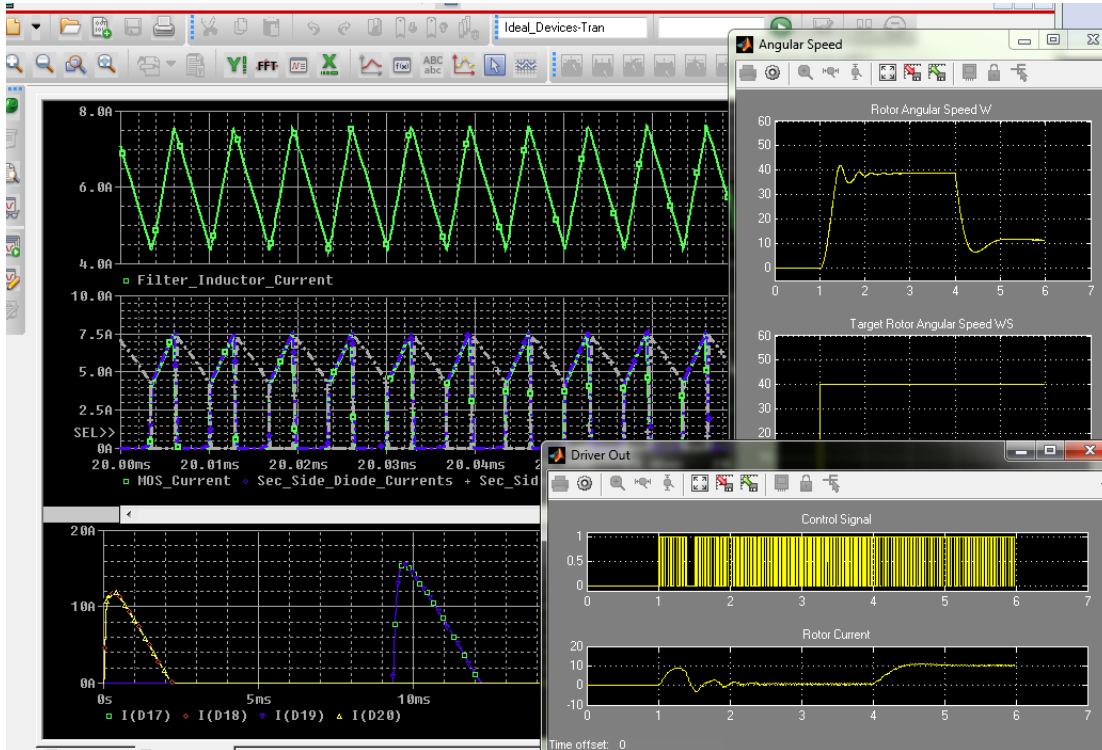


Figure 4: PSpice SLPS brings two best-in-class tools together

These two aspects have become the key drivers of actuator design and integration in the IoT space. PSpice SLPS offer a unique solution to meet these design challenges by bringing two of the best-in-class tools, the PSpice electrical circuit simulator and MATLAB mathematical modeling and electro-mechanical simulation tools. This collaboration enables designers to model and refine the electro-mechanical or electro-thermal module in MATLAB, then interface with actual electronics in PSpice A/D to get true real-life simulation, and optimize system interfaces, power consumption, and performance.

Summary

PSpice A/D technology along with PSpice AA and the PSpice SLPS Option offer a comprehensive feature set to meet the IoT design challenges. They offer comprehensive modeling capabilities and concurrent verification of the analog and digital portions of a design. Verifying top-level mixed-signal systems is a challenge for designers because of the time it takes to run simulations. The 64-bit multithreaded simulation engine enables designers to exploit today’s computing power to its fullest potential and perform complete system simulations. System simulation and modeling technology that enables unified design environment for mixed-signal design can help overcome these challenges, and consequently, help you deliver a high-quality product within your time-to-market window.