

Using Mixed Signal Analysis to address IoT design challenges

setting up a virtual prototype

Marcel Wezenberg, CB-Distribution



CB Distribution

Facts

- Cadence Channel Partner Netherlands, Belgium, Luxembourg, Spain and Portugal
- Located in Hengelo (Netherlands) and Madrid (Spain)
- Sales and Support of
 - Cadence IC design tools
 - Virtuoso, Incisive, Encounter
 - Dassault
 - Enovia DDM/PLM environment
 - Cadence PCB, Simulation and Packaging-Tools
 - OrCAD, PSpice, Allegro
 - WISE GerbTool
 - Nextra three-dimensional PCB design

IoT Design Challenges

Sensing a complex environment

position/motion material composition
proximity gas current/power
temperature chemical **environment** light humidity
occupancy biosensing pressure

Multiple connectivity options



Power is critical



Security is a must



The IoT is complex

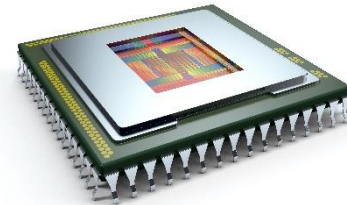
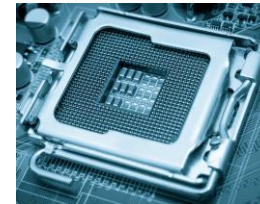
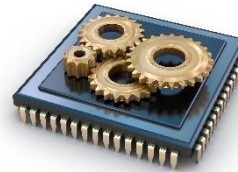
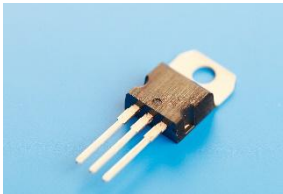


Connecting to the cloud



Source: Texas Instruments

System Integration



Electronic systems trending to large devices for lower power, higher reliability, and increased functionality in smaller package

HDL-level simulation in PCB systems, with multiple large ICs, is prohibitively slow

SPICE Models

Mixed-Signal Models

System Models

Higher abstraction and lower accuracy and lower simulation time →

SMPS Designs

S/W algorithm Controlled Power Supply

Digital controllers enabled
Power Supplies



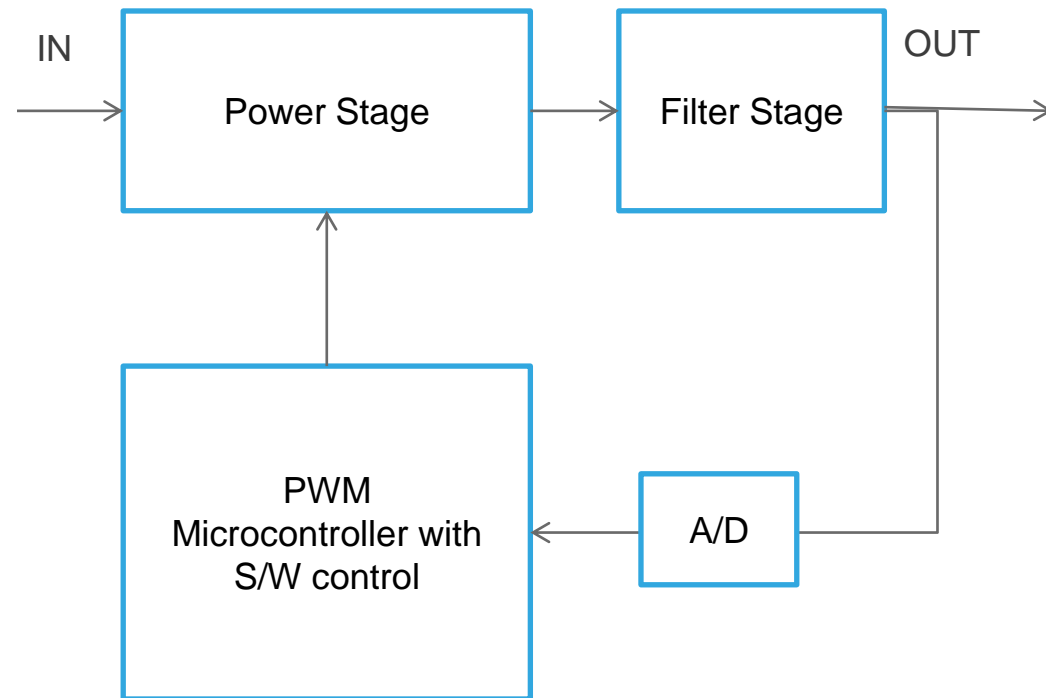
Advanced control algorithm
(non-linear control,
improved transient)



Enable easy management
of multiple control loops

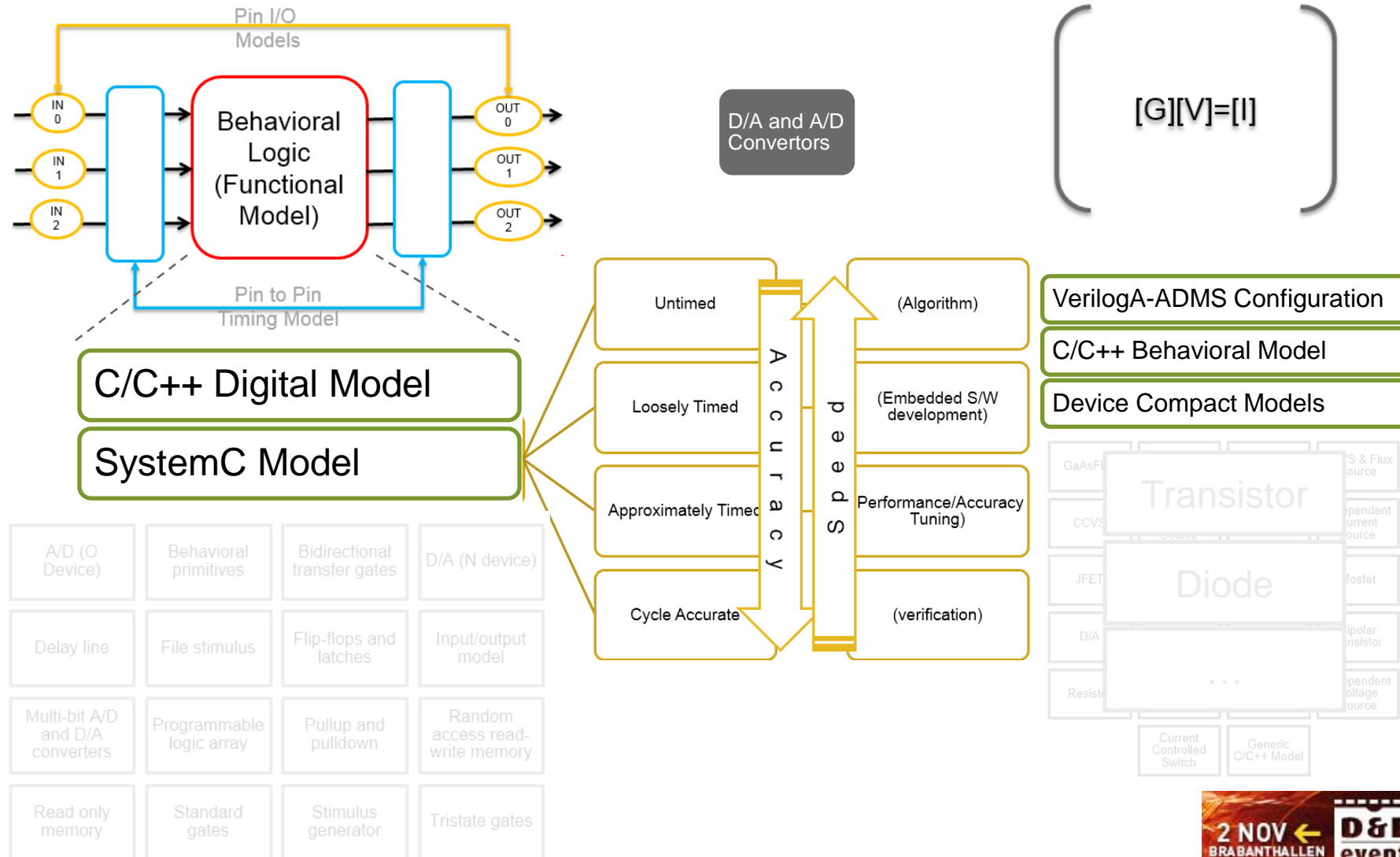


Better precision tolerance
to aging, temperature
effects, etc.



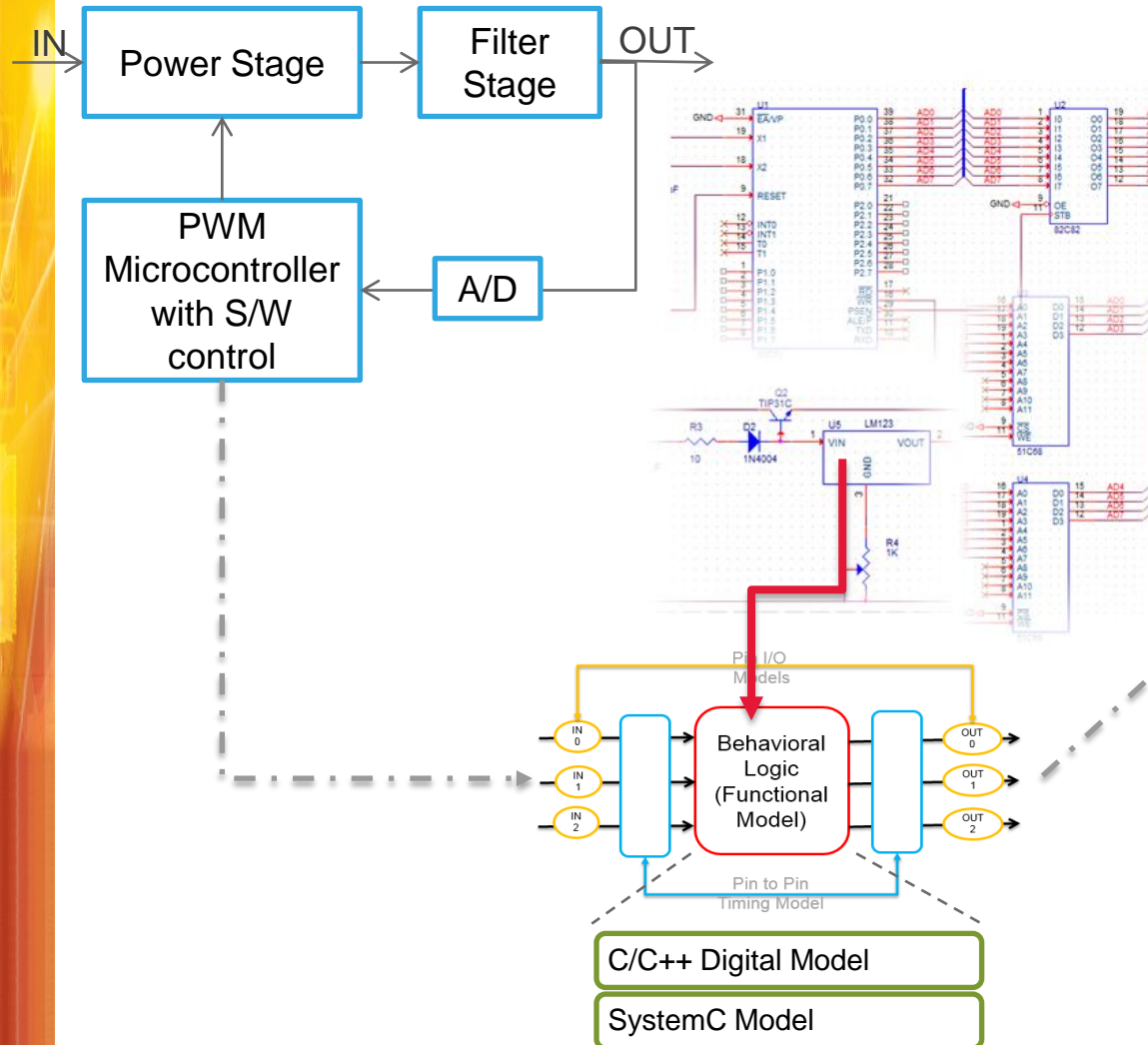
PSpice mixed-signal macro model

C/C++/SystemC extensions



Example

Software algorithm-controlled PWM in power supply



Develop and test MCU targeted algorithms in PSpice models

```
for (int i = 0; i < 8; i++) {  
    FB[i] = pVectorSti  
}  
for (int i = 0; i < 8;  
    REF[i] = pVectorStates[16 - i].getLevel();  
}  
oldPW=PW = pVectorStates[17].getLevel();
```

Get Interface values

```
pspBits2Int(FB, FBInt, 8);  
pspBits2Int(REF, REFInt, 8);
```

```
if (mCurrentCLKCount <= 0) {  
    mCurrentCLKCount = mPER;  
}  
if (mCurrentCLKCount > mDuty  
    mPWStatus = false;  
else  
    mPWStatus = true;
```

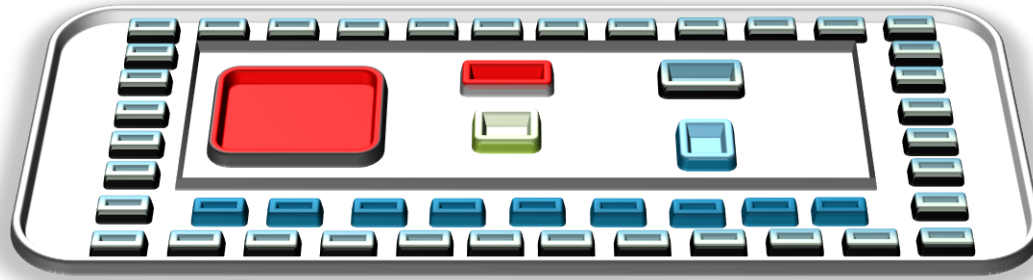
PWM S/W Control Code

```
if (mPWStatus==true && (int)PW != 1){  
    PW = pspBit::HI;  
}  
else if (mPWStatus == false && (int)PW != 0){  
    PW = pspBit::LO;  
}
```

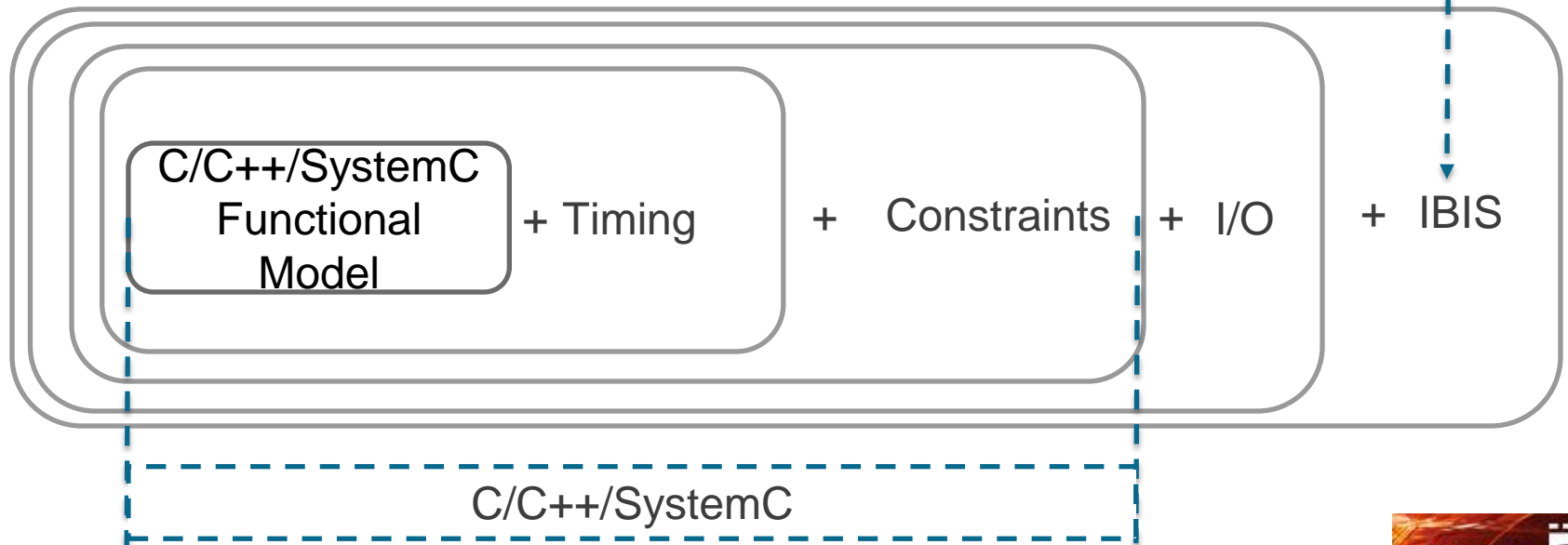
Set Interface values

```
if ((char)oldPW!=(char)PW) {  
    PSpiceState lState = (pVectorStates)[0];  
    lState = PW;  
    fp_SetState(mRef, 0, &lState, NULL);  
}
```

Digital Block Model Implementation

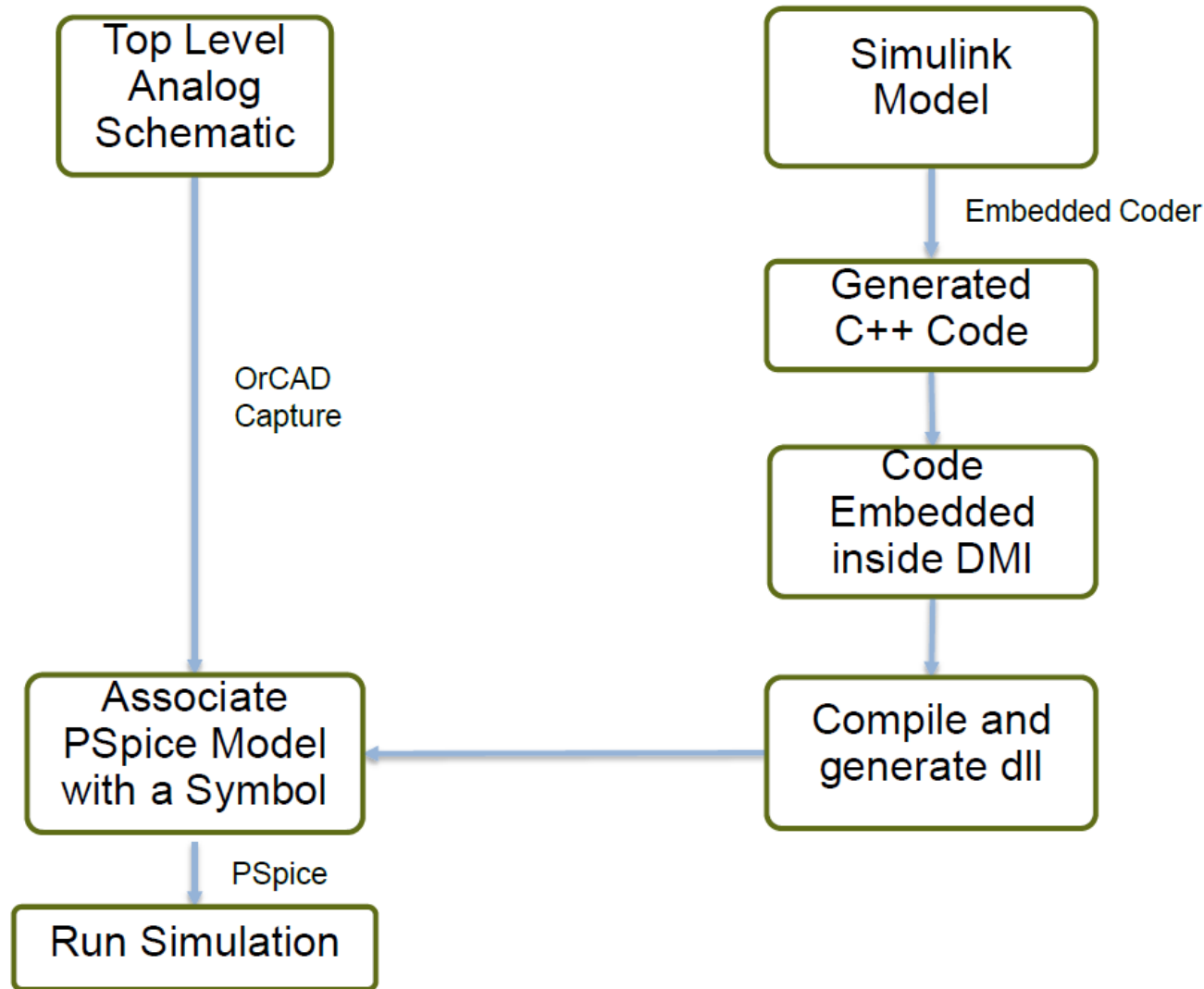


IBIS2Spice



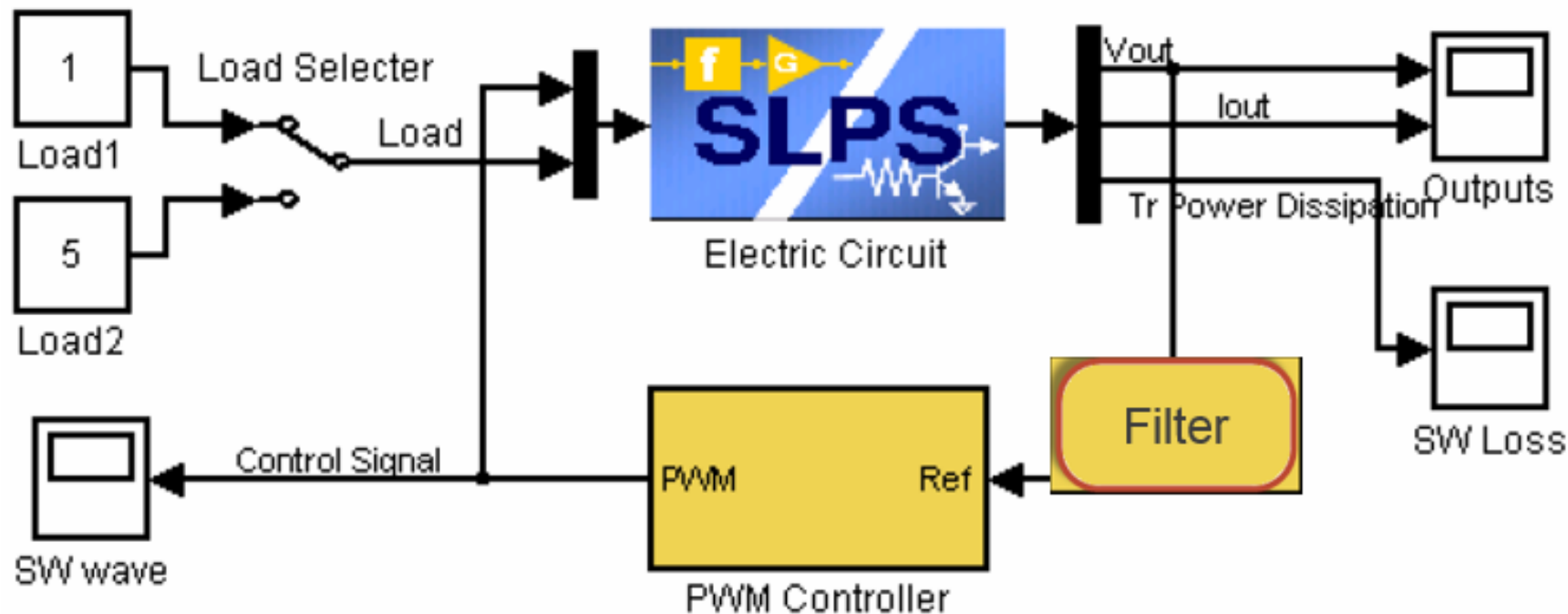
Matworks interface Top-Down

Embed Simulink/MATLAB Generated Code in PSpice



Matworks interface Bottom-Up

Co-simulation Pspice and Simulink/MATLAB



PSpice Model Code Generator

PSpice DMI Template Generator

You can use this UI to auto-generate template code for PSpice-DMI models. Analog, Digital and SystemC model templates are supported. The UI also supports import of Verilog-A Compact Device models using ADMS.

Recommended steps:

1. Test the model code stand-alone by building an exe.
2. Create the PSpice-DMI adapter code, and edit it in Visual Studio to insert model code.
3. Use the generated PSpice library (.lib file) to create a schematic symbol.

The generated symbol can be placed in the schematic for PSpice simulation.

Part Details

Part Name:

Part Type:

Ports

Interface Type:

Port Entry: ☐ Manual ☐ CSV File

Parameters

Global Parameters:

Instance Parameters:

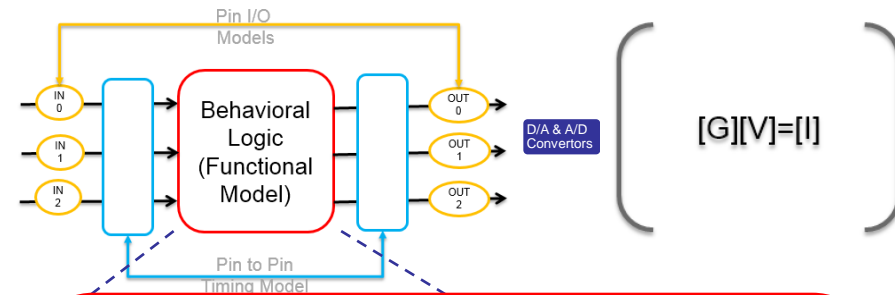
Output

DLL File Name:

Log File Name:

DLL Location:

Hover over what you want to know about

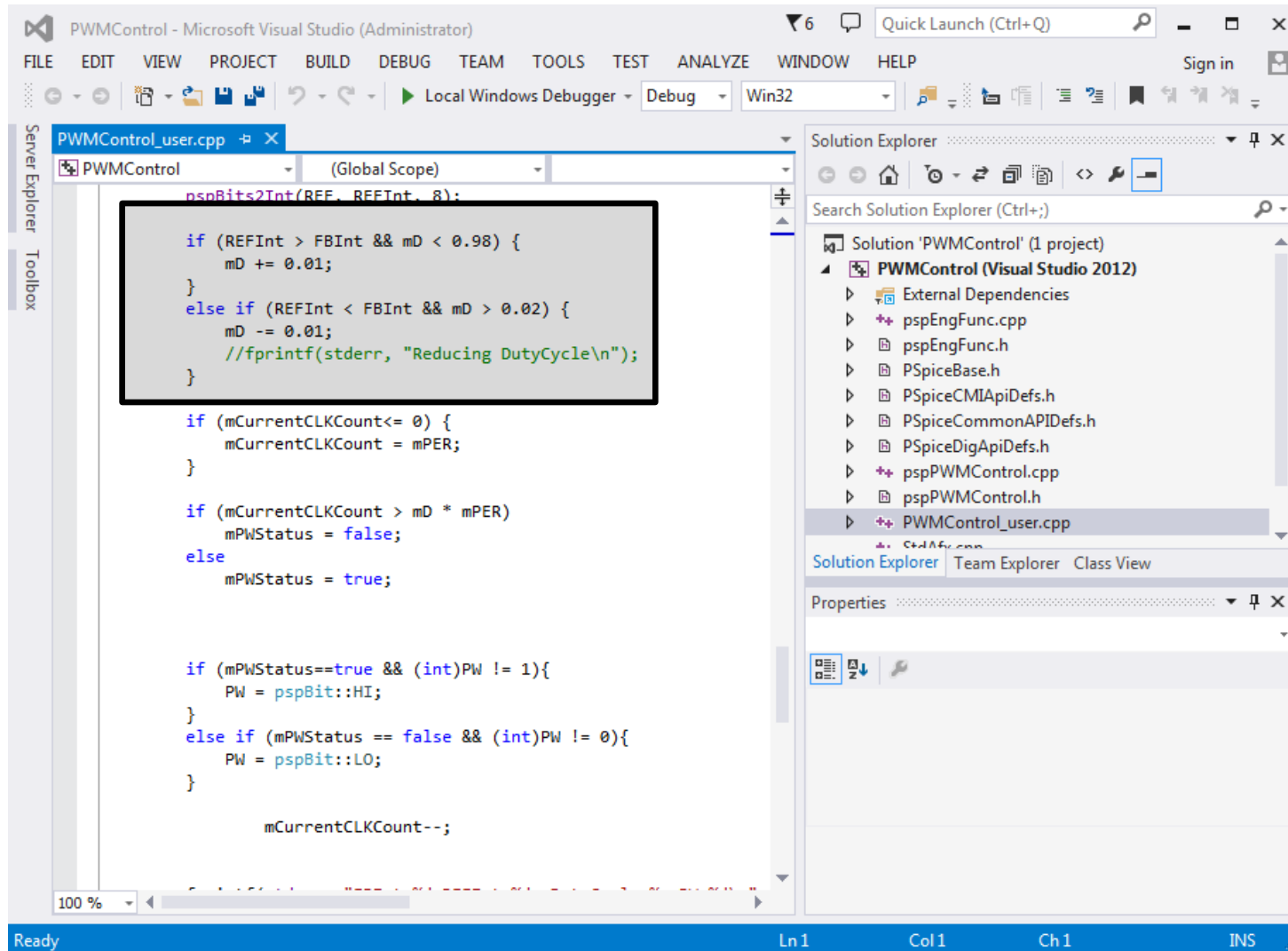


Model Plug-in Code

User Code
C/C++/SystemC

Microsoft Visual Studio

Complete, Compile, Link and debug the user code

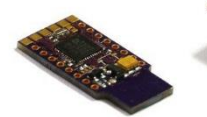


IoT

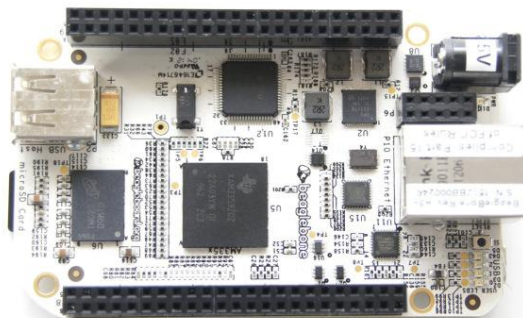
Hardware Platforms



ESP8266

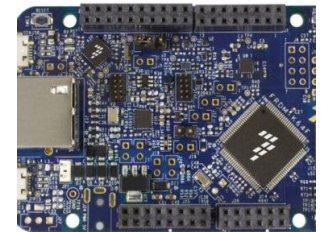


Espruino Pico



Name	Arduino Uno	Raspberry Pi	BeagleBone
Model Tested	R3	Model B	Rev A5
Price	\$29.95	\$35	\$89
Size	2.95"x2.10"	3.37"x2.125"	3.4"x2.1"
Processor	ATMega 328	ARM11	ARM Cortex-A8
Clock Speed	16MHz	700MHz	700MHz
RAM	2KB	256MB	256MB
Flash	32KB	(SD Card)	4GB(microSD)
EEPROM	1KB		
Input Voltage	7-12v	5v	5v
Min Power	42mA (.3W)	700mA (3.5W)	170mA (.85W)
Digital GPIO	14	8	66
Analog Input	6 10-bit	N/A	7 12-bit
PWM	6		8
TWI/I2C	2	1	2
SPI	1	1	1
UART	1	1	5
Dev IDE	Arduino Tool	IDLE, Scratch, Squeak/Linux	Python, Scratch, Squeak, Cloud9/Linux
Ethernet	N/A	10/100	10/100
USB Master	N/A	2 USB 2.0	1 USB 2.0
Video Out	N/A	HDMI, Composite	N/A
Audio Output	N/A	HDMI, Analog	Analog

Comparing the three platforms.

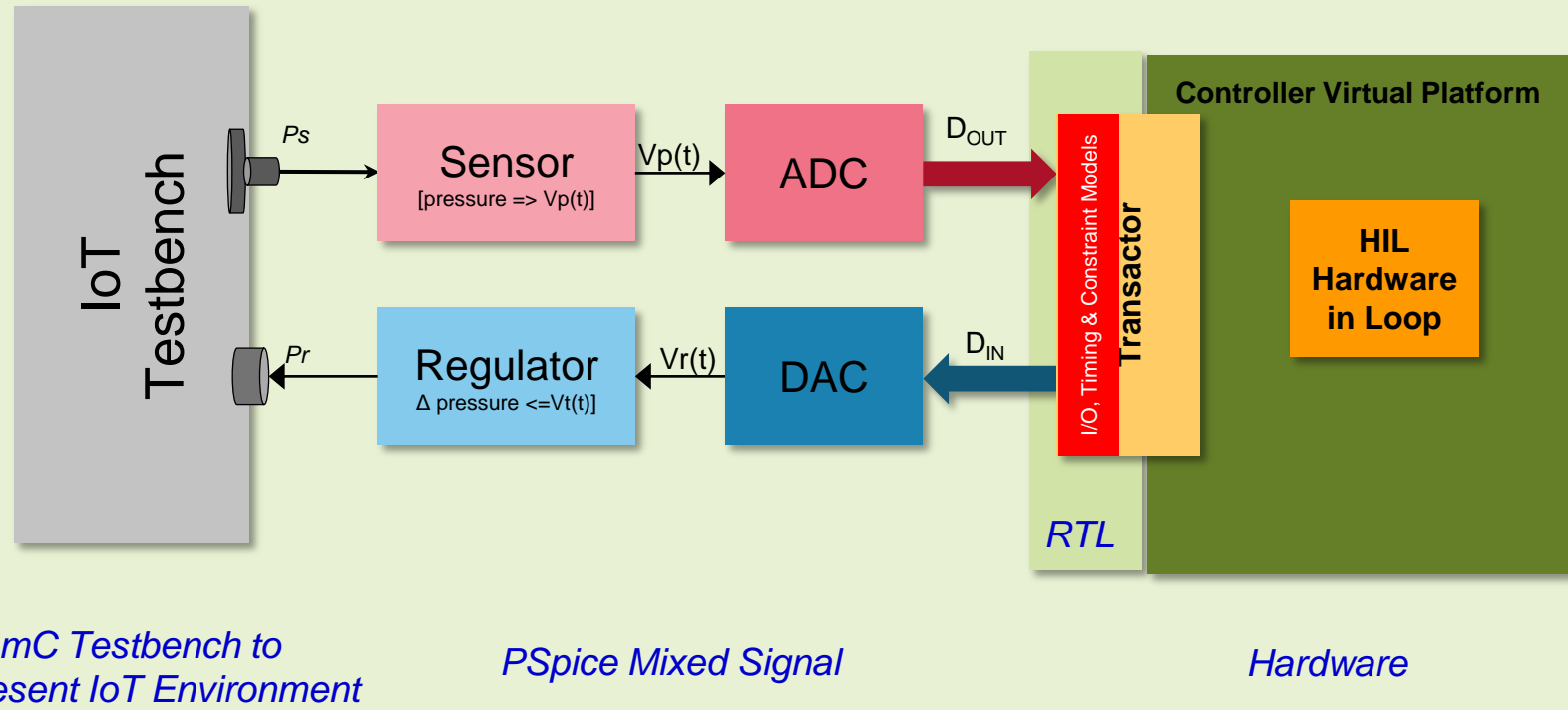


FRDM-K64F



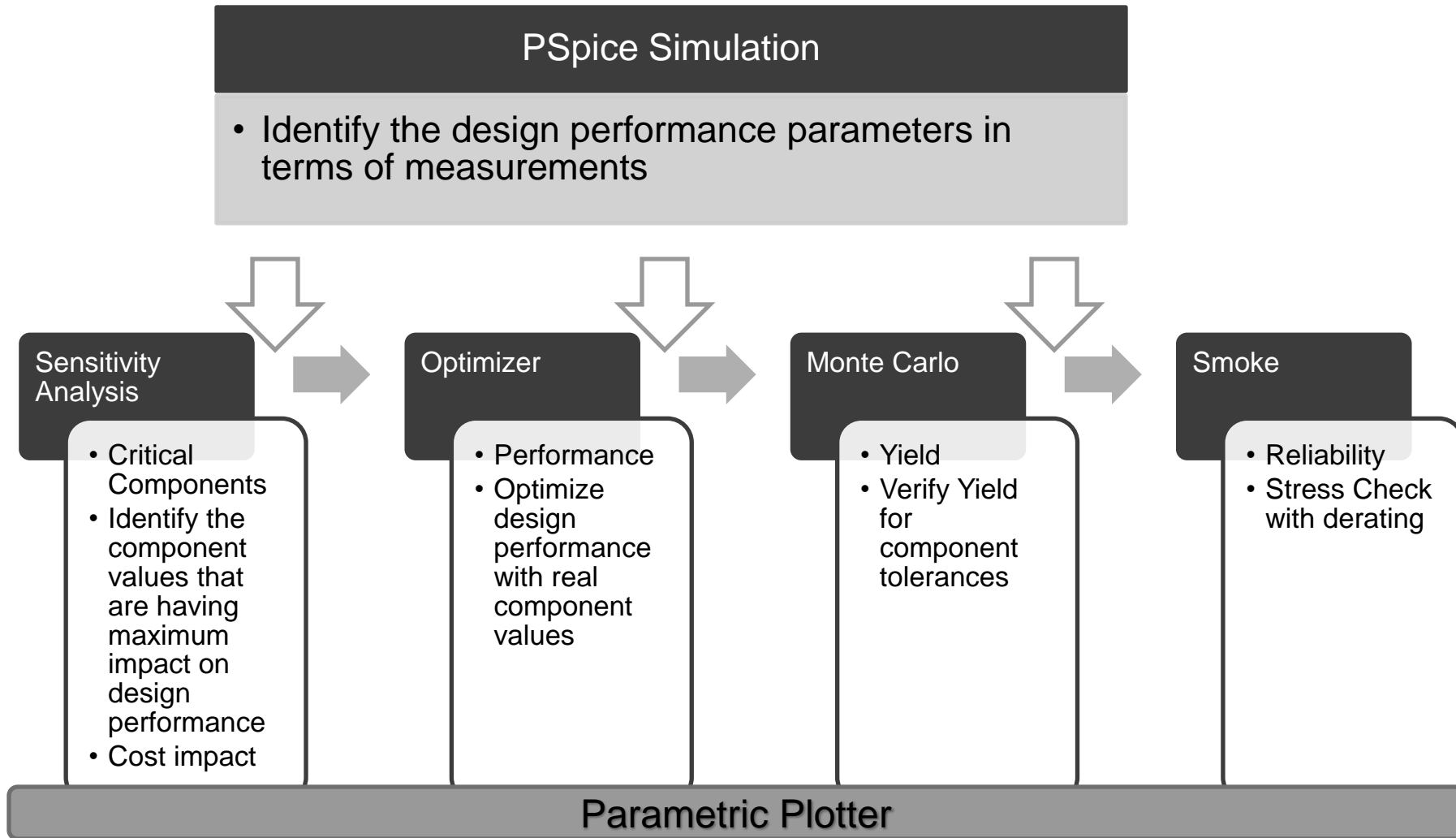
CC3200

IoT Design with PSpice

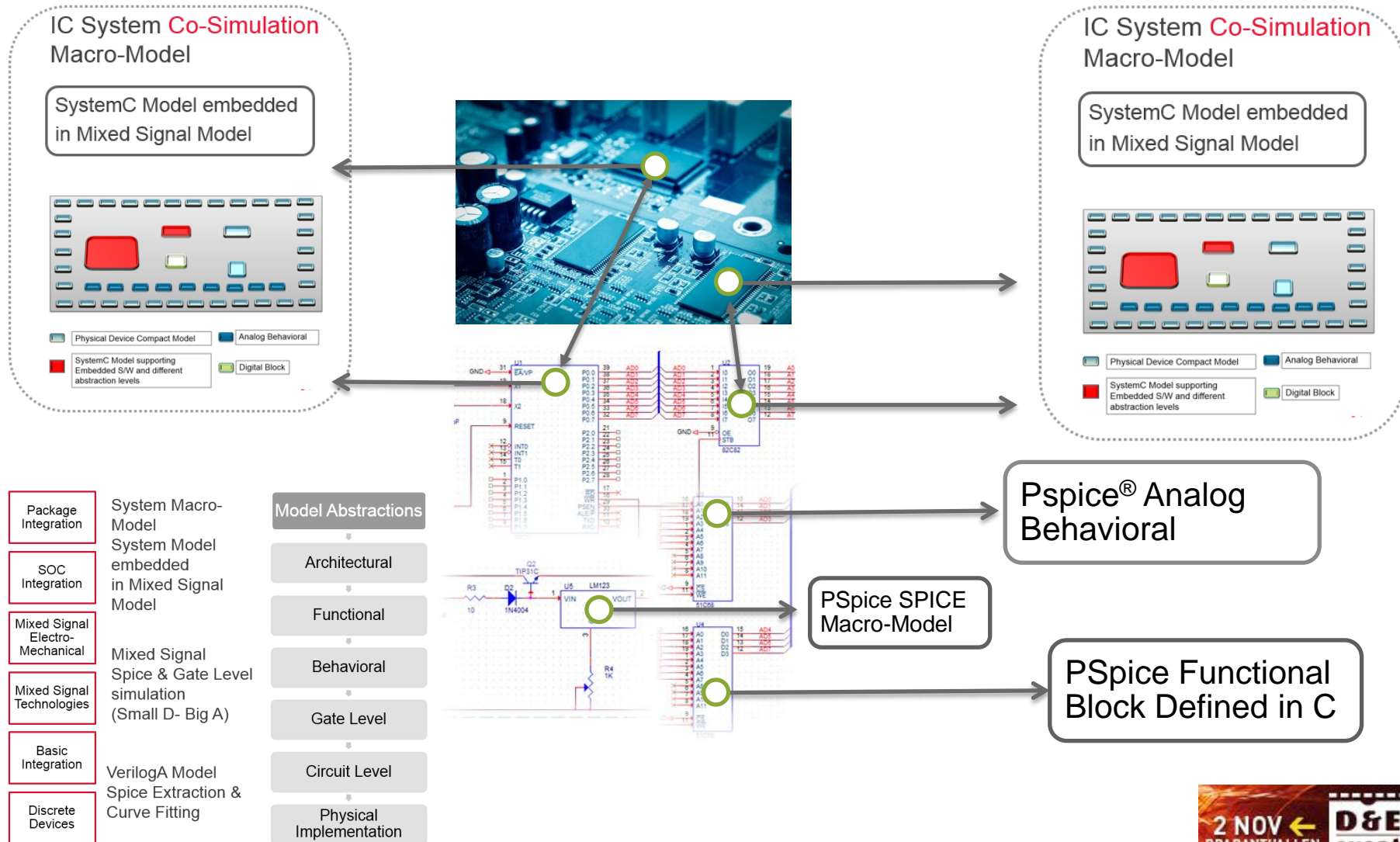


Minimal Modeling Effort

Reliability



PSpice virtual prototyping PCB systems



PSpice.com

New website for PSpice

- One-stop-shop for all PSpice resources and information
- Model library
- Forum

The screenshot shows the PSpice Community website. At the top, there is a navigation bar with the PSpice Community logo, links for Model Library, Resources, Applications, and User Forums, a 'My account' link, a 'Log out' button, and a search icon. The main content area is split into two sections. On the left, a red background features the 'PSpice User Forum' heading, a paragraph about the forum's purpose, a list of benefits (access to 33,000 models, premium resources, vibrant community, learning skills, and free trial), and a 'User Forums' button with a right arrow. On the right, a dark background with a car image features the 'LATEST NEWS' section, a headline 'Tata Motors Standardizes on PSpice for Automotive Design Simulations', and a brief description of a case study.

PSpice Community

Model Library Resources Applications User Forums

My account Log out

PSpice User Forum

Have questions about Cadence® PSpice® technologies? Ask the PSpice User Forum! The PSpice user community is the destination for you to find PSpice resources, ask and answer questions, and interact with your industry peers and PSpice experts!

- Get access to over 33,000 PSpice models
- Premium resources for subscribed users
- Engage with a vibrant community
- Learn new skills for using PSpice
- Try out PSpice for free

User Forums →

LATEST NEWS

Tata Motors Standardizes on PSpice for Automotive Design Simulations

Read this customer case study to find out how OrCAD® PSpice Designer helped Tata Motors find design problems much earlier, saving crucial time and money spent in building and debugging ECU boards within system prototypes.

Forum Activity

Test

OrCAD Products admin, 3 weeks ago

Latest Resources



OrCAD PSpice Designer Plus

[View](#)

Upgrading to OrCAD® PSpice® Designer Plus provides the PSpice...

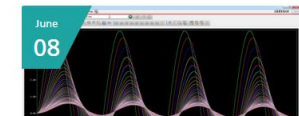


OrCAD PSpice Designer

[View](#)

OrCAD® PSpice® and OrCAD Capture combine to

Upcoming Event



Announcement

Cadence and Mathworks Collaboration

- Provide system-level simulation solutions
- For customers using MATLAB/Simulink and PSpice for mixed signal, IoT and Automotive applications
- Thursday, Nov 3rd at MATLAB Expo in San Jose.



Verify Your Design
From Concept to
Product Realization

Cadence and MathWorks partner to provide
seamless bi-directional integration
for system-level simulation

[Learn More](#)

PSpice® < > Simulink

The banner features a dark blue background with a complex circuit board pattern. On the right side, there are several interlocking gears, some of which are white and others are blue. The text is white and red, with the PSpice and Simulink logos prominently displayed in the center.

Q&A

- For more information visit the CB-Distribution booth, we're at booth nr. 8

