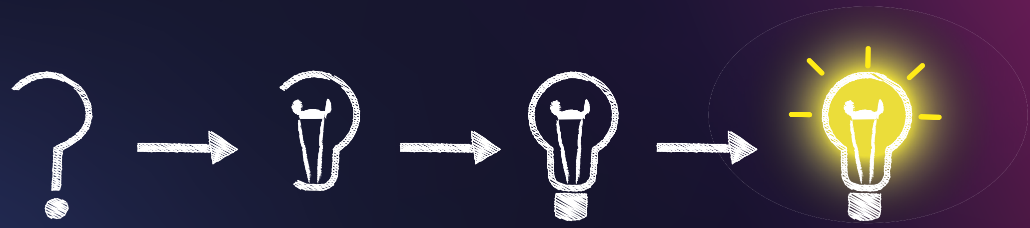
POWER YOUR IDEAS!



April 17th, 2024
9:00 - 17:00 CEST
(Central European Summer Time)

13th Power Analysis & Design Symposium

Advanced Characterization, Simulation and Troubleshooting of Electronic Power Systems

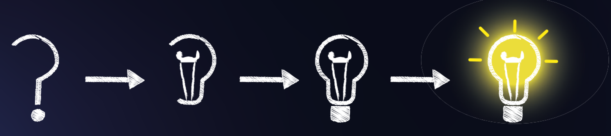
Live Virtual Event

With lectures, practical examples and demonstrations presented by international experts from:

Biricha Digital, Future Electronics,
K&K Prime, Microchip Technology,
University of Zaragoza and OMICRON Lab

More information & free registration at:
www.omicron-lab.com/event

OMICRON LAB



Topics

A Power Supply Filter with On-board Voltage Socket for 12V 10A

by Günther Klenner - K&K Prime

Our customer with a portable, battery-powered device sold his system for an in-car use. He said: "We just need a cable with an on-board 12V voltage socket," and the project started. Join me on our way through the whole chain of developing such a filter. The project includes setting up the specifications, selecting the components, performing SPICE simulations, considering the Middlebrook stability criteria, placing the components on the PCB, designing a housing using 3D printing, and verifying the final results using the Bode 100.

Power Supply Compensator Design Using Operational Transconductance Amplifiers

by Dr. Ali Shirsavar - Biricha Digital

The design of compensators with standard voltage error amplifiers, i.e., a typical operational amplifier (op-amp), is well documented in many books, application notes, design examples, and even automated software. What received less attention is compensator design using operational transconductance amplifiers (OTAs). Unfortunately, our PWM controller IC often contains only an internal transconductance op-amp, leaving us no choice. In this session, Dr. Ali Shirsavar from Biricha Digital will demystify how this particular op-amp operates and explain in simple terms how we design PSU compensators for stable operation down to the component level. As always, simulations and experimental results will be presented.

All attendees will receive a six-month free license for Biricha WDS automated power supply design software, which includes OTA compensator design.

Designing Digital Control Loops and Firmware

by Andreas Reiter - Microchip Technology

Over the past 15 years, digital control of power supplies has conquered specific markets and applications. Although this approach provides a wide range of new capabilities required to enable new technologies, such as driving new, complex converter topologies or solving system-level challenges in hyper-adaptive power distribution networks, it also introduces a significant number of design challenges on its own.

In this session we will cover common practices and tools used in the design process of software-based, digital control systems. In short live demos we will learn how to derive plant transfer functions using simulation and bench measurements, how to close the feedback loop in software, and how to use extended simulation to test and validate the complete control software using Model-Oriented Design tools.

An Introduction to Power Supply Simulations with SIMPLIS

by Christophe Basso - Future Electronics

A simulation engine is a convenient and powerful tool when designing switching power supplies. If many simulation engines are available nowadays, few can deliver all the needed information to design the entire circuit: SPICE is a valuable assistant but it is prone to convergence issues and has difficulties with long-run PFC simulations. Besides, you need to resort to an equivalent small-signal model to explore a compensation strategy to close the feedback loop safely. If this is easy for the basic switching cells, the exercise complicates when no average model exists, like in the LLC converter case, for instance. SIMPLIS, on the other hand, builds on a different simulation engine and lets you simulate at a quicker speed than SPICE. Furthermore, it can extract the small-signal response of any switching converter. This seminar offers an introduction to how SIMPLIS operates compared to SPICE and how it can improve your design cycle in control loop design.

This seminar targets switching power supply designers and requires an intermediate background in simulation.

Using S-Parameters for the Design of EMC Filters

by Arturo Mediano - University of Zaragoza

This is a talk to explain to non-RF engineers what s-parameters are, how you can measure them using a vector network analyzer like the Bode 100 or Bode 500, how to apply them in EMC filter characterization, and how to export data for simulation (i.e. LTSPICE) to evaluate the effect of different terminal impedances.

PDN Impedance Measurements using the Bode 500 and Picotest PDN Probes

by Florian Hämmerle - OMICRON Lab

The power delivery network (PDN) describes the complete system delivering the power from the power source (i.e. a battery or a AC/DC power supply) to the power sink (i.e. a processor or FPGA). The PDN must be able to provide a sufficiently low source impedance such that the voltage drop does not exceed the limits when the maximum current is flowing. The faster the current transients, the more challenging this gets. Adding decoupling capacitors to the PDN helps to provide a low impedance over a wide frequency range but also brings the risk of resonance peaks in the PDN. A simple PDN impedance measurement can help in verifying the PDN design. It will reveal critical resonances that could lead to system failures. This presentation will cover the basics of dynamic output impedance and PDN impedance, focusing on measuring low impedance using the Bode 500 in conjunction with Picotest PDN probes.