



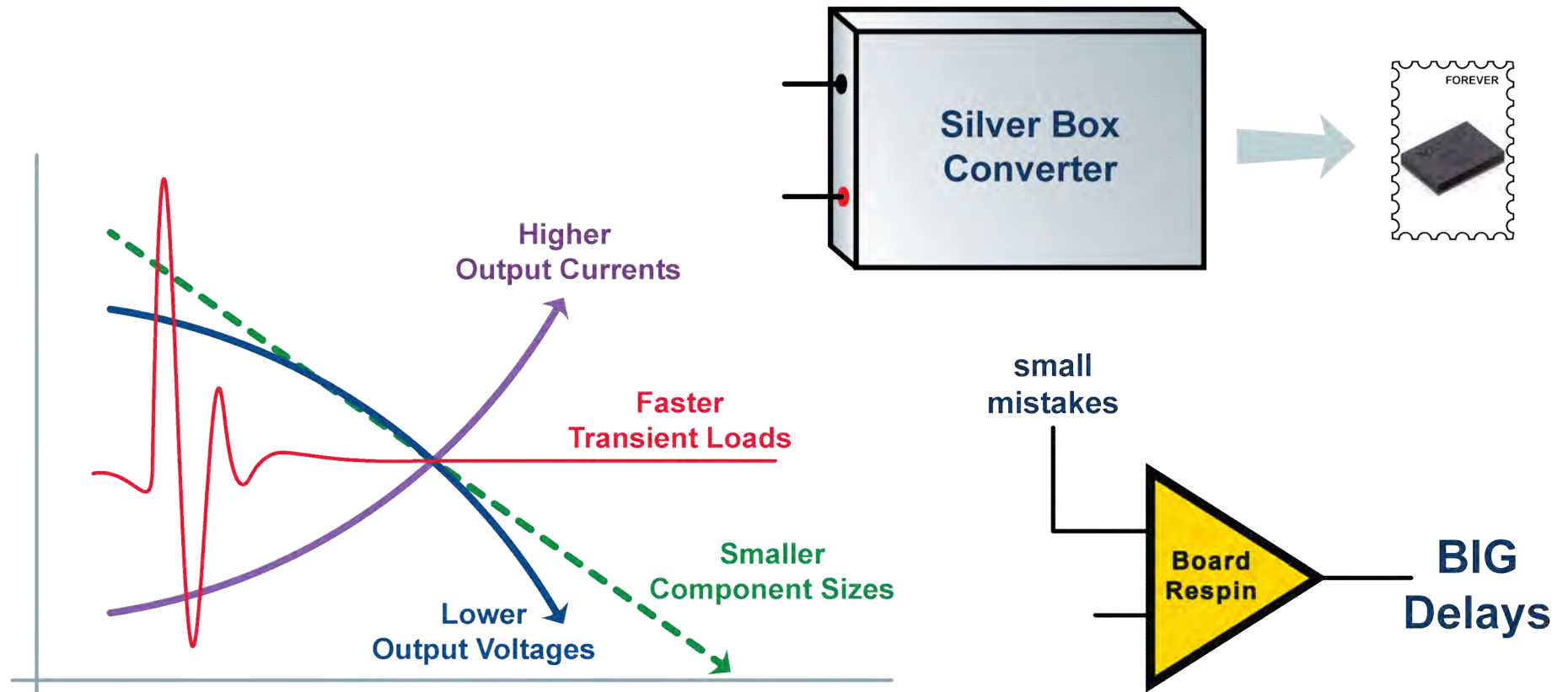
Circuit Simulation Primer For The Power Electronics Engineer

C. R. Swartz; Senior Principal Engineer/Manager
Vicor Advanced Systems Design Engineering
IEEE Power Seminar, Long Island, NY 3-NOV-2022

Seminar Roadmap

- Power Electronics Today
- Circuit Simulation Requirements
- Circuit Simulators Suitable For Power Electronics
- Narrowing The Focus
- Understanding The Simulator: SPICE & PWL
- PWL & SPICE: Choosing The Best Tool
- Strategy For Efficient Power Simulation

Power Electronics Today



My First Complex Simulation: A Cautionary Tale

Task

- Model Isolated AC-DC converter with PFC to diagnose a customer problem with hot-swap current sharing.

Approach

- Impressively detailed models using SIMPLIS

First real-world test

- The learning begins! Simulation run time was too long to be practical in diagnosing the system problem!



Circuit Simulation Requirements

Power Engineers have many needs.....

Simulation Requirements

- Efficiency
- Line and load regulation
- Converter input and output impedance, closed and open loop
- Transient response
- Stability analysis, gain and phase margin
with and without multiple loops (also digital control loops)
- Audio susceptibility
- Capacitor input and output ripple voltage
- MOSFET SOA analysis to include voltage and current stress (ringing, etc.)
- Capacitor input and output RMS ripple current
- MOSFET switching losses and conduction losses

More Simulation Requirements

- Conducted EMI
- Line surge and immunity
- Input filter interaction, damping and attenuation to meet EMI specifications
- Current sharing (multiple converters)
- Synchronization (multiple converters)
- Interleaving (multiple converters)
- Input impedance and filter interaction
with multiple converters either independent or in parallel
- Thermal analysis
- PDN (Power Delivery Network) analysis
- Protection circuit performance (current limit, OVP, etc.)



Circuit Simulators Suitable For Power Electronics

Circuit Simulators Suitable For Power Electronics

- **SPICE-based tools** – PSpice®, SIMetrix, LTspice®, TINA (DesignSoft), ICAP/4 (Intusoft), NgSPICE
- **SIMPLIS** – **SIM**ulator for **P**iecewise **L**inear **S**ystems
(www.simplistechnologies.com)
- **PSIM**® – PWL simulation engine with SPICE add-on capability
(www.powersimtech.com) now part of Altair
- **NL5** – PWL implementation but ideal components can be simulated
(www.sidelinesoft.com)
- **RidleyWorks**® – a powerful Excel calculation tool with LTspice® and **PSIM**® export
(www.ridleyengineering.com)
- **MATLAB**®/**Simulink**® – you need to do the math but the power is extreme. Many toolboxes...
(www.mathworks.com)

Narrowing The Focus

Simulation is a wide-ranging topic.....

Simulation Strategies In SPICE & PWL

- All of the simulators are equally good for what they are intended to do.
- This seminar is about simulation strategies, not which simulator is best.
- Most simulators use either SPICE- or PWL-based engines.
- SPICE-based simulators have been the standard for many years.
- Most customers ask for either SPICE-based or PWL-based simulations.
- Most PWL simulators were created specifically for power electronics.

Experience the simulators for yourself

Comments about PWL- and SPICE-based simulators as a group may not apply to all.

Circuit Simulation Is Only One Design Tool

- Simulation is NOT a substitute for testing the actual hardware.
- Theory and practice must match: if they don't, one of them is WRONG!
- Simulations can lie!
- Models should be verified with data sheets.
- Most OPAMP models do not model actual silicon for bias currents, CMR, offsets and capacitive loading.
- Many MOSFET and diode models' reverse recovery characteristics need checking!
- Stability criteria for references are often not modeled accurately.
- Thousands of models are available for download, many not completely checked!

Simulation is a tool, along with design calculations, analysis and testing.

Proper simulation can predict problems while in the design stage and help understand how to fix them.



Understanding The Simulator

SPICE-based tools and models

SPICE-Based Tools

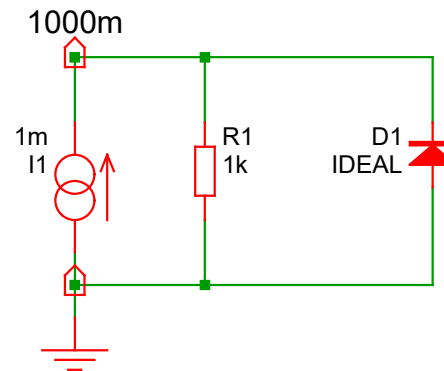
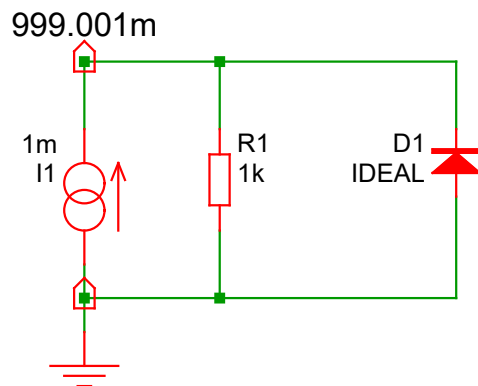
Simulation Program with Integrated Circuit Emphasis

- **Industry standard** in circuit simulation (in various modified commercial versions)
- **Invented to model actual silicon** integrated circuit implementation of bipolar and MOS devices
- **Equations describe the actual physics** behind semiconductors and are very accurate and detailed
- **Optimized for circuits that can be linearized** around a DC operating point
- **Detailed and complex models** for semiconductors
- **Simulates virtually any circuit** within simulator-controlled accuracy (with some exceptions)
- **Robust and time-tested** simulator algorithms
- **Many options and tolerance settings required** to allow for convergence to succeed
- **Analysis mode settings at minimum:** operating point (.OP), DC (.DC), Transient (.TRAN), AC (.AC)
- **Supports temperature analysis**

What You See May Not Be What You Get

A hidden dimension in SPICE

- Activity behind the scenes aids simulation convergence.
- Simulated netlist (schematic) may include invisible extras.



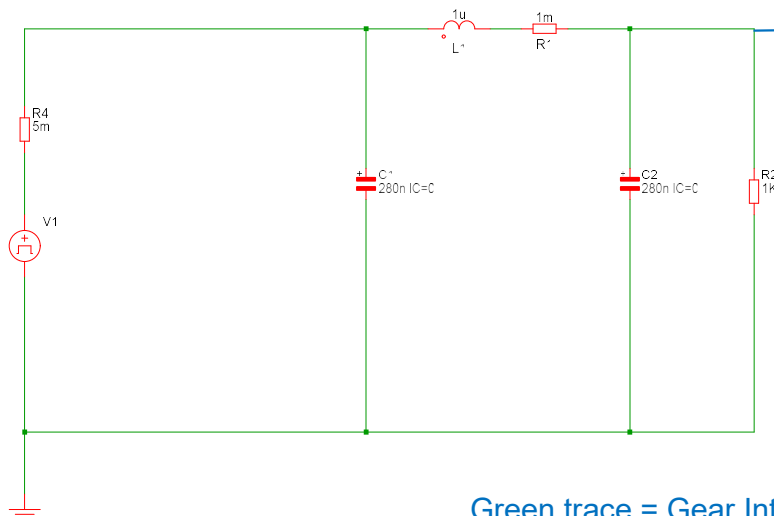
GMIN setting places conductance across semiconductor junctions.

GMIN was set too low!
Default setting fixed issue.

What You See May Not Be What You Get

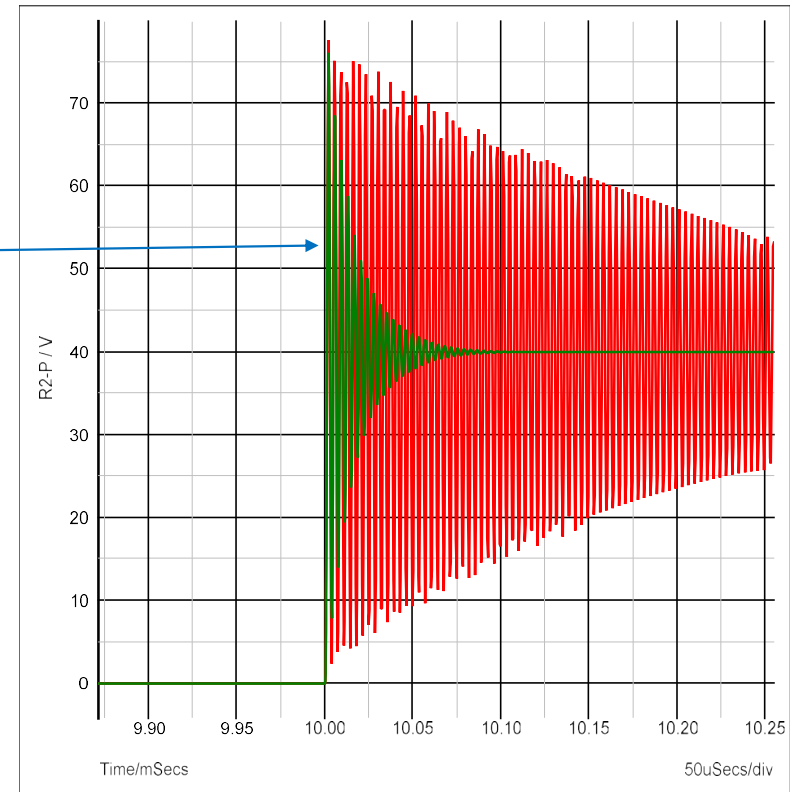
Another surprising result

- Consider an un-damped resonant circuit (filter)



Green trace = Gear Integration
Red trace = Trapezoidal Integration

Gear Integration adds damping to the circuit via the integration method used



Analysis Of Small-Signal Response For Switching Power Supplies Presents Challenges

AC analysis is *possible* in SPICE with a switching converter topology...

- It requires multiple transient simulations at steady state while injecting perturbations
- Steady state determined manually; circuit may be unstable or not really in steady state
- Injected signal must be correct over many frequencies
- Each transient simulation must run long enough to resolve the lowest frequency to be measured
- If you want to plot a 100 point Bode Plot, 100 transient simulations must be run.
- Each run must be saved to a file and then you need to extract the gain and phase data from it.

...but may not be *advisable*.

- Parasitics make the model tricky and time consuming. Getting the injection signal right is difficult.
- Imagine sweeping a PFC front end or doing Monte Carlo analysis on loop components!
- Computers are faster, but this method can still be impractical for bigger circuits.

Averaged Models Offer Powerful Solutions

The averaged model is a powerful tool for small-signal analysis on switching power supplies

- Equations describe average voltages and currents across and through terminals over the switching period.
- Result: a large-signal model for a closed-loop system.
- SPICE then performs the linearization during AC analysis for free.

Serious advantages

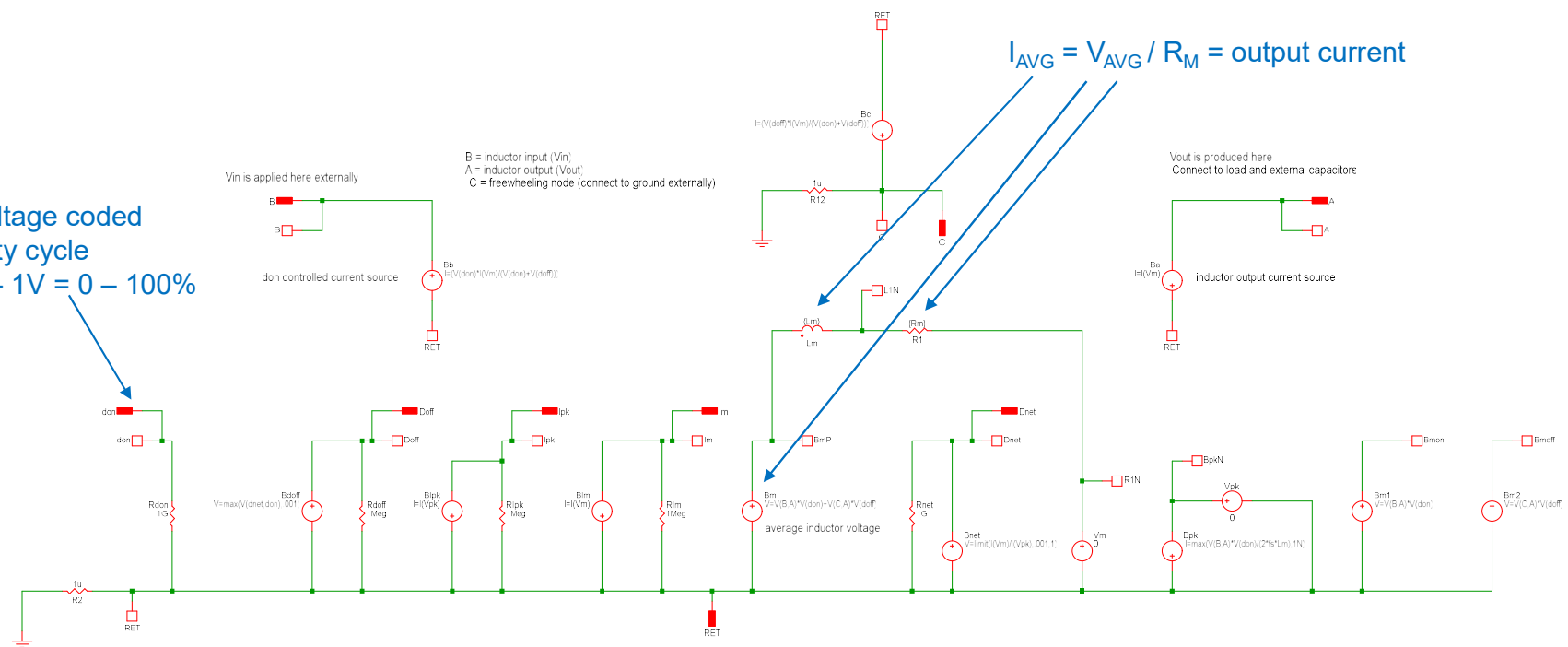
- One model for both transient (load step) and AC analysis.
- Connect SPICE models of the rest of the circuit to it — they will work like the real circuit. (OPAMPS etc.)
- AC analysis and model validity is good up to $\frac{1}{2}$ the switching period.
- Fastest model with very low parts count.
- Can be used in any simulator that supports SPICE syntax

Caution: be careful to prevent model discontinuities; convergence issues may occur or bias point may fail

Averaged Model: VM Control Buck Regulator

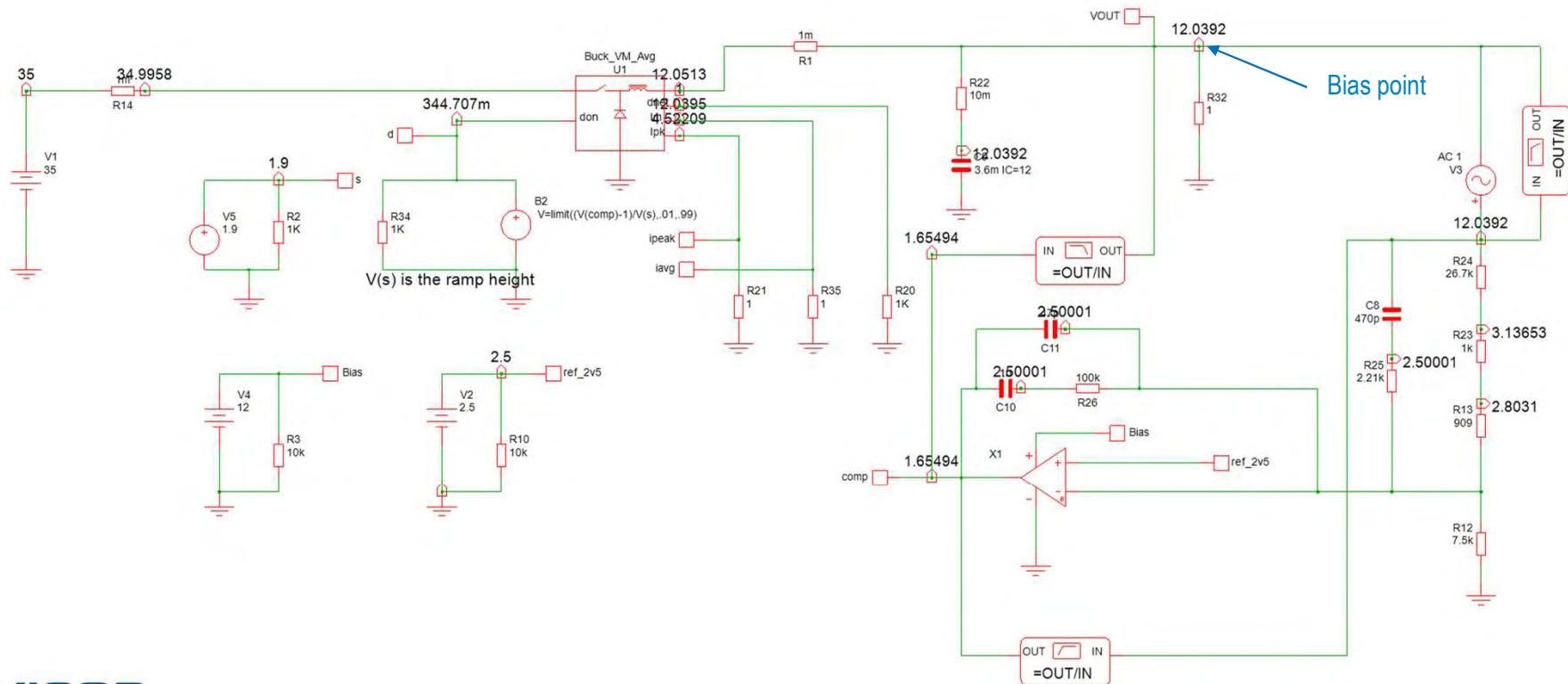
SPICE averaged model (valid for buck, forward topologies) example

Voltage coded
duty cycle
 $0 - 1V = 0 - 100\%$



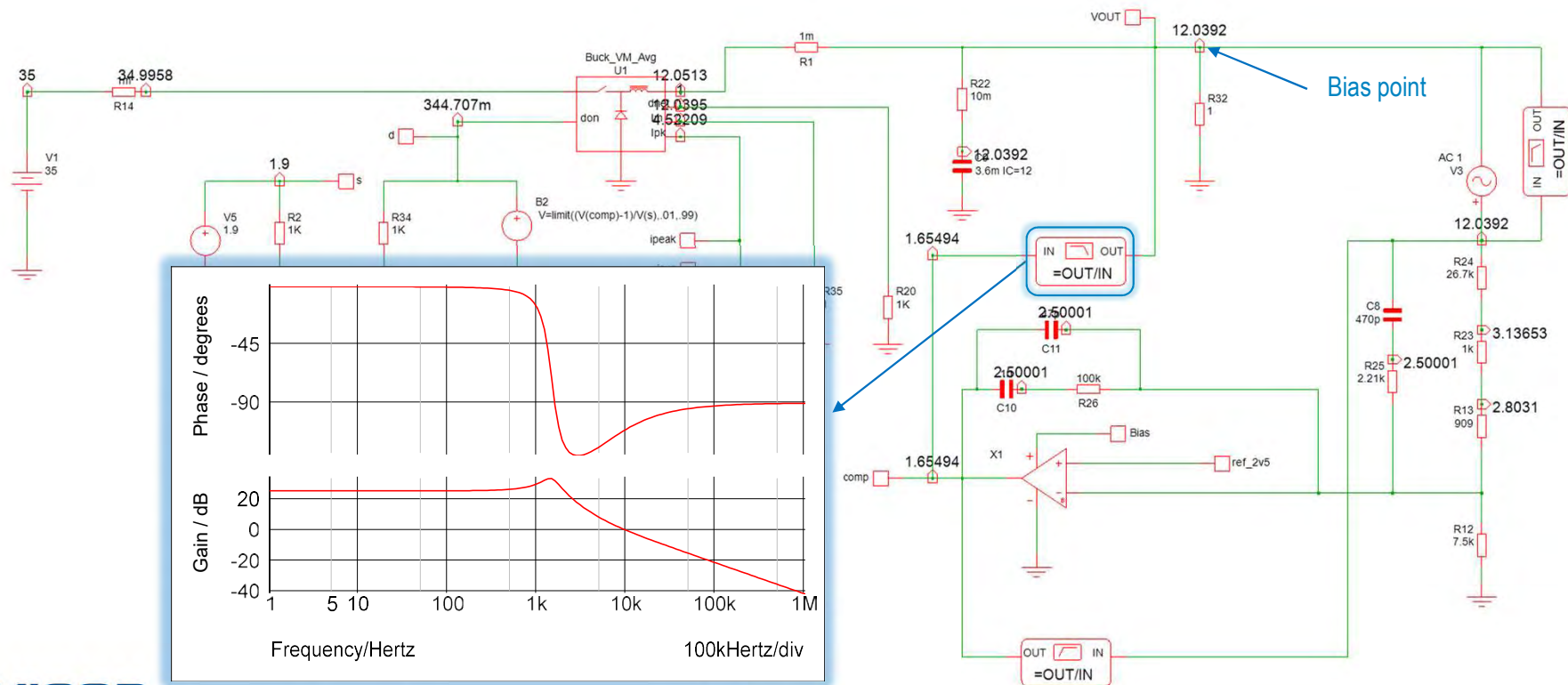
Averaged Model: Double-Ended Forward Converter

SPICE averaged model, closed-loop system



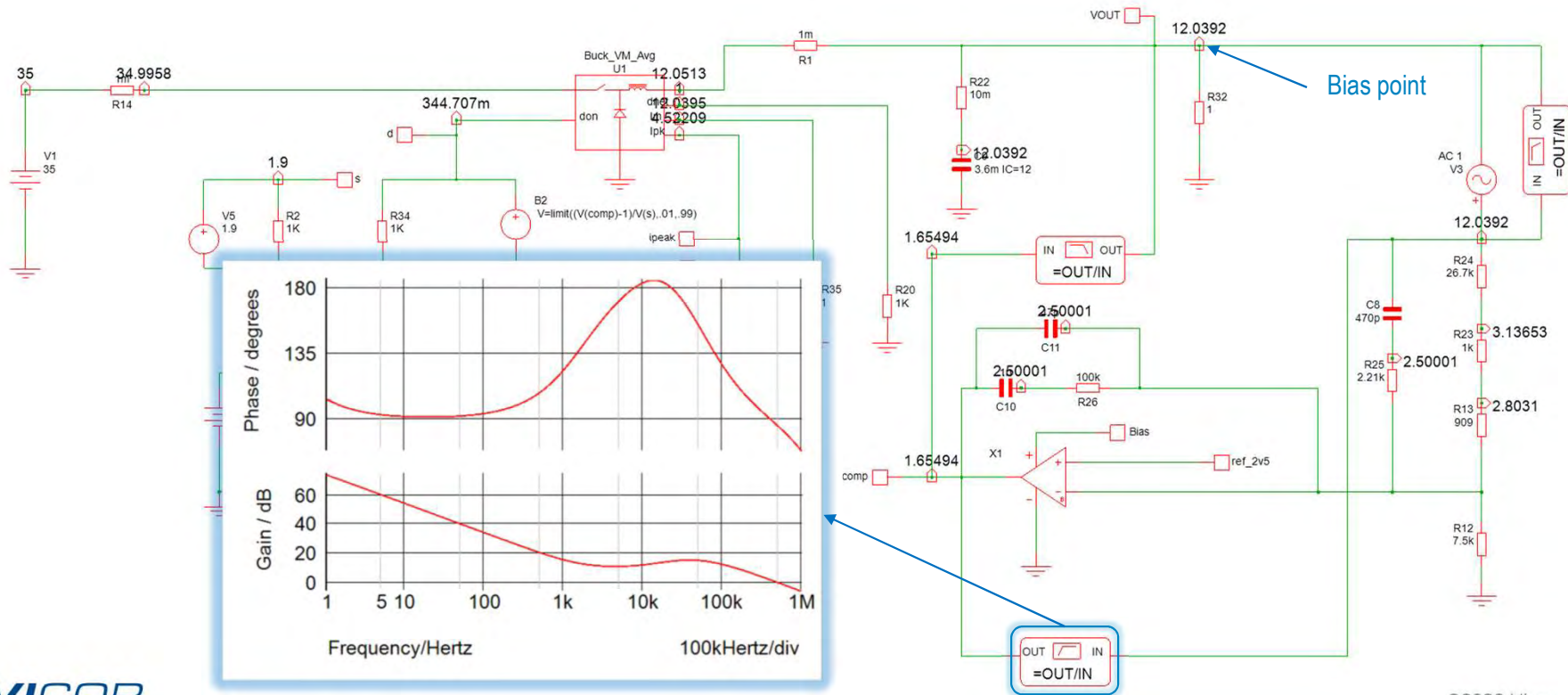
Averaged Model: Double-Ended Forward Converter

SPICE averaged model, closed-loop system



Averaged Model: Double-Ended Forward Converter

SPICE averaged model, closed-loop system

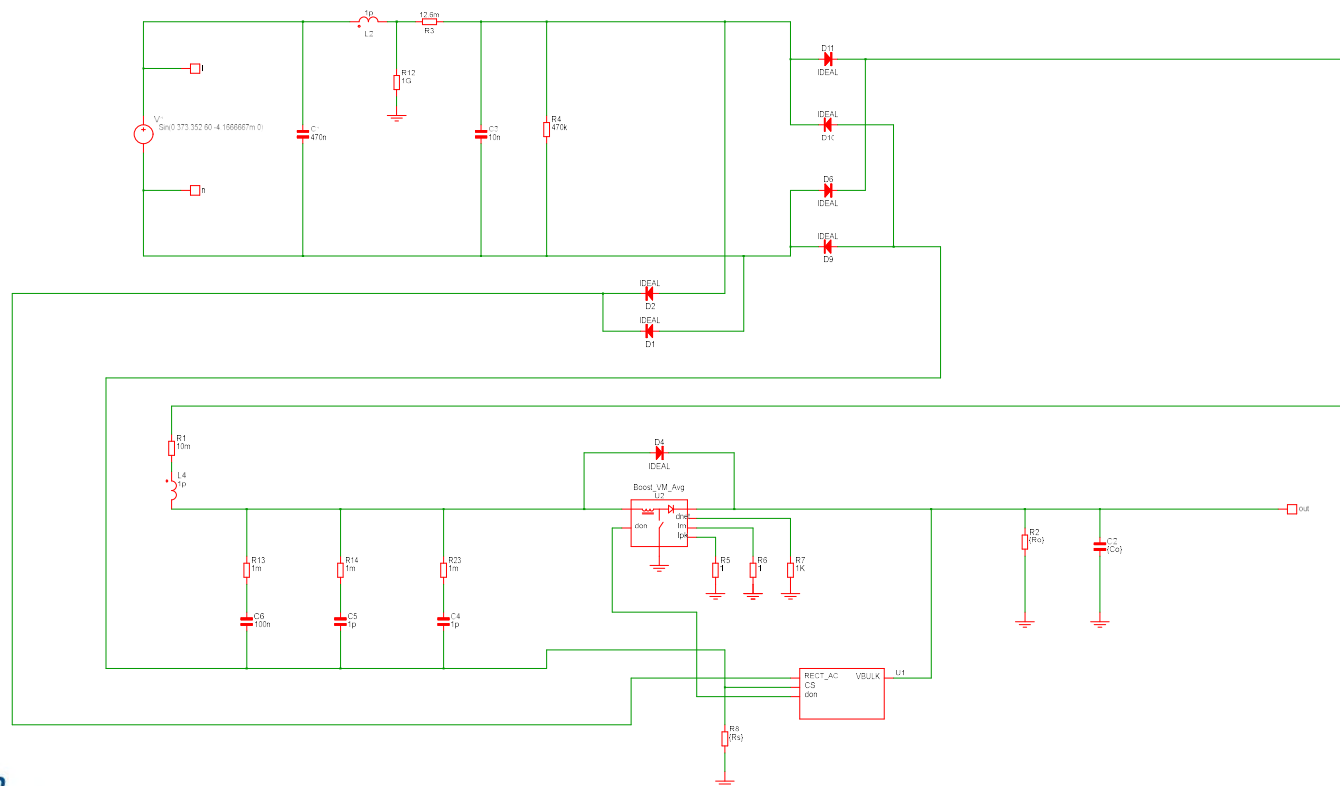


SPICE averaged model, closed-loop system



Averaged Model: Boost-Mode PFC

SPICE averaged model extended to boost-mode PFC

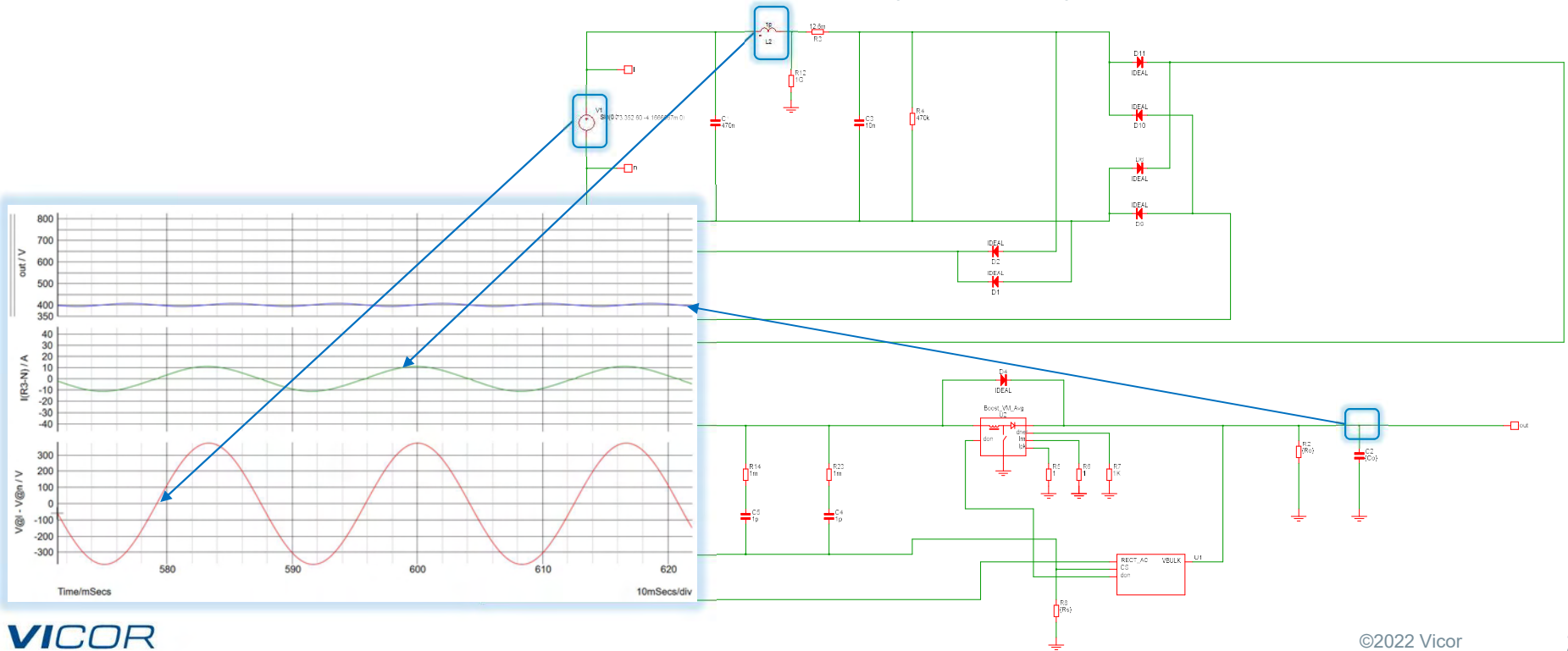


VICOR



One-Second Transient Performance

SPICE simulation of a one-second transient run: 8 seconds, fully-closed loop!





Understanding The Simulator

PWL-based tools and models

SIMPLIS, PSIM & NL5 Simulators

- **Faster transient analysis:** most are 10 – 50x (or more) faster than SPICE on switching power supplies
- **Same switching model for AC analysis**
- **Speed advantages in all cases**

SIMPLIS

PWL linear simulator with all models also already PWL

- No exponential expressions
- No conversion of non-linear models into linear equivalents
- More accurate solutions to the system of equations
- Solving faster than SPICE
- Larger time step
- The accuracy allows exact determination of the steady-state periodic operating point
- Very fast small-signal AC analysis in the time domain.

PSIM & NL5

Multi-sine or multi-sim AC analysis during transient simulation

- **Multi-sim:** transient analysis for every point of AC injection, computes the response from data
- **Multi-sine injection:** multiple frequencies in a single transient run

Some Limitations:

- **Limits the number of points swept:** transient simulation time must be equal to lowest period injected
- **Tedious with a large circuit with resonances:** steady state found manually for both PSIM and NL5 AC analysis
- **SIMPLIS POP is incompatible with multiple asynchronous converters** (only one POP trigger allowed): multi-sine method can also be employed for AC analysis, longer run times and point limitations apply, although all three simulators can achieve this MUCH faster than typical SPICE simulators.

All PWL-Based Tools

Benefit from simpler, faster models
(for what they are intended to do)

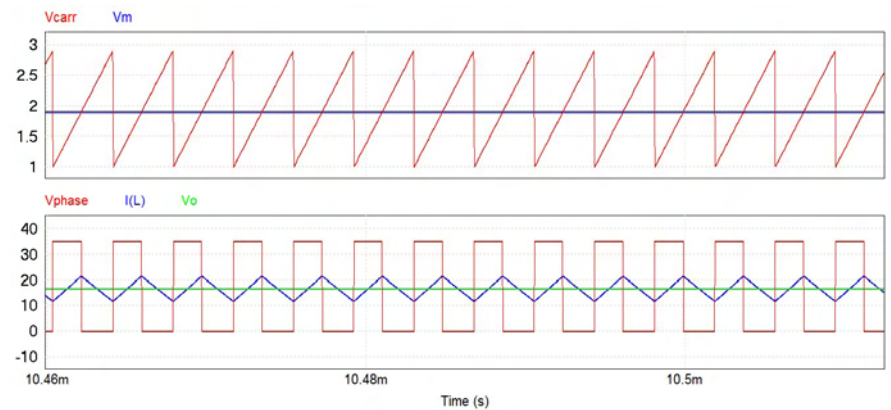
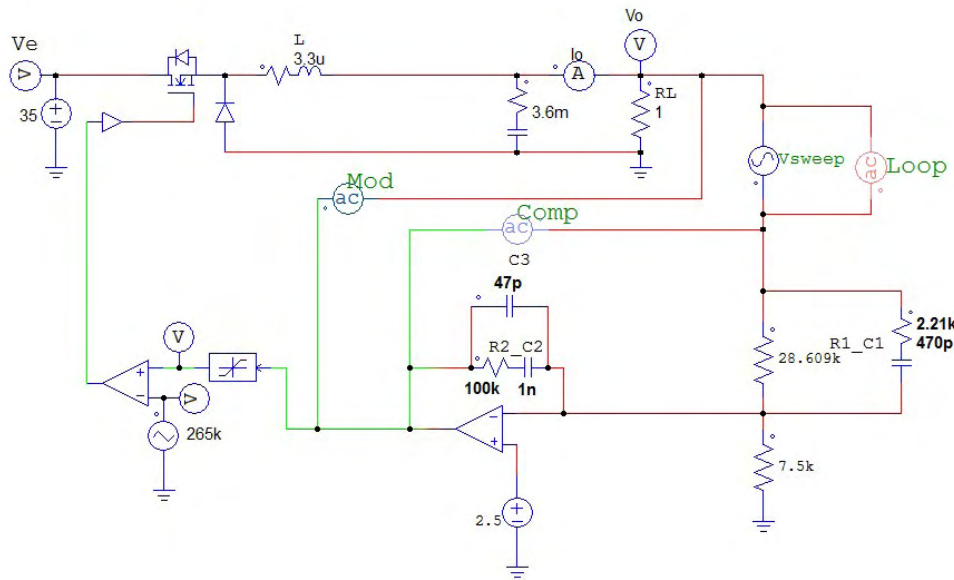
- **“ON” and “OFF” topologies:** MOSFET and power semiconductor are two topologies in their basic form
- **PSIM and SIMPLIS**
 - Higher level models, more topologies, small to moderate speed penalty
 - Determine switching losses fairly accurately

PWL simulators aren't perfect

- Semiconductors modeled as simple PWL approximations
- SPICE is better for the linear region of MOSFETs and log anti-log amplifiers
(or anything that requires the semiconductors to work as the device physics dictate)

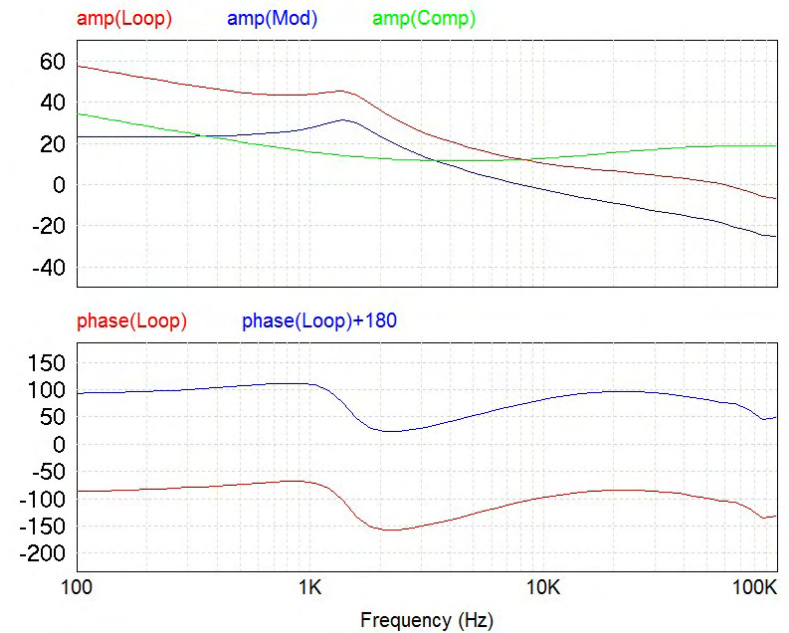
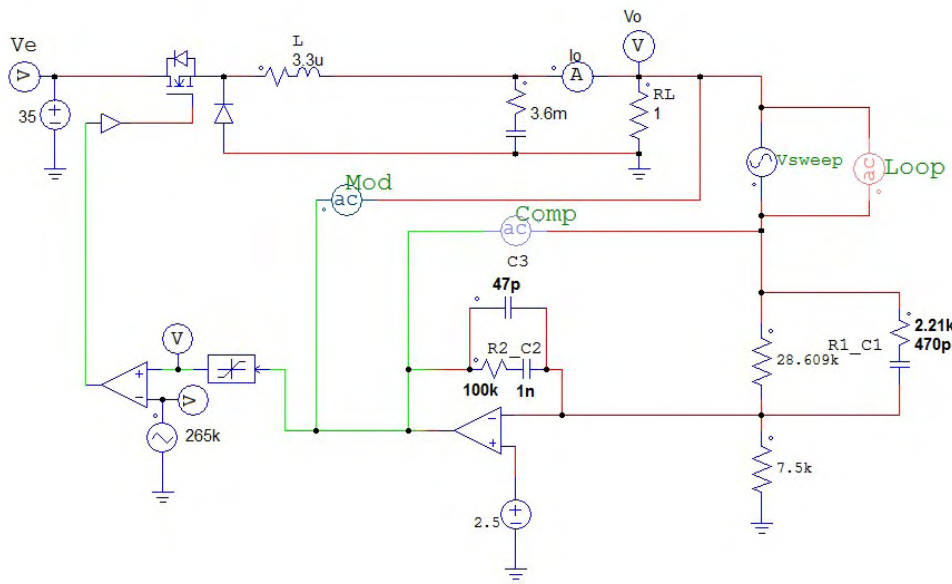
PWL Simulation: Buck Regulator Transient

PSIM switching buck regulator (equivalent of SPICE averaged model):
Only 7 seconds for full transient analysis of a 20ms period!



PWL Simulation: Buck Regulator AC Analysis

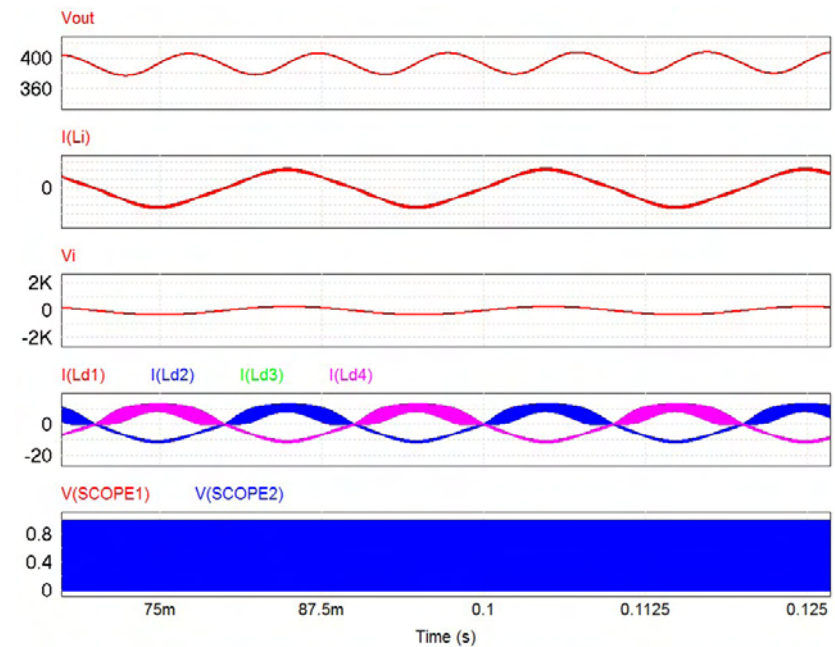
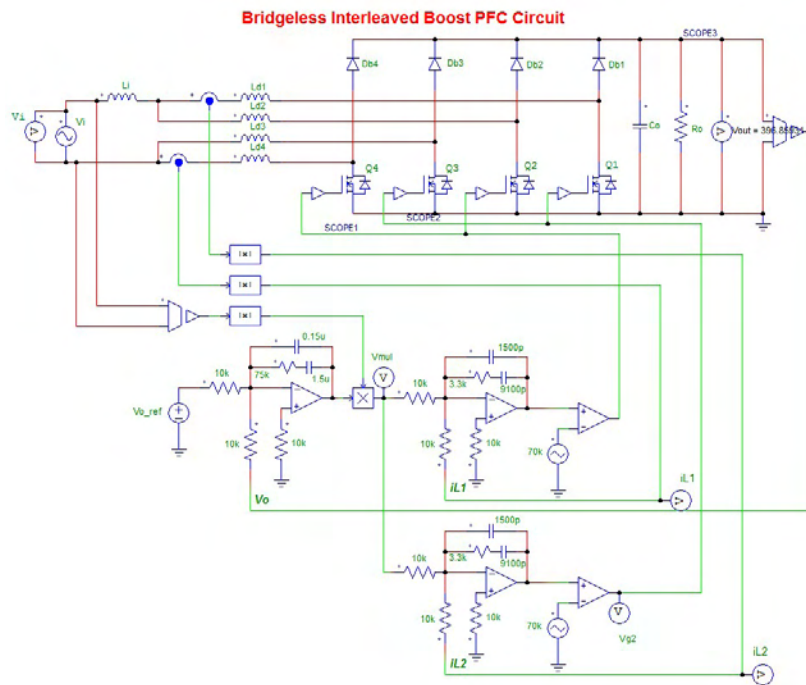
PSIM AC analysis of switching regulator:
Matches SPICE averaged model very closely



PWL Simulation: Interleaved PFC

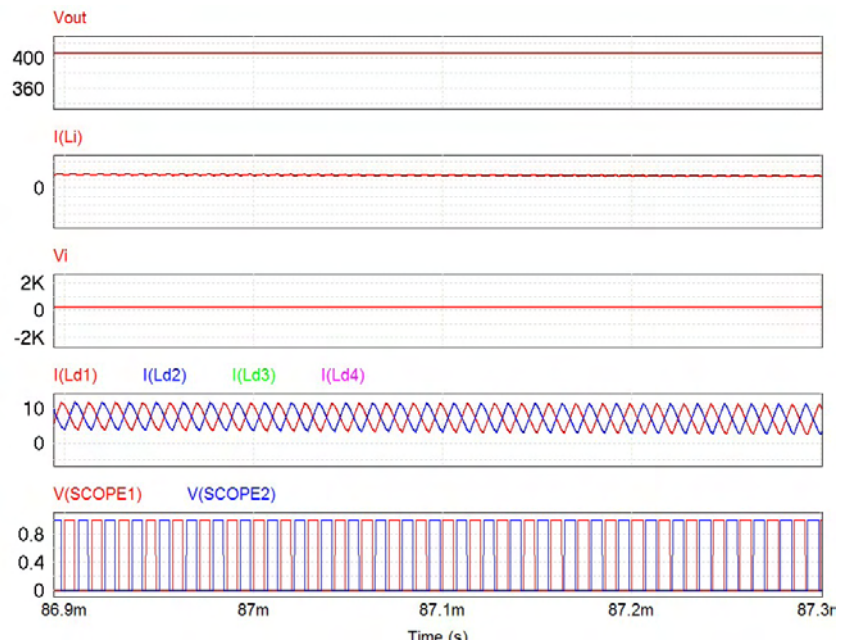
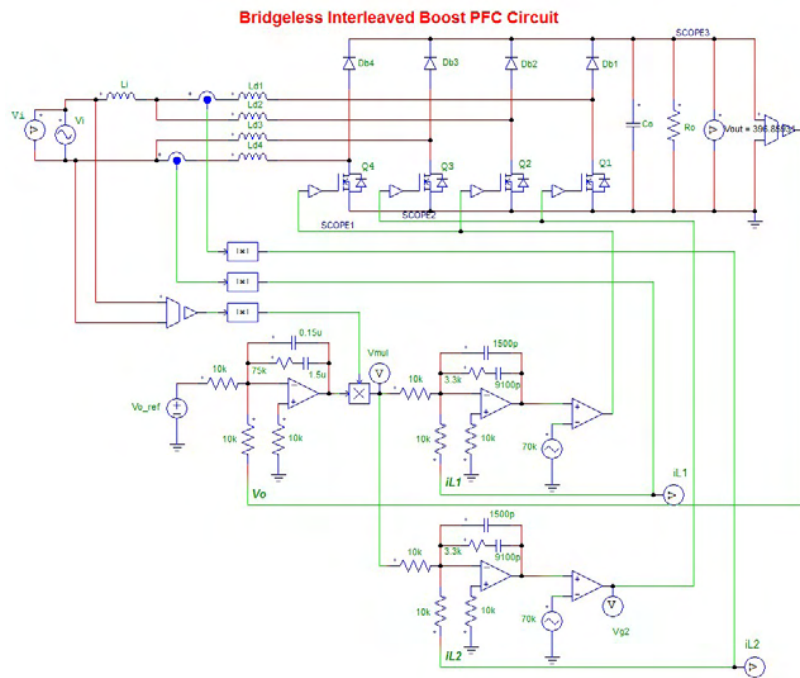
PSIM bridgeless interleaved PFC-switching model:

Runs closed loop 125ms in 34s with 100ns time step!



PWL Simulation: Interleaved PFC (Zoom In)

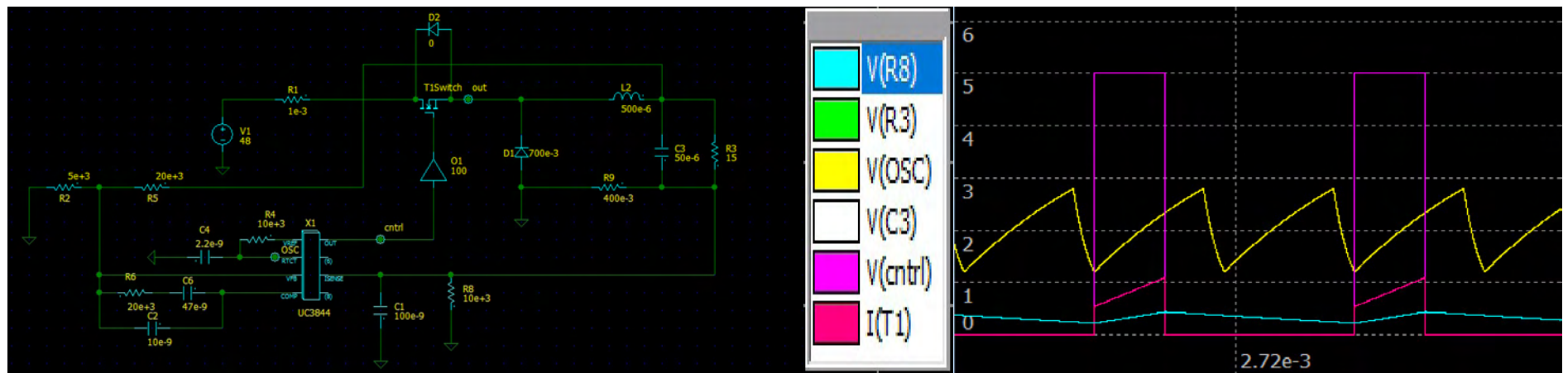
It is really switching, and no convergence issues.



PWL Simulation NL5: Current Mode Control

NL5 CM buck regulator:

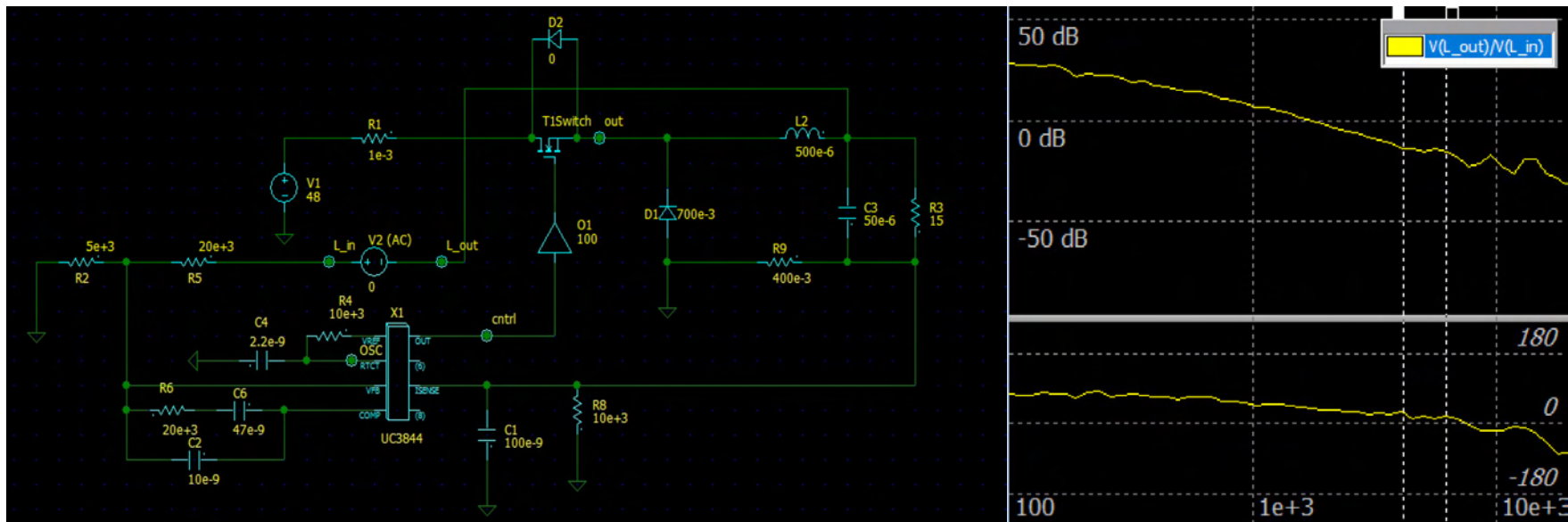
10s switching simulation takes just a few seconds with 10ns step



PWL Simulation: NL5 Current Mode Control

Just like the lab!

Insert AC source and generate the loop-gain plot while in switching mode with NL5.





PWL & SPICE

Choosing the right tool

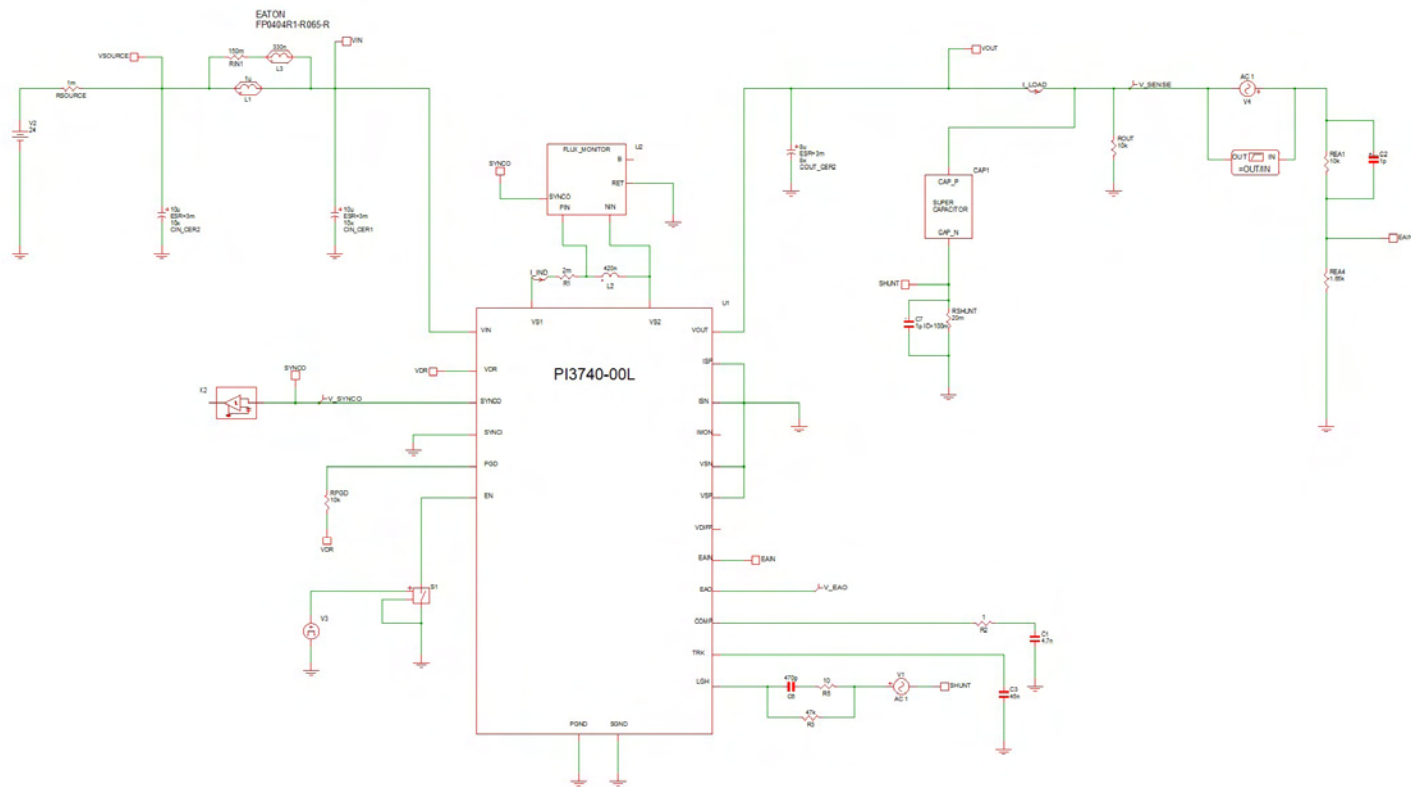
PWL & SPICE: Complementary Simulators

All of these great tools are useful!

- **Time domain challenge:** GPU processor PDN with 1500 R, L and C components, with tiny time constants
 - Averaged models (SPICE-based) run faster than the fastest PWL switching model but less accuracy with large PDN
 - Averaged model plus SPICE switching model: detailed results faster than switching alone using SPICE
 - High-level SPICE models offer higher fidelity simulations for switching loss analysis with detailed models
- **AC analysis on switching models:** SIMPLIS is the best choice
- **Long-run simulations (supercapacitor chargers):** SPICE-based averaged models are best for start up
- **Loop tuning with PDNs:** SPICE-based averaged models are best for rough in unless PDN is simplified

SIMPLIS Model

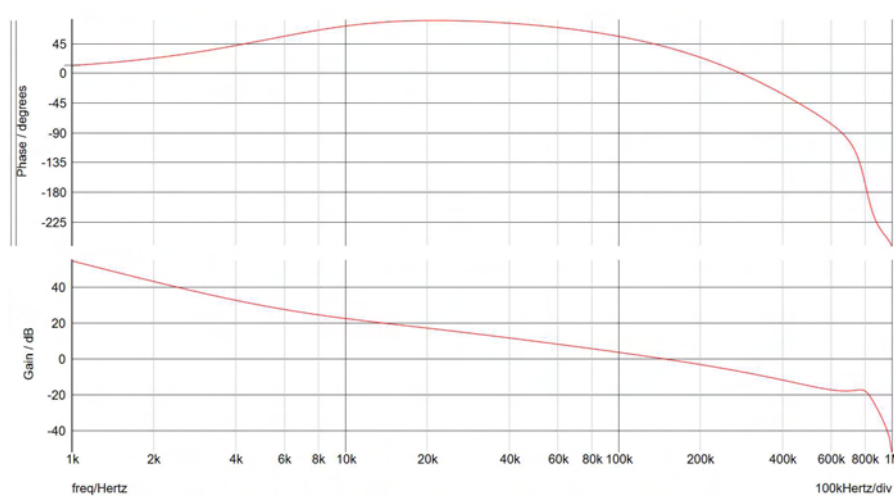
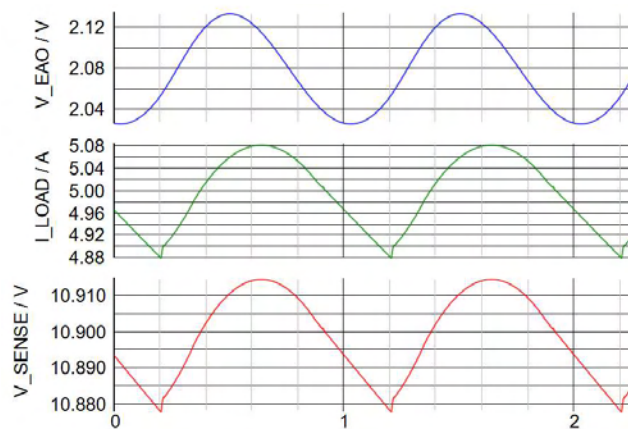
Vicor PI3740 Charging 5F Super-Capacitor Bank



SIMPLIS Model

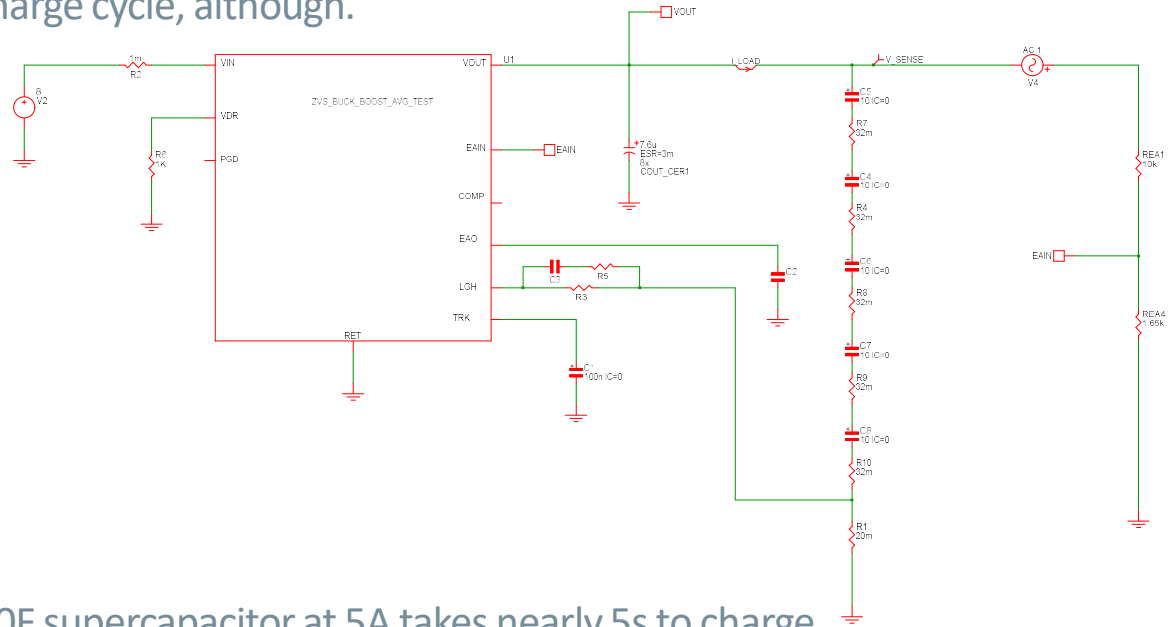
Finds Steady State With Capacitor At 10.9V

Can find and analyze current loop at any point during the charging in a few seconds!



Averaged Model & SIMPLIS Compared

SPICE-based averaged model: PI3740-00 start-up test simulation takes a few seconds. Much harder to do AC analysis at any point in the charge cycle, although.



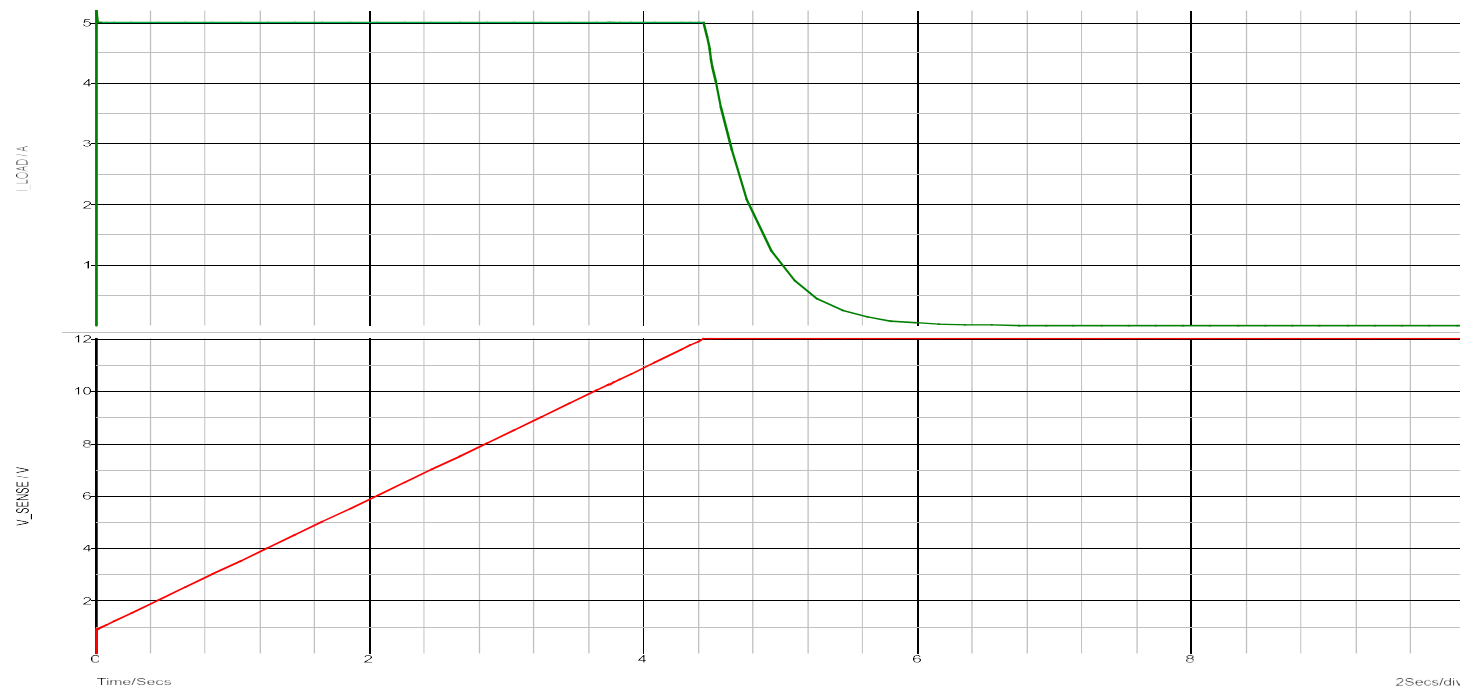
VS.

SIMPLIS simulation: each 10F supercapacitor at 5A takes nearly 5s to charge, entire simulation takes 2 hours with enormous data file from start up

VICOR

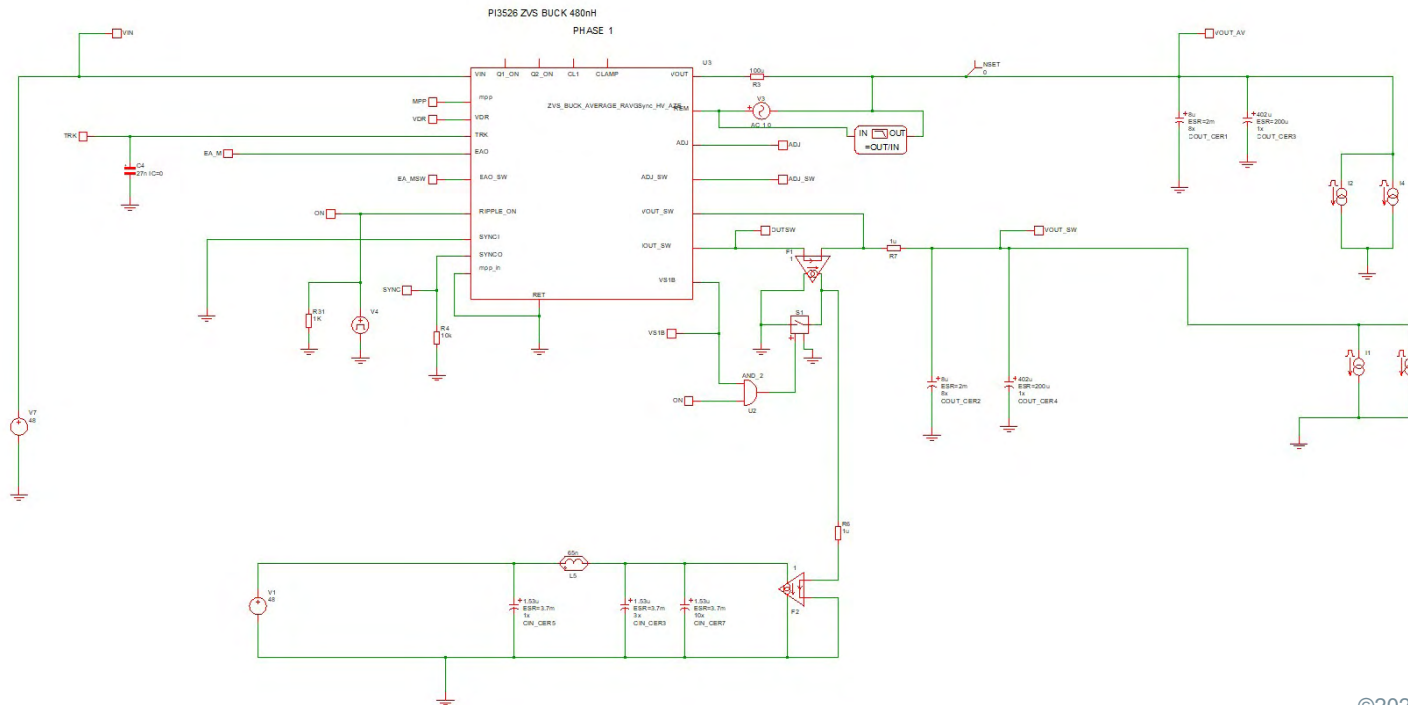
Transition From CC To CV Mode

Much more efficient in only a few seconds



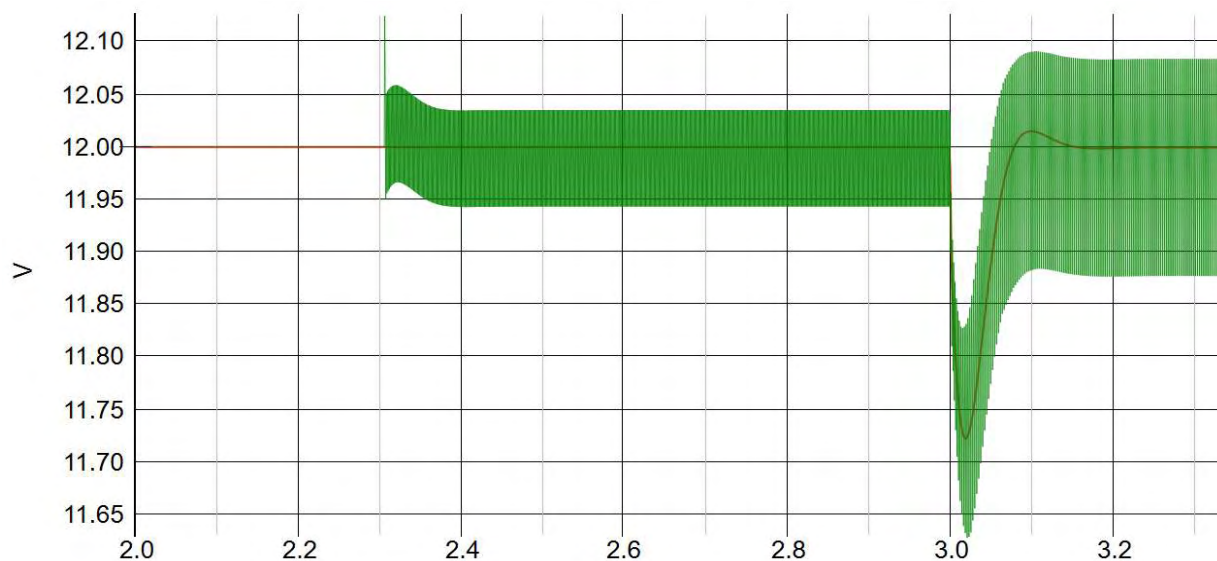
Combined Models: Averaged & SPICE Switching (SIMetrix)

- Averaged model programs the initial conditions for the switching to begin



Switching block turned on at 2.3ms, load step applied at 3ms

- Switching model quickly finds steady state
- Does not deviate much from the output voltage
- Averaged section used for AC analysis, switching section for ripple currents, switching losses, etc.





Strategy For Efficient Power Simulation

Use The Best Of Both

- **Pick your favorite SPICE-based and PWL simulators**
- **Develop models**
 - Averaged models for SPICE-based tools (not necessarily controller dependent)
 - PWL models especially when modeling controller behavior details (delays etc.)
 - Many PWL tools discussed here allow digital control and firmware simulation also.
- **Models should be generic:** parameter changes allow them to morph into another model.
 - Reuse of as many models as possible
 - Very fast creation of new models
- **Use cases**
 - Fastest models to rough in the design
 - Most accurate models for final analysis

Best Uses For Averaged Models (SPICE)

- **Multiple-converter current sharing**
- **Input filter and stability analysis of systems** with common input filters
- **Transient analysis for huge PDN networks:** this can swamp a switching model
- **Hot-swap, hot-plug, soft-start system-level analysis:** especially with multiple converters
- **Monte Carlo Analysis:** many runs needed to access CpK parameters for loop compensation, regulation, etc.
- **Loop tuning and analysis of multiple loop converters:** SPICE can break loops and find the correct bias point by using infinite inductors and capacitors (hard to do with PWL converters)
- **Detailed switching analysis:** averaged model can program a detailed switching model

Best Uses For PWL Models

- **Exact AC analysis** of the switching converter in time domain (SIMPLIS)
- **Very fast determination of switching waveforms** to measure ripple, EMI, RMS currents (PFC)
- **Transient analysis** where switching delay can affect transient response significantly
- **Verify exact switching theory of topologies:** averaged model can't predict sub-harmonic oscillation or flux walking in magnetics; switching model can verify accuracy of averaged model
- **Analysis of new topologies** (especially resonant ones) not well suited to averaged models or topologies you don't know how to model as an averaged model
- **Switched capacitor filters and charge pumps**
- **Interleaved converters** and topologies where ripple cancellation happens
- **Closed-loop output impedance, input impedance, audio susceptibility**

Required Power Simulations

| Simulation | Recommendation |
|--|---|
| Efficiency | Relative only to gauge changes – magnetics very difficult to model correctly using any circuit simulator. |
| Line and load regulation | PWL with higher level switches (PSIM or SIMPLIS) |
| Converter input/output impedance, closed/open loop | PWL unless many multiple converters |
| Transient response | All PSIM SIMPLIS or SPICE hybrid |
| Stability analysis, gain and phase margin with and without multiple loops (also digital-control loops) | SIMPLIS/PWL |
| Audio susceptibility | Averaged first PWL as final check |

Required Power Simulations

| Simulation | Recommendation |
|--|---|
| Capacitor input and output ripple voltage | PWL |
| MOSFET SOA analysis to include voltage and current stress (ringing etc.) | SPICE hybrid |
| Capacitor input and output RMS ripple current | PWL |
| MOSFET switching losses and conduction losses | SIMPLIS PSIM SPICE hybrid as final check |
| Conducted EMI | PWL |
| Line surge and immunity | SPICE |
| Input filter interaction, damping and attenuation to meet EMI specifications | PWL then SPICE |
| Current sharing (multiple converters) | SPICE averaged model |

Required Power Simulations

| Simulation | Recommendation |
|---|-------------------------|
| Synchronization (multiple converters) | PWL |
| Interleaving (multiple converters) | PWL or SPICE hybrid |
| Input impedance and filter interaction with multiple converters either independent or in parallel | SPICE |
| Thermal analysis | SPICE – see Appendix |
| PDN (power delivery network) analysis | SPICE averaged then PWL |
| Protection circuit performance (current limit, OVP etc.) | PWL |

References

Basso, Christophe P. *Switch-Mode Power Supplies: SPICE Simulations and Practical Designs.*

New York: McGraw-Hill, 2014

Kielkowski, Ron M. *Inside SPICE: Overcoming the Obstacles of Circuit Simulation.*

New York: McGraw-Hill, 1994

Ben-Yaakov, Sam *A unified SPICE compatible average model of PWM converters*

IEEE Transactions on Power Electronics, 1991

eCircuit Center (www.ecircuitcenter.com)

Powersim (www.powersimtech.com)

NL5 (www.sidelinesoft.com)

SIMetrix/SIMPLIS (www.simplistechnologies.com)



Thank you

Questions?

The information contained herein and presented by Vicor is for general informational purposes only. Vicor assumes no responsibility for inaccuracies, errors or omissions in this presentation. Users of power supply products remain responsible for the design, testing and operational safeguards related to such use.

Excel is a registered trademark of Microsoft Corporation.

ICAP/4 is a registered trademark of Intusoft

LTspice® is a registered trademark of Analog Devices, Inc.

MATLAB® and Simulink® are registered trademarks of The MathWorks, Inc.

PSIM® is a registered trademark of Altair Inc.

RidleyWorks® is a registered trademark of Ridley Engineering, Inc.

SIMPLIS is a registered trademark SIMPLIS Technologies, Inc.

TINA is a registered trademark of DesignSoft, Inc.

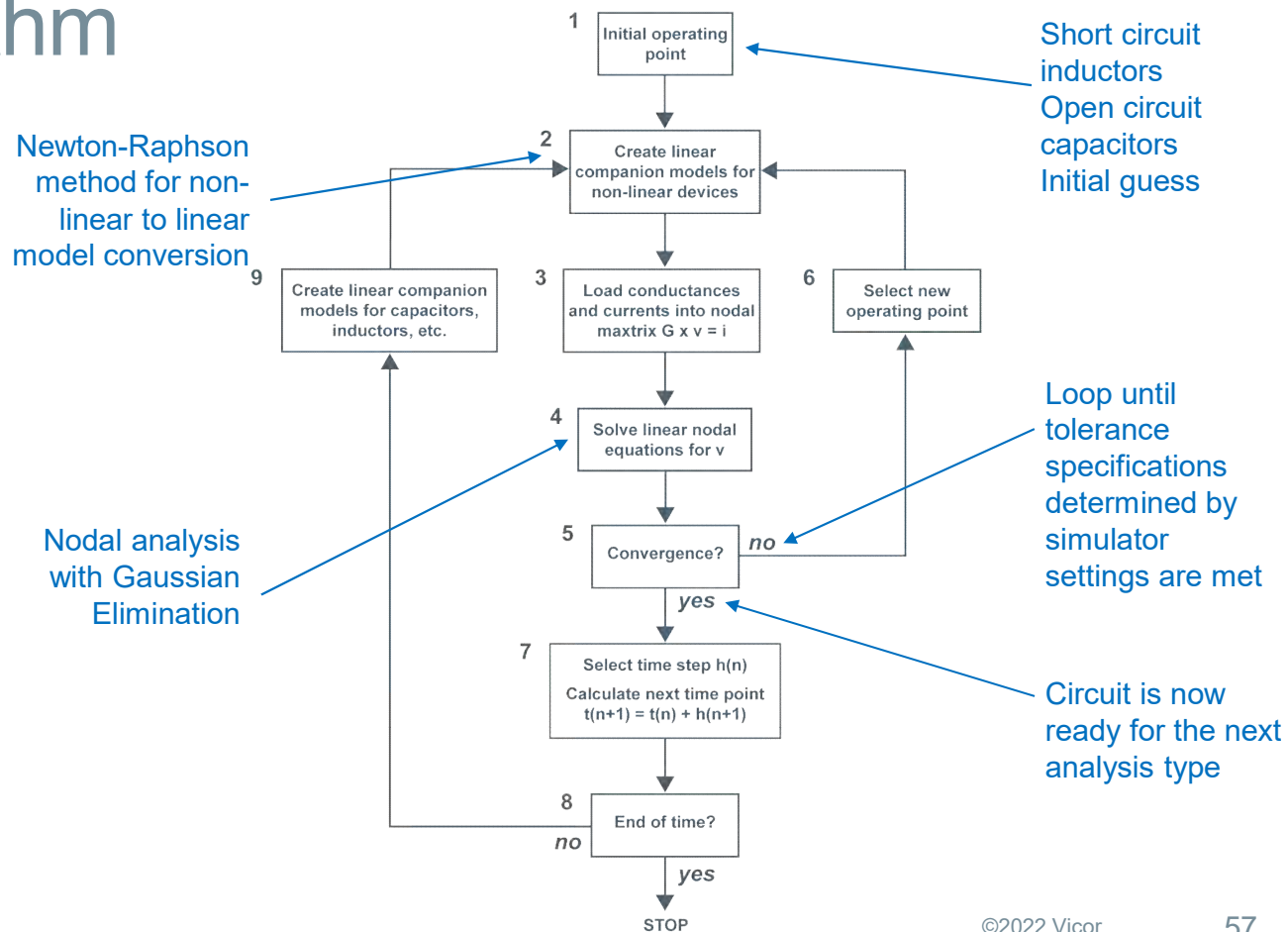
All other trademarks, product names, logos and brands are property of their respective owners.



Appendix

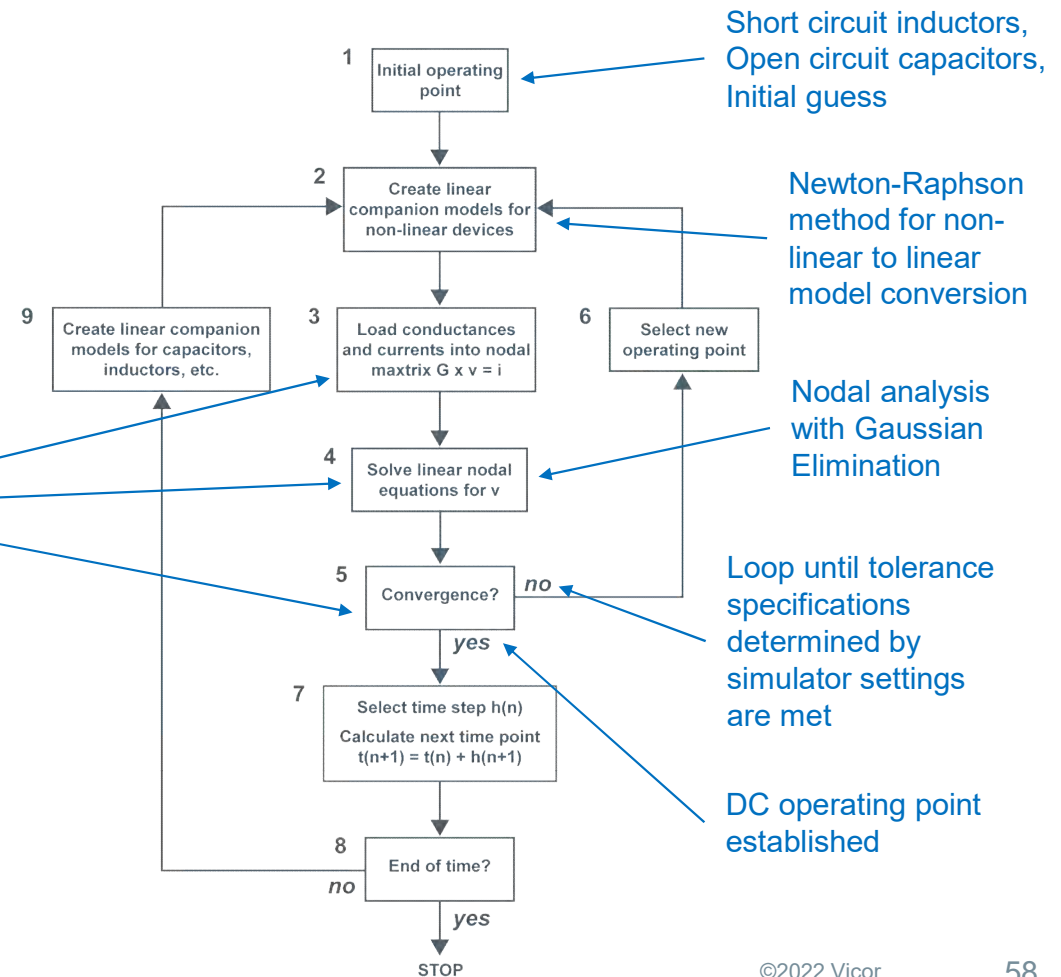
Further material for your enjoyment

SPICE .OP & .DC analysis simulator algorithm



SPICE .AC analysis simulator algorithm

1. Load matrix with impedance for R, L and C (real, imaginary) components
2. Replace non-linear circuits with small-signal equivalent
3. Set DC sources to zero, AC signal to unity, and repeat blocks 3 and 4 for each frequency of interest



SPICE .TRAN (Transient) Analysis Simulator Algorithm

1. Blocks 7 – 9

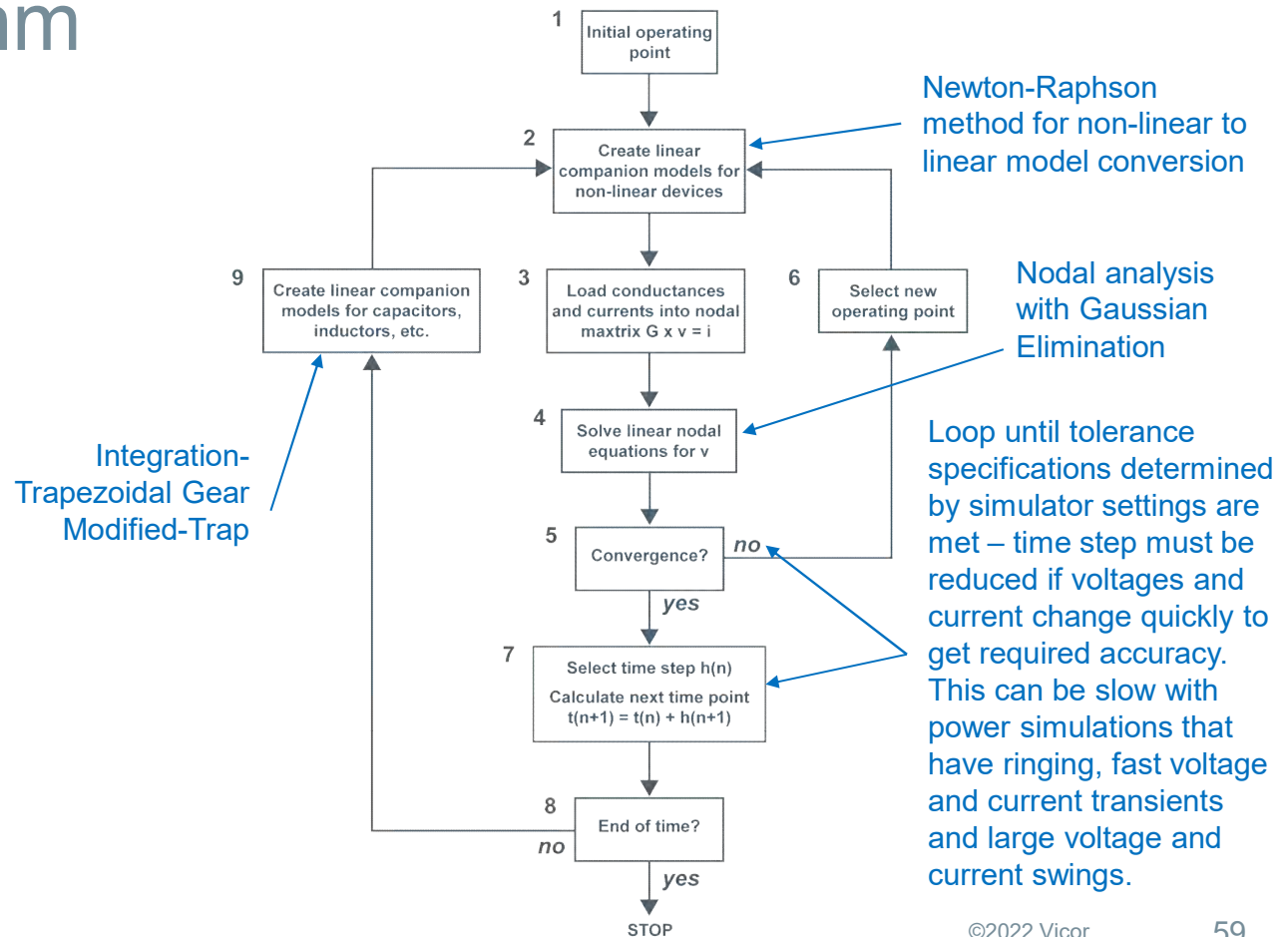
Create linear companion models for charge storage devices (Ls,Cs and Semiconductors)

2. Blocks 2 – 6

Non-linear to linear models

3. Blocks 2 and 5

Skipped for linear models



Thermal Modeling A Double Sided PCB In SPICE

Based on the following

- › Lateral thermal resistance for 3 Oz copper is 24 degrees C/W
- › Thermal resistance (convection) to ambient per square inch of copper is 155 degrees C/W
- › Via thermal resistance assumed to be 10 degrees C/W from slug to bottom copper layer
- › Copper is 3 Oz, two sided.
- › Assume 5 degrees C/W for Junction to PCB.
- › Model is a 1 X 10 inch circuit board assuming 100% copper coverage on both sides. (planar model)
- › Current is the dual of heat flow (Amps = Watts)
- › Voltage is the dual of temperature (Volts = degrees C)
- › Squares can be added to assess any size board and copper arrangement
- › Can provide first order analysis for those without thermal simulation capability

Thermal Modeling A Double Sided PCB –Use .DC or .OP Voltage = Temperature

