2023 **Annual Conference & Exposition**
Baltimore Convention Center, MD | June 25 - 28, 2023

The Harbor of Engineering
Education for 130 Years

ASEE

Paper ID #40162

Board 89: Work in Progress: Use of Simscape in an Introductory Power Electronics Course

Dr. Cherian Mathews, University of the Pacific

Cherian P. Mathews is a Professor in the Department of Electrical and Computer Engineering at the University of the Pacific. He received a B.E. degree in Electrical and Electronics Engineering from Anna University, Chennai, India, and M.S. and Ph.D. degrees in Electrical Engineering from Purdue University. He was a tenured faculty member at the University of Florida / University of West Florida Joint Program in Electrical and Computer Engineering prior to joining University of the Pacific and has held visiting faculty positions at Purdue University and Rose-Hulman Institute of Technology.

Use of Simscape in an Introductory Power Electronics Course

Abstract

University of the Pacific was a member of an 82-university consortium led by the University of Minnesota (UMN) that was supported by a three-year Department of Energy grant to "revitalize electric power engineering education by state-of-the-art laboratories." Hardware developed at UMN served as the core of a new power electronics course developed in conjunction with this grant. UMN hardware labs were complemented using PSpice for circuit simulation. The PSpice exercises developed at UMN were built around PSpice 9.1 and required custom blocks to simulate components such as ideal transformers with variable turns ratios. Recent offerings of the power electronics course had difficulties with using PSpice for simulation such as: difficulty porting and accessing the PSpice 9.1 custom blocks in the Cadence PSpice software available on campus computers; IT issues resulting in frequent crashes of Cadence that left students with unreliable software access; inability of students to access Cadence remotely. Due to these challenges the author decided to work on replacing PSpice by Simscape for simulations in the Spring 2022 offering of Power Electronics.   The transition required significant effort but was successful: all prior PSpice simulations were able to be implemented in Simscape. The paper presents simulation of a buck converter, the dynamic average model of a buck converter, and closed-loop voltage mode control of a buck converter using Simscape and contrasts the features of these simulations against PSpice-based simulations. While there is some published work on use of Simscape in power electronics, the present work relates to UMN labs available in the public domain and is thus of relevance to the 82 universities in the Department of Energy grant consortium.

Formal assessment of the effectiveness of Simscape versus PSpice as a pedagogical tool in the context of an introductory power electronics course has not been performed yet. A survey on student preferences for Simscape versus PSpice for simulation showed a definite leaning towards Simscape, with ease of access to Matlab/Simscape software being a strong contributing factor.

Introduction

The University of Minnesota (UMN) was the lead institution that was awarded a Department of Energy (DOE) grant [1] over the 2010-2013 period to create "A nationwide consortium of universities to revitalize electric power engineering education by state-of-the-art laboratories." The consortium consisted of 82 universities that used UMN developed laboratory hardware and software resources to set up laboratories in their home institutions. University of the Pacific was a member of the consortium and used the grant to develop a new power electronics course and lab. Hardware lab experiments were complemented by PSpice exercises for simulation of power electronic circuits. The UMN developed PSpice labs were built around PSpice 9.1; the lab manual and the PSpice 9.1 software download are publicly available [2]. The PSpice labs use a custom block for generation of pulse-width modulated signals and another to implement an ideal transformer to model the dynamic average behavior of the switching power poles present in buck, boost, and buck-boost converters. PSpice 9.1 is not officially supported under recent versions of Windows and it took some effort to port these blocks to work under Cadence for

which a campus license is available. This made running the simulations on campus computers possible, but there were difficulties; simulations often would not work on some lab computers. The access problems were compounded when Cadence began to experience repeated crashes that required the license server to be rebooted each time. This made student access to Cadence unreliable. Even apart from the technical issues, Cadence was not accessible to students from off campus without a virtual private network, for which permission was not normally granted. All these factors led the instructor to consider other options for simulation of power electronic circuits. PLECS was a simulation option – involving licensing costs – that appears to have been subsequently adopted by UMN (the lab manual for the PLECS-based labs is also available [2].) The instructor chose to try MathWorks' Simscape for simulation since the university already had a campus-wide license that allowed students to download Matlab/Simulink/Simscape on their personal computers. The transition from PSpice to Simscape occurred during the Spring 2022 offering of the Power Electronics class and was successful; all simulations were able to be performed in Simulink/Simscape and students had easy access to the software on their own laptops. A student survey comparing use of Simscape versus PSpice for simulation of power electronic circuits indicated a strong preference for Simscape: 8 or more out of the 10 survey respondents rated Simscape as better than PSpice in each of the following categories: effectiveness in analyzing circuit behavior and understanding circuit operation, ease of use of the software, and accessibility of the software.

A literature searched revealed a few sources that address the use of Simulink/Simscape for simulation of electronic circuits. The use of Simulink/Simscape for simulation of power electronic circuits is addressed in broad terms in [3,4], while [5] describes the use of Simscape in an Electronics course. Finally, [6] describes the use of Simscape in Power Electronics and Electric Machines courses. While [6] does describe simulation of a boost converter using Simscape, the present work is tailored more to the UMN developed software simulations which are likely to be of interest to members of the DOE consortium. It includes dynamic average simulation of converters, which is not covered in [6].

Circuit Simulation using PSpice 9.1 and Simscape

Fig. 1 depicts the schematic diagram (created in PSpice 9.1) of an ideal buck converter. The converter parameters are input voltage $V_{in} = 20V$, switching frequency $F_s = 100kHz$, switching period $T_s = \frac{1}{F_s} = 10\mu s$, inductance $L = 0.1mH$, output capacitance $C = 10\mu F$, and load resistance $R = 15\Omega$. The control voltage $0.75V$ sets the duty cycle D and causes the custom block *pwm_saw* to create a 75% duty cycle Pulse Width Modulated (PWM) signal to drive the switch $S1$. The switch S1 closes when the PWM signal is High and has internal resistance of $1m\Omega$ when closed. The average output voltage of the converter (assuming ideal devices) is $V_o = DV_{in} = 20 \times 0.75 = 15V$. The expected inductor current ripple is $\Delta i_L = \frac{V_o(1-D)T_s}{L} = 0.375A$, and the output voltage ripple is $\Delta V_o = \frac{T_s\Delta i_L}{8C} = 47mV$.

Fig. 2 shows traces of the converter output voltage (left trace) and inductor current (right trace.) Cursors placed on the peak and the trough allow us to measure the peak-to-peak values of the traces. The output voltage ripple is seen to be 48.67mV (visible in the Probe Cursor display) and

the inductor current ripple is seen to be 0.387A, values that are close to the theoretical predictions.
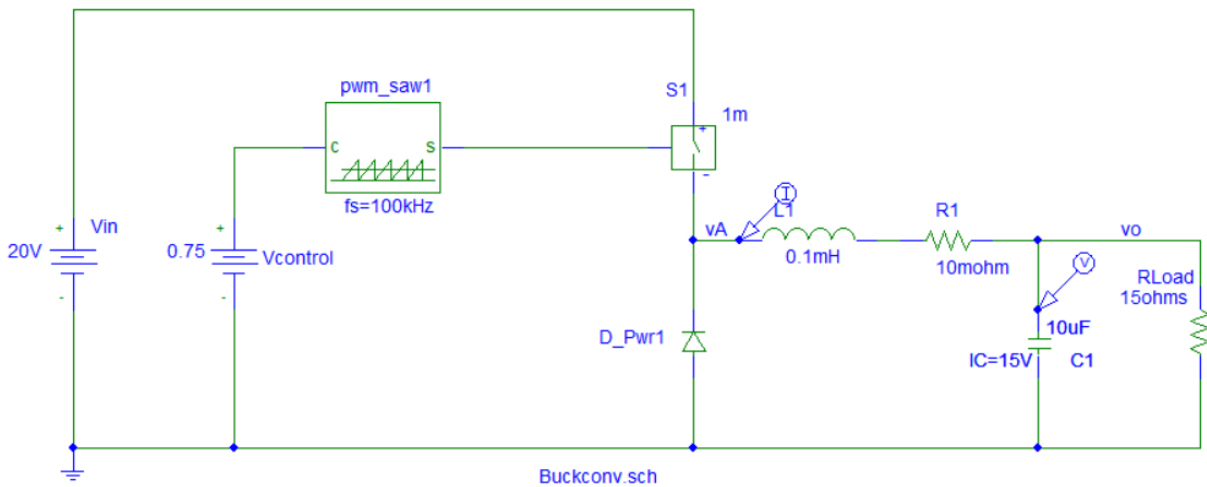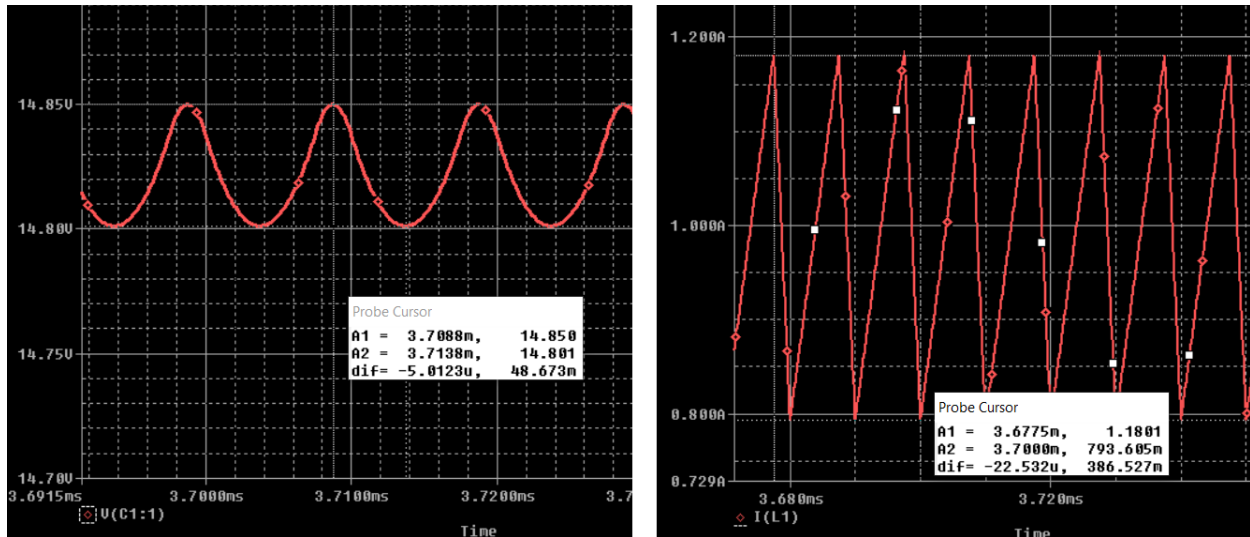

Figure 1: Ideal buck converter schematic in PSpice 9.1

Figure 2: Output voltage V_o (left trace) and inductor current I_L (right trace) in PSpice 9.1

Fig. 3 depicts the schematic diagram of the buck converter in Simulink/Simscape, with the names of each of the blocks listed in blue font. The *Series RLC branch* block is also used to implement the load resistor and capacitor in the schematic. The converter output voltage and the inductor current signals are fed to *Scope* block parts where the corresponding waveforms, depicted in Fig. 4, can be viewed. Signal statistics can be displayed for each scope waveform; the statistics show that the output voltage ripple (peak-to-peak) is 49.75mV and the inductor

current ripple is 0.377A. These measurements are very close to the theoretical predictions and the results from PSpice.
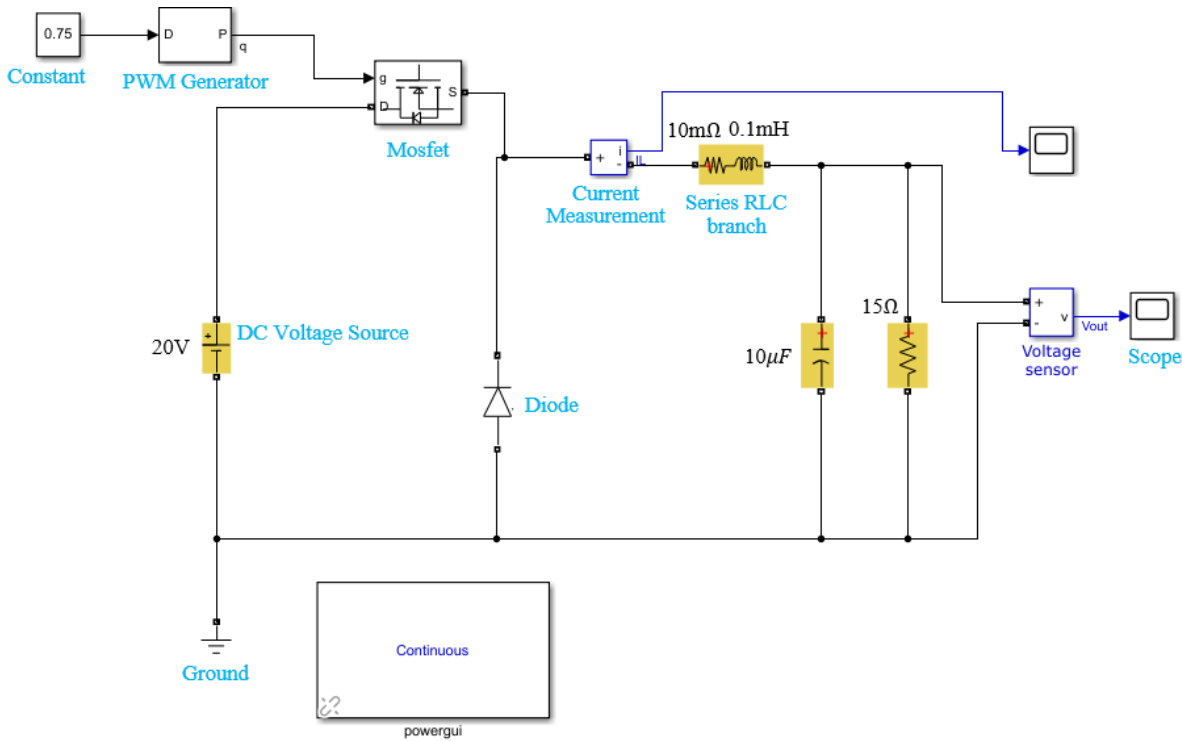

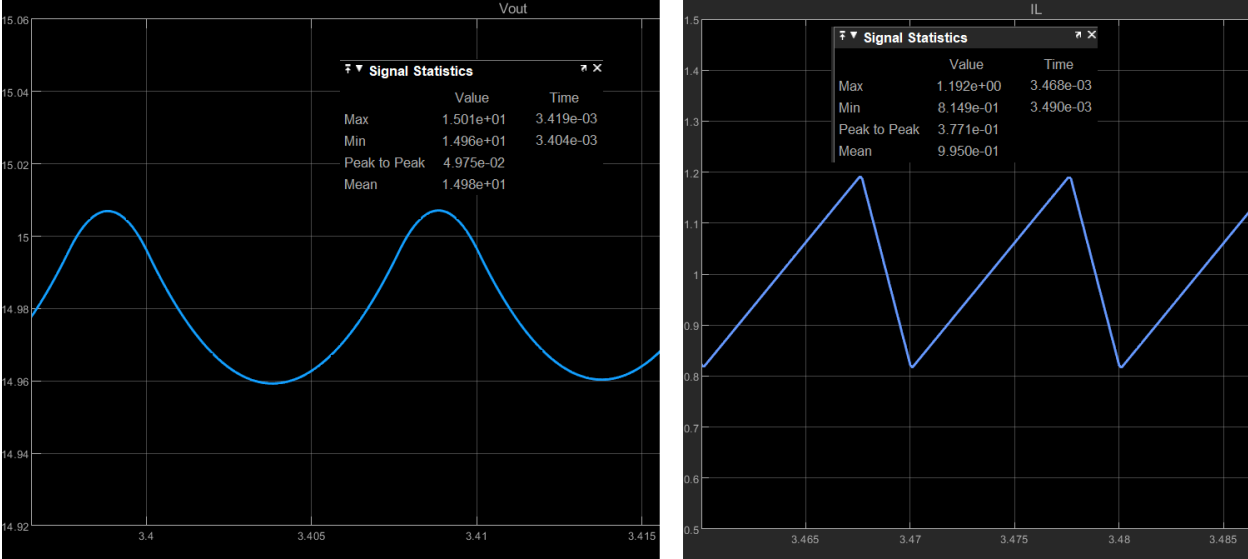Figure 3: Schematic of the buck converter in Simscape

Figure 4: Simulink waveforms of converter output voltage V_o (left trace) and inductor current I_L (right trace)

The next set of simulations involve the dynamic average model of a buck converter. The heart of a buck converter is a switching power-pole, which consists of a switch (MOSFET in Fig. 5) and a diode. The voltage and current at the input of the power pole have subscripts vp (denoting a voltage port) while the output port quantities have subscripts cp (denoting a current port.) The PWM signal $q(t)$ of duty cycle D turns the switch off and on. If variations in duty cycle occur at a very low frequency compared to the transistor switching frequency, the dynamic averages of the power-pole voltages and currents are known to related as follows:

$$V_{cp} = DV_{vp} \qquad\qquad (1)$$
$$I_{vp} = DI_{cp} \qquad\qquad (2)$$

These relationships (in which upper case letters denote the average voltage in the vicinity of a specific time instant) match those of an ideal transformer with turns ratio $\frac{V_{vp}}{V_{cp}} = \frac{I_{cp}}{I_{vp}} = \frac{1}{D}$.

Replacing the switching power pole by an ideal transformer (with variable turns ratio $= 1/D$) that is placed between the voltage and current ports allows for buck converter circuits to be simulated without including high frequency transistor switching. Simulations run a lot faster and predict the dynamic behavior of the buck converter well.
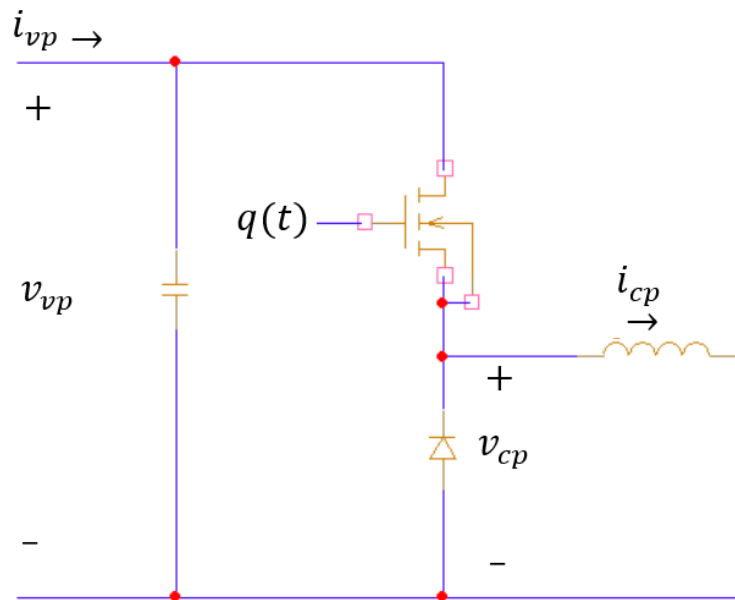

Figure 5: Voltages and currents of a switching power-pole

Fig. 6 depicts the PSpice schematic for dynamic average simulation of a buck converter. The custom block *ideal-trans-CCM* in the schematic lies between the voltage and current ports and implements the relationships of Eqs. (1) and (2). The duty cycle $D = 0.75$ (implemented by the voltage source $V_{control}$) sets the turns ratio of the transformer. Fig. 7 depicts the Simscape schematic for dynamic average simulation of the converter. The *Variable-ratio transformer* block implements the relationships of Eqs. (1) and (2). The simulation includes a switched load; the switch depicted in Fig. 6 closes at $t = 5ms$, reducing the load resistance from 15Ω to 10Ω at $t = 5ms$. The *Ideal Switch* in Fig. 7 and the *Step,* which occurs at $t = 5ms$ implements the

switched load in Simscape. The expected converter output voltage is $V_o = DV_{in} = 0.75 \times 20 = 15V$. The load current rises from $\frac{V_o}{15} = 1A$ to $\frac{V_o}{10} = 1.5A$ at $t = 5ms$.
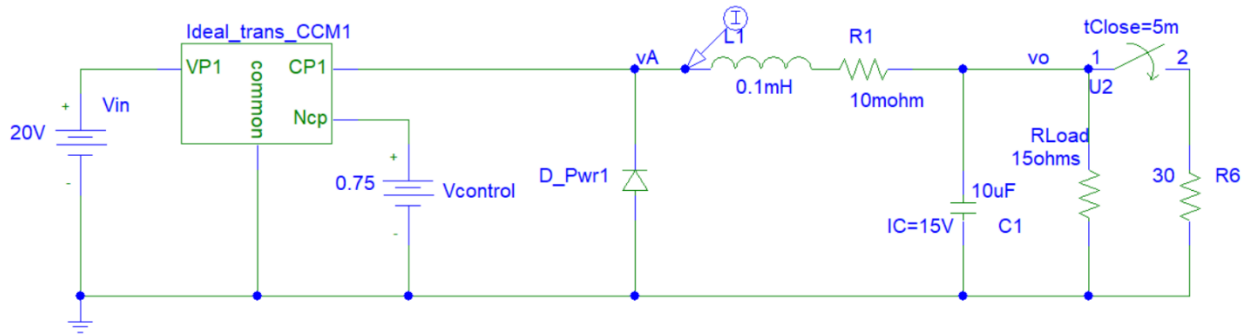

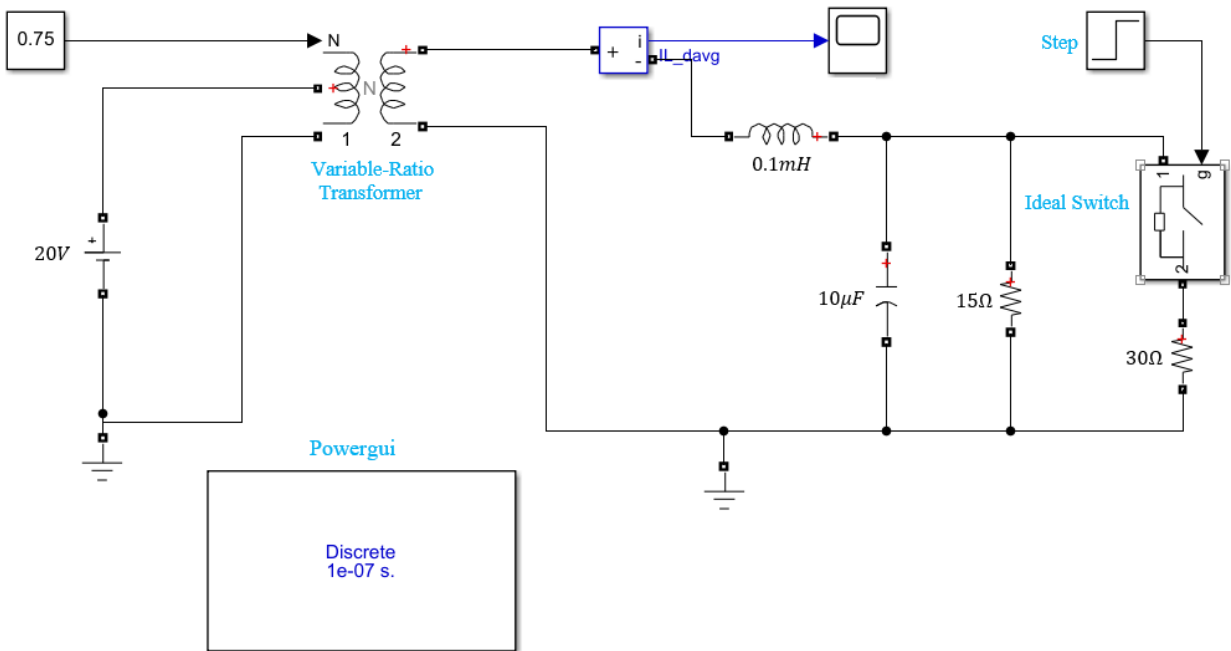
Figure 6: Dynamic average simulation using PSpice.

Figure 7: Dynamic average simulation using Simscape.

Fig. 8 shows the dynamic average inductor current waveform (red trace) resulting from the simulation of Fig. 6. The actual inductor current waveform that includes transistor switching (green trace) is superimposed on the plot. As expected, the dynamic average current is $1A$ at $t = 5ms$ and rises and settles at $1.5A$ after the transients. The dynamic average model predicts the trajectory of the inductor current accurately, while the exact waveform shows the inductor current rise and fall in each switching cycle. The corresponding waveforms from the Simscape simulation of Fig. 7 are depicted in Fig. 9. The Simscape simulation results clearly are very similar to the PSpice simulation results.
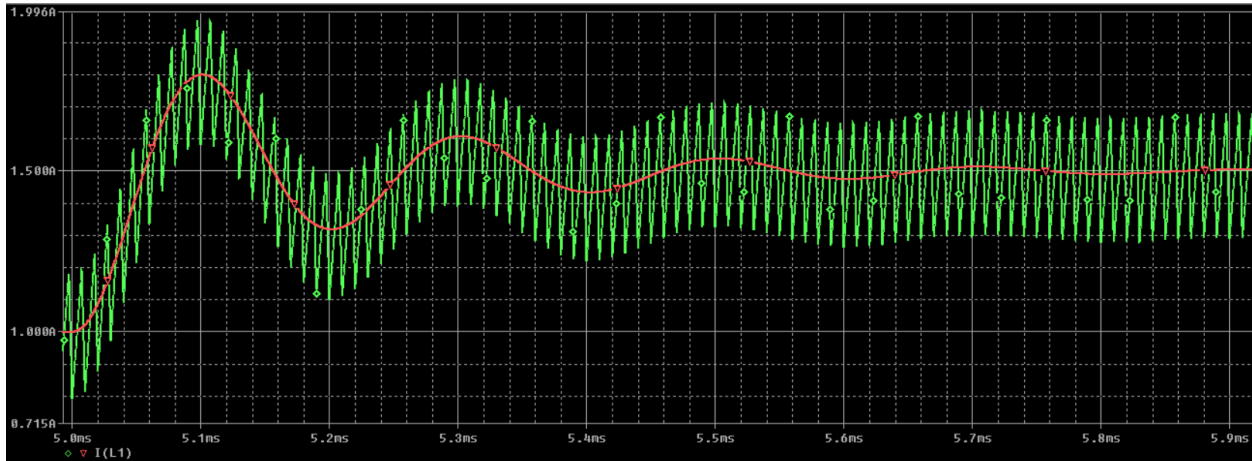
Figure 8: PSpice inductor current waveforms: dynamic average (red) and exact (green)
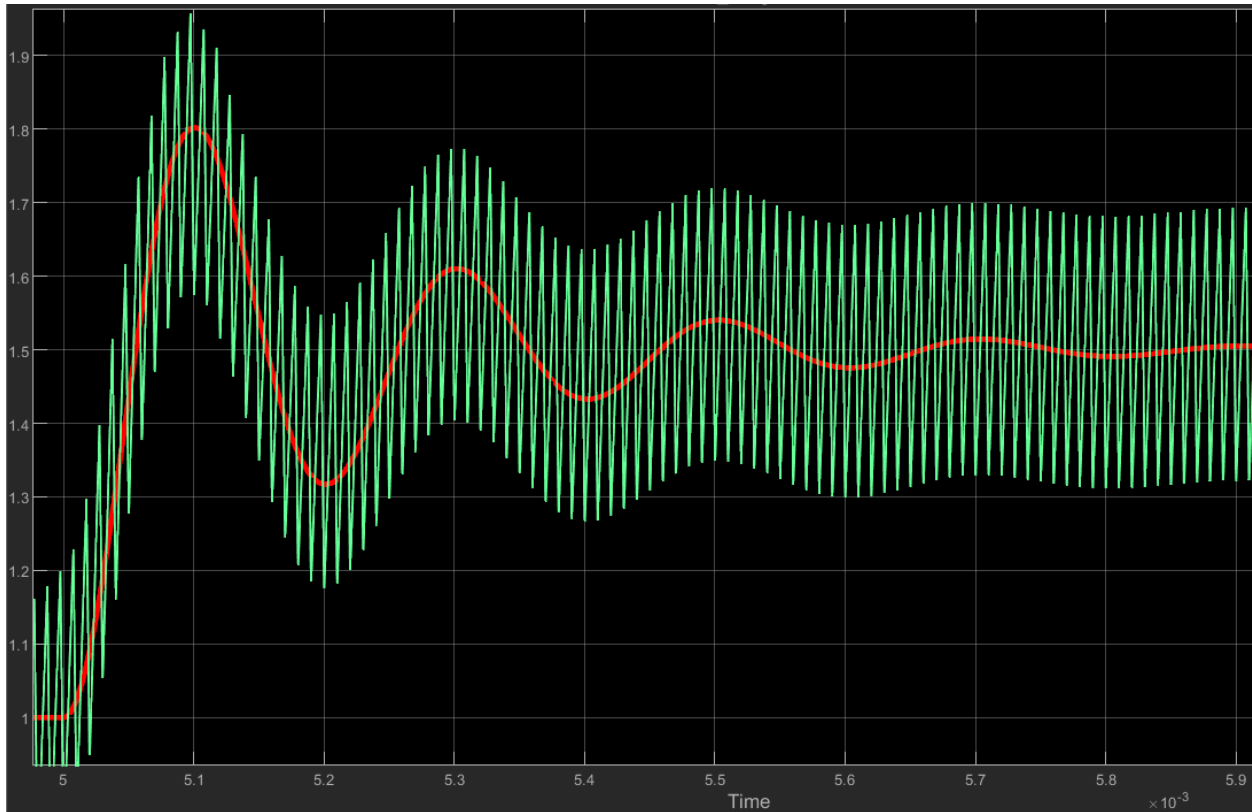

Figure 9: Simulink inductor current waveforms: dynamic average (red) and exact (green)

The final set of simulations involve voltage mode feedback control of a buck converter: Fig. 10 depicts the corresponding block diagram. A frequency domain approach is used for the design, which seeks to achieve zero steady state error to a step input and a phase margin of 60°; the design approach and small signal transfer functions can be found in [7]. The small signal

transfer function of the pulse-width modulator on the power electronics hardware platform is 0.556, and the gain of the feedback path is $k_{FB} = 0.2$. The reference voltage V_{ref} is set to $2V$ to achieve an output voltage $V_o = 10V$. The error signal $e(t)$ drives the controller, whose output $v_c(t)$ is the control voltage that drives the pulse-width modulator. The pulse-width modulator output is the PWM signal of duty cycle $d(t)$ that switches the transistor in the power stage.
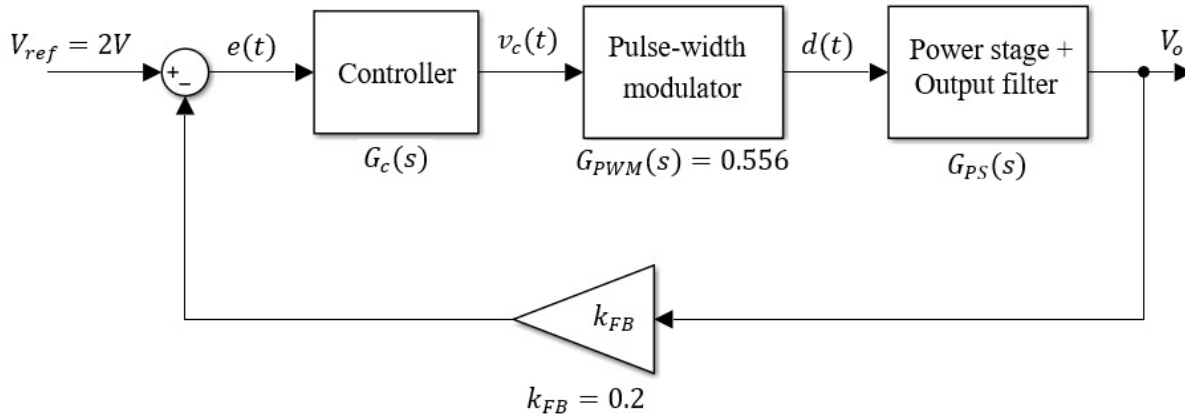

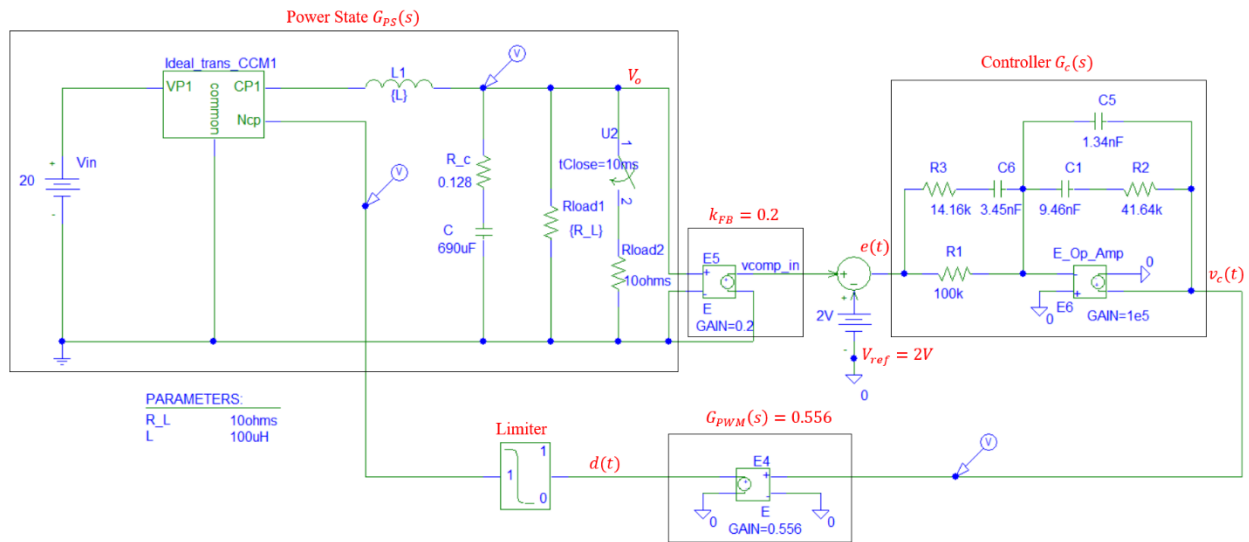Figure 10: Block diagram of the buck converter with feedback control

Figure 11: PSpice simulation of buck converter with feedback control

The PSpice schematic of the dynamic averaged buck converter with feedback control is depicted in Fig. 11. The portions of the schematic that correspond to blocks in Fig. 10 are enclosed in rectangles and labeled in red font. The converter input voltage is $V_{in} = 20V$, and the reference voltage $V_{ref} = 2V$ causes the converter output V_o to be 10V. The converter includes a switched load (10Ω that turns on at $t = 10ms$,) causing the load resistance to drop from 10Ω to 5Ω when

the switched load is turned on. The reduction in load resistance causes the output voltage to dip from $10V$ at $t = 10ms$. The controller $G_c(s)$ is implemented using a high bandwidth operational amplifier on the hardware platform; the op-amp is implemented via a voltage-controlled-voltage- source with gain 10^5 in the simulation. The op-amp is in an inverting amplifier configuration and causes phase inversion. This phase inversion is offset by reversing the signs on the summing block (the reference voltage of $2V$ connects to the $+$ terminal of the summing block rather than the $-$ terminal.) The controller output voltage $v_c(t)$ passes through the pulse-width modulator and the resulting duty cycle signal $d(t)$, limited to lie in the range $0 < d(t) < 1$, controls the turns ratio of the ideal transformer in the dynamic average model of the converter.

Fig. 12 depicts the Simscape schematic of the buck converter with feedback control. The 0.128Ω resistor models the total ESR of the capacitor (there is a 0.1Ω resistor in series with the capacitor on the hardware platform to facilitate probing of the capacitor current.) This schematic mirrors the PSpice schematic except for the fact that the controller $G_c(s)$ is implemented using a "transfer function" block. No phase inversion occurs (as there is no op-amp) and the signs on the summing block match those in Fig. 10.
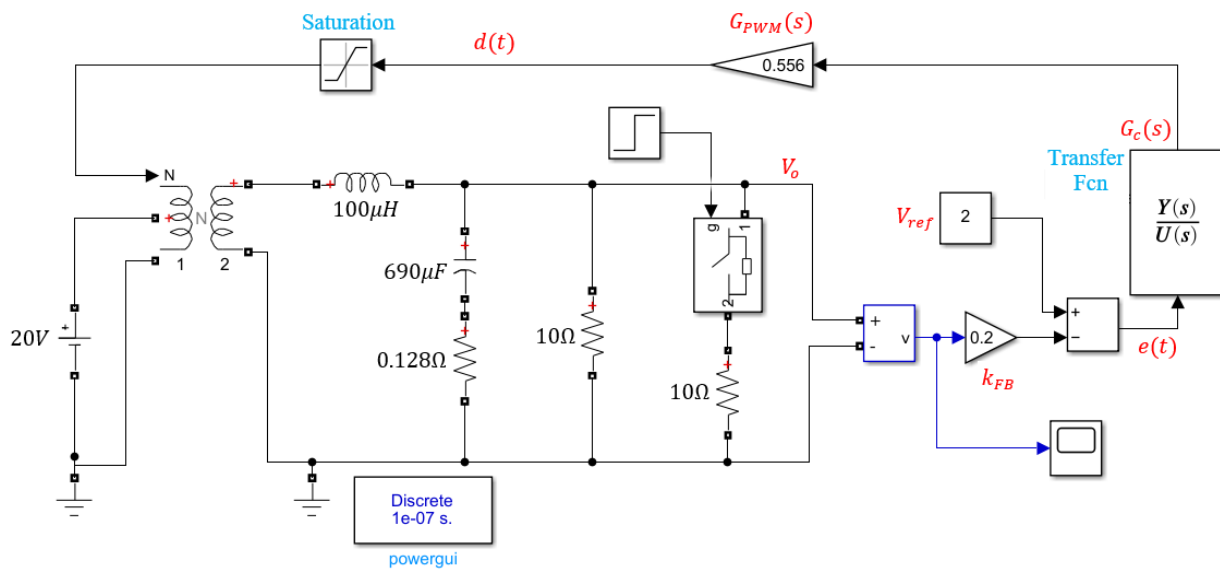

Figure 12: Simscape simulation of buck converter with feedback control

The results from the PSpice and Simulink simulations are depicted in Fig. 13. When the switched load comes in at $t = 10ms$, the converter output voltage drops below 10V. The feedback loop acts and restores the output voltage to $10V$ in about $4ms$. The dynamics of the output voltage in both traces are very similar; cursors show that the voltage drops as low as $9.835V$ in the PSpice simulation, while the Simulink simulation records $9.834V$. Both PSpice and Simulink record the peak overshoot at $10.06V$.

While the paper only presents buck converter simulations, other converters such as boost converters, buck-boost converters and rectifiers are easily simulated using Simulink/Simscape.
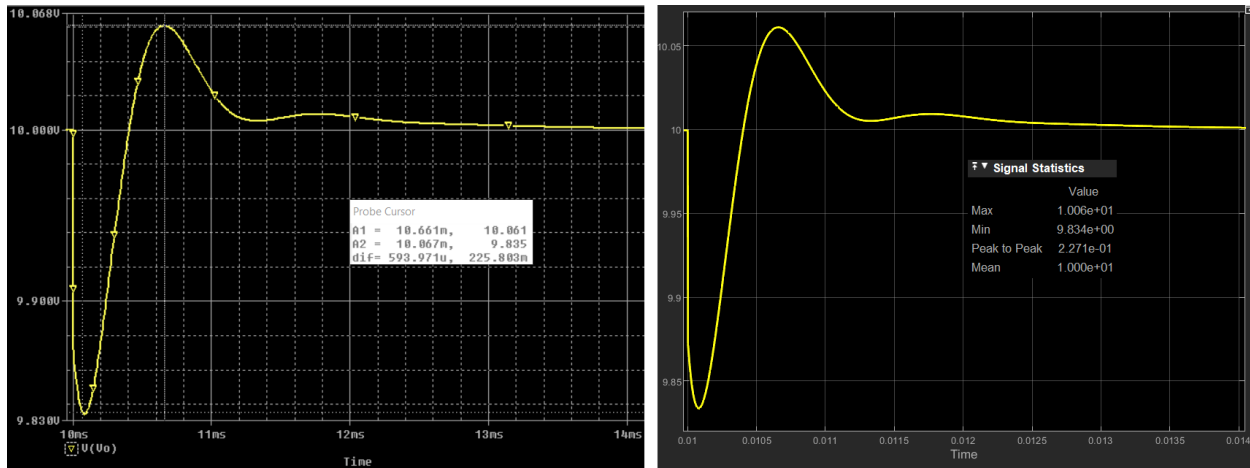
Figure 13: Feedback control simulation results for PSpice (left trace) and Simulink (right trace)

Student and faculty perspectives on the simulation tools

At the end of the semester a student survey was conducted to evaluate student perspectives on the use of PSpice (which they have used in prerequisite classes) and Simulink/Simscape for circuit simulation. The following three survey questions were asked:

Q1. How effective were the Simulink/Simscape simulations at analyzing circuit behavior and understanding circuit operation? Make comparisons against effectiveness of PSpice simulation.
Q2. Compare the ease of use of Simulink/Simscape software for simulation against use of PSpice for simulation.
Q3. Compare the accessibility of Simulink/Simscape software versus PSpice.

All ten students in the class responded to the survey; their responses (percentages of students responding in each category) are listed in Table 1.

Simulink/Simscape is <phrase in row below> than PSpice	Q1	Q2	Q3
Much more effective / easy / accessible	40%	60%	80%
More effective / easy / accessible	50%	20%	20%
About the same	10%		
Less effective / easy / accessible		20%	
Much less effective / easy / accessible			

Table 1: Student response percentages to survey questions

The results for question 1 show that 90% of students believed that Simulink/Simscape was more effective than PSpice at analyzing circuit behavior and understanding circuit operation. For question 2, 80% of students believed that Simulink/Simscape was easier to use than PSpice (60% recorded it as much easier to use.) Results for question 3 show that all students believed that Simulink/Simscape was more easily accessible that PSpice (80% believed it was much easier to access.) Across all three categories, at least 80% of students indicated that Simulink/Simscape

was better or much better than PSpice in terms of effectiveness in analyzing and understanding circuits, ease of use, and ease of access. Written student comments reflect the same sentiments; a few of the student comments are listed below.

- Simscape was much more user friendly. The interface is very clean, and I would say it has a less steep learning curve than spice.
- Matlab is easier to use and debug. More reliable. The main reason I like it more is because I can easily get it on my device. I was never able to get PSPICE on my own laptop.
- Spice is not available on Mac and Matlab is, which makes tasks easier to do at home.
- MATLAB is more accessible than Spice in general. Spice takes much longer to start up and work with because the software feels very heavy. Sometimes Spice doesn't even work due to licensing issues. MATLAB is already used in most other classes, so it is much more convenient.

One student pointed out that the same part could be found in different Simscape libraries but that they were not interchangeable; the comment is listed below.

- I think Spice is easier to use but it didn't have as much pre-coding advantages like Matlab. It is a little confusing about when to use fundamental library or special library in Matlab to add a component. In PSpice, resistor is resistor. It doesn't stop working if they are from 2 different libraries like Matlab does. Either worked well for me in the class.

From a faculty perspective, Simulink/Simscape offered valuable advantages over Cadence PSpice. First, the custom blocks provided by University of Minnesota in PSpice 9.1 were difficult to port to work under Cadence, and even after porting there were issues with student permissions to access libraries, making it difficult for students to run simulations. The features of these custom PSpice blocks are already available in existing Simscape blocks (e.g., variable-ratio transformer) which makes it much easier to run simulations in Simulink/Simscape. Second, the university has a license that allows students to download Matlab on their own laptops. Students therefore have easy access to the software. This was important to students as it freed them from being tethered to campus lab computers and allowed then to run simulations at home, even if they owned a Mac (PSpice does not run on a Mac computer.) Third, it was easy to share simulations with students in the Simulink/Simscape environment. All it took to share a schematic was to upload a single file to the campus online learning platform. With PSpice, each project has its own folder with multiple files that must be copied over. Another advantage of Simscape can be highlighted by comparing the buck converter feedback control simulations of Figs. 11 and 12. To explore the effect of different phase margins on closed-loop dynamic response, the values of the resistors and the capacitors that implement the op-amp based controller must be modified. To do the same task in Matlab, all it takes is to run the script that implements the controller design for a different phase margin. The script automatically updates the controller transfer function (the $G_c(s)$ block in Fig. 12.) Running the Simulink simulation then depicts the effects of the change to the controller transfer function; no parameter values need to be changed in the schematic.

Reflections on the transition from PSpice to Simscape

The power electronics course is an elective that is offered once every two years, and the author is the only instructor for the course at present. The Department Chair and other faculty members knew of the issues related to reliability of PSpice access and this provided sufficient rationale to make the transition from PSpice to Simscape. During the COVID-19 pandemic the university moved to a Matlab license that allowed students to access Matlab on their own computers. Students were used to having access to software at their fingertips; PSpice was the exception in terms of ease of access. Students were very happy to circumvent the PSpice access issues and perform circuit simulations in the Matlab environment from their own laptops. The transition from PSpice to Simscape was challenging due to the author's limited prior experience with Simscape. Every simulation needed to be worked through before it was presented or assigned to students. Thankfully blocks such as the variable-ratio transformer were available, and all circuits were able to be simulated.

Being conversant with industry standard tools such as Cadence PSpice is a student learning outcome for the program. Though Simscape was used in the power electronics course, PSpice is still used in the Circuits and Electronics course sequences. The Department recently made Cadence PSpice accessible to students via Amazon AppStream and this has improved access significantly; even students with Apple Mac computers can access Cadence PSpice in this environment. Given this improvement, some PSpice simulations may be employed in future offerings of Power Electronics in contexts where custom blocks are not required. During the next offering of the course, student perspectives on the PSpice and Matlab/Simscape simulation environments will not be colored by the problems with PSpice access. It will be interesting to see what student perspectives are given that both software suites are equally accessible.

Conclusions

A consortium of 82 universities led by the University of Minnesota (UMN) received a DOE grant to "revitalize electric power engineering education by state-of-the-art laboratories." One component of the curricular efforts was PSpice simulation of power electronic circuits to accompany hardware labs. The UMN-developed simulations were in PSpice 9.1, although they subsequently developed the simulations using PLECS as well. The present work demonstrates how Simulink/Simscape can replace PSpice or PLECS for simulation of power electronic circuits. Our university has a campus-wide license that allows students to install the MathWorks software suite including Simulink/Simscape on their personal computers. The wide accessibility of Matlab/Simulink across universities may make the present work of interest to other universities in the consortium, as it allows for simulation of power electronic circuits without additional cost.

References

[1] DOE award DE-OE0000427: "A nationwide consortium of universities to revitalize electric power engineering education by state-of-the-art laboratories.", University of Minnesota. www.doeconsortium.ece.umn.edu (accessed Feb. 13, 2023).

[2] "Introduction to Power Electronics." University of Minnesota. https://cusp.umn.edu/power-electronics/intro-power-electronics (accessed Feb. 13, 2023)

[3] E. Hossain, "Application of Simulink in Power Electronics," in *MATLAB and Simulink Crash Course for Engineers*, Cham: Springer International Publishing, 2022, pp. 495-568. https://doi.org/10.1007/978-3-030-89762-8_17

[4] N.P.R Iyer, *Power Electronic Converters: Interactive Modelling Using Simulink*, CRC Press, 2018. https://doi.org/10.4324/9781351255752

[5] D.A. Aponte-Roa and M.A. Lau, "Incorporating a Model-Based Approach in an Introductory Electronics Course Using Simscape," in *2018 IEEE Frontiers in Education Conference (FIE), Covington, KY, USA, October 16-19, 2018*, pp. 1-6.

[6] T.J. Hassell, W.W. Weaver, and A.M. Oliveira, "Using Matlab's Simscape Modeling Environment as a Simulation Tool in Power Electronics and Electric Machines Courses," in *2013 IEEE Frontiers in Education Conference (FIE), Oklahoma City, OK, October 23-26, 2013*, pp. 477-483.

[7] N. Mohan, *Power Electronics: A First Course*, Wiley Global Education, 2011