Introduction to DC-to-DC Power Conversion Systems Using QSPICE Simulator

Richard Tymerski

06/02/2025

Lab 1
Introduction to a Basic DC-to-DC Power Conversion System and the QSPICE Circuit Simulator

1.1 Objectives

1. To introduce a basic DC-to-DC power conversion system using the QSPICE simulator.

2. To provide hands-on experience with the QSPICE simulator, focusing on the use of switches (SW), pulse width modulators (PWM), comparators, and the NE555 timer component for sawtooth generation, which are utilized in this and subsequent labs.

Before proceeding with the QSPICE simulator, review the QSPICE documentation available within the tool or online. This documentation provides an overview sufficient for understanding the simulator's interface and functionalities for this and subsequent labs.

1.2 Circuit #1: Buck Converter with Synchronous Switching and Diode Variation

The circuits in this section implement a buck converter topology that steps down a DC voltage source (V1) by switching it ON and OFF to produce a rectangular waveform. This waveform is filtered through an LC network (L1 = 560 µH, C1 = 100 µF) to generate a lower DC voltage across a load resistor (R1 = 25 ohm). The output voltage is monitored at node N02, while the input to the filter is observed at N01. This section includes two sub-circuits: Circuit 1a, which uses synchronous switching with a 40% duty ratio, and Circuit 1b, which modifies Circuit 1a by replacing one switch with a diode.

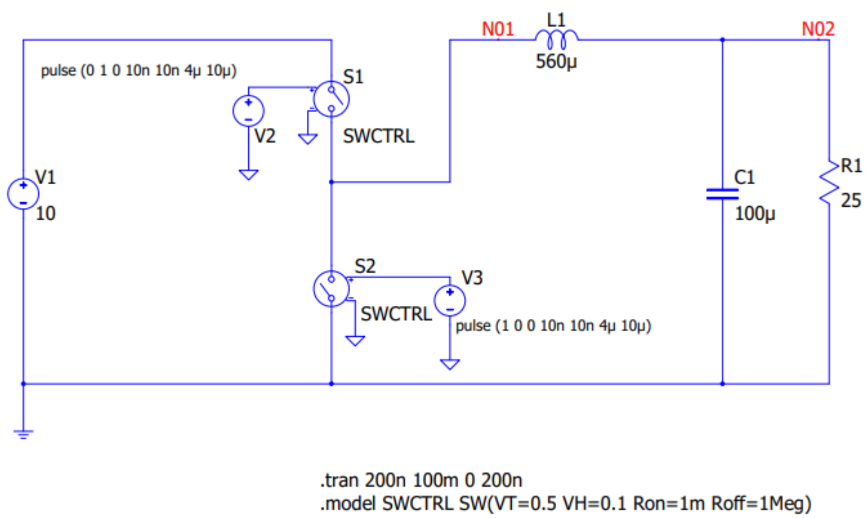
Figure 1.2.1a: Synchronous Buck Converter with 40% Duty Ratio

Components:

- Voltage source (V1): 10 V DC
- Switches (S1 and S2): Controlled by complementary pulse sources
- Inductor (L1): 560 µH
- Capacitor (C1): 100 µF
- Load resistor (R1): 25 ohm

Configuration: S1 and S2 operate synchronously with a pulse signal set to a period of 25 µs (40 kHz switching frequency) and a 40% duty ratio (10 µs ON time).

QSPICE Schematic File:



Circuit #1a: A pulse generator followed by a second order (LC) filter, (L = 560 µH, C = 100 µF and R = 25 ohm) constitutes a dc-to-dc voltage converter.

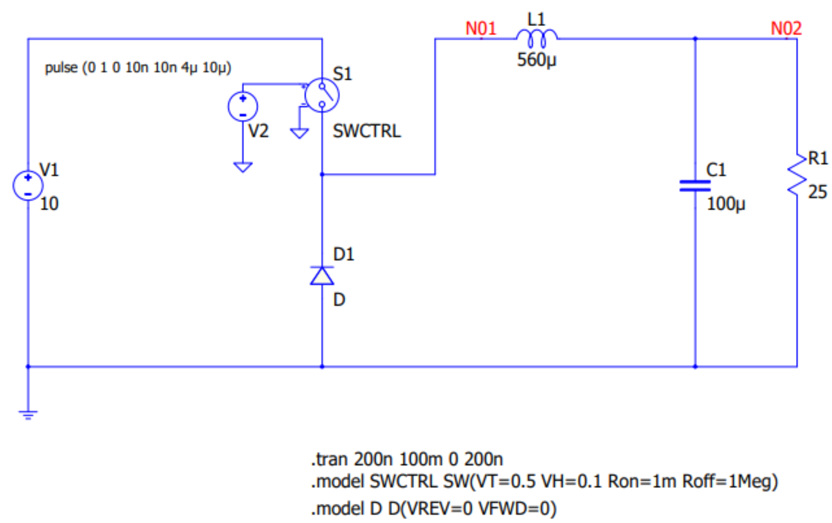
Figure 1.2.1b: Buck Converter with Diode Replacement

Components:

- Voltage source (V1): 10 V DC
- Switch (S1): Controlled by a pulse source
- Diode (D1): Replaces S2
- Inductor (L1): 560 µH
- Capacitor (C1): 100 µF
- Load resistor (R1): 25 ohm

Configuration: S1 is controlled with a 40% duty ratio and a 25 µs period, while D1 conducts during the OFF period, replacing synchronous switching with asynchronous rectification.

QSPICE Schematic File:



Circuit #1b: A pulse generator followed by a second order (LC) filter and a diode (L = 560 µH, C = 100 µF and R = 25 Ω)

1.3 Tasks

1. **Construct Circuit 1a in QSPICE:**
   - Open QSPICE and create a new schematic.
   - Place components from the QSPICE library: V1 (10 V DC), S1 and S2 (switch models), L1 (560 µH), C1 (100 µF), and R1 (25 Ω).
   - Set initial conditions for L1 and C1 to zero
   - Configure S1 with a pulse source (V_PULSE1): PULSE(0 1 0 10n 10n 4u 10u) for 40% duty ratio (4 µs ON, 10 µs period).
   - Configure S2 with a complementary pulse source (V_PULSE2): PULSE(1 0 0 10n 10n 4u 10u) to ensure synchronous operation.
   - Define the switch model: Add .model SWCTRL SW(VT=2.5 VH=0.1 Ron=1m Roff=1Meg) to the schematic text area.
   - Ensure all connections are made at component nodes.

2. **Construct Circuit 1b in QSPICE:**
   - Duplicate the Circuit 1a schematic.
   - Replace S2 with a diode (D1) from the QSPICE library & define .model D1 D(VREV = 0 VFWD = 0)
   - Retain the pulse source for S1 as in Circuit 1a.
3. **Set Simulation Parameters:**
   - For both circuits, configure a transient analysis: Step Size = 200 ns, Start Time = 0, End Time = 100 ms
   - Access this by adding .tran 200n 100m 0 200n to the schematic text area.
   - These settings simulate the circuit for 100 ms, recording points every 200 ns, including switching discontinuities, ensuring smooth waveform plots.
4. **Run the Simulation:**
   - Select Simulation → Run or click the green "Run" triangle in the QSPICE toolbar for both Circuit 1a and Circuit 1b.
5. **Plot Waveforms:**
   - Open the QSPICE waveform viewer post-simulation.
   - Plot the output voltage at N02 for both circuits. Right-click the N02 node in the schematic to view the voltage or select "Plot Voltage".
   - Determine the steady-state value of N02 at the end of the simulation (around 100 ms).
6. **Zoom into Cycles:**
   - In the waveform viewer, use View → Zoom. Left-click and drag to isolate 4-5 cycles near the end of the simulation (e.g., 98 ms to 100 ms). Adjust as needed for clarity.
7. **Add N01 Waveform:**
   - Add the N01 waveform below N02: Go to Waveform → Add Plot, select N01 from the list of available nodes.

8. **Measure Peak-to-Peak Ripple:**
   - Use Plots → Measure in the waveform viewer. Place markers on N02 by left-clicking at maximum and minimum points within the zoomed cycles. Fine-tune to measure the peak-to-peak ripple voltage.
9. **Analyze N01 Waveform:**
   - For VP1, note the peak amplitude (should be 10 V), period (10 µs), and pulse width (4 µs). Calculate the duty ratio: 4 µs / 10 µs = 0.4 (40%).
   - Verify the duty ratio matches the 40% setting.
10. **Explain Steady-State Output:**
    - Explain why the steady-state N02 voltage is approximately 4 V (V1 × Duty Ratio = 10 V × 0.4 = 4 V) for Circuit 1a, considering ideal synchronous switching and LC filtering. For Circuit 1b, discuss any deviations due to the diode's forward voltage drop (e.g., 0.7 V loss).

1.4 Circuit #2: PWM Comparator with Sawtooth Source

The circuit in figure generates a sawtooth voltage waveform across a voltage source V6 (C1, with periodic discharge through a resistor (R1)) controlled by a PWM comparator and an external sawtooth source. The capacitor voltage and current are monitored at nodes `out` and `N2`, respectively. This circuit demonstrates the application of a PWM comparator in waveform generation.

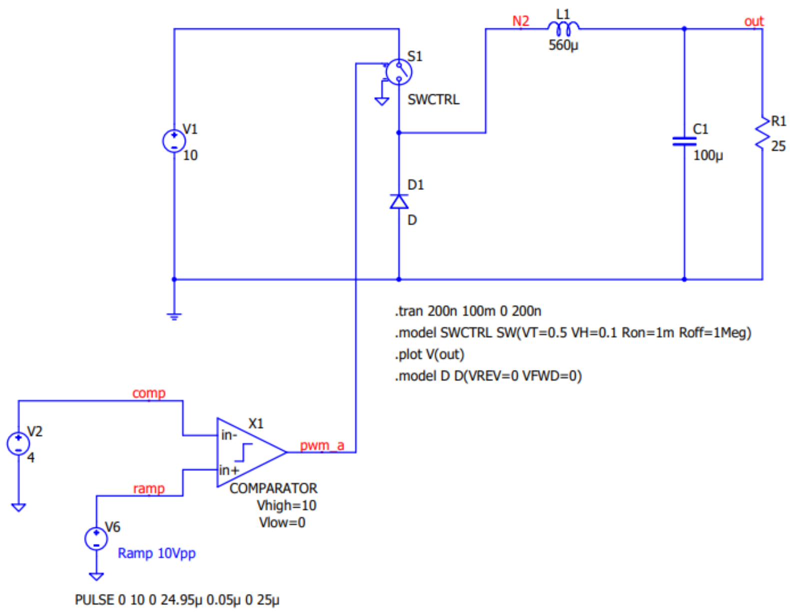
Figure 1.4.1: Sawtooth Generator with PWM Comparator

Components:

- Capacitor (C1): 100 µF
- Resistor (R1): 25 Ω
- PWM Comparator: Configured with a sawtooth source
- Sawtooth Source (V6): Peak = 10 V, Period = 25 µs

Configuration: The PWM comparator compares the capacitor voltage. When it exceeds the sawtooth voltage, the comparator triggers a switch to discharge C1 through R1.

QSPICE Schematic File:



Circuit #2: A comparator which generates PWM with a voltage source for sawtooth input followed by a second order (LC) filter and a diode (L = 560 µH, C = 100 µF and R = 25 Ω).

1.5 Tasks

1. **Construct the Circuit in QSPICE:**
   - Create a new schematic in QSPICE.
   - Place C1 (100 µF), R1 (25 Ω), a switch (S1), and a comparator (e.g., QSPICE's ideal comparator model).
   - Add a sawtooth voltage source (V6): `PWL(0 10 0 24.95u 0.05u 0 25u)` for a 10 V peak and 25 µs period, repeating indefinitely.
   - Connect V6 to the comparator's inverting input (+), V2 to the non-inverting input (–), and the comparator output to S1's control pin.
   - Define the switch model as in Circuit 1a.

2. **Simulation Parameters:**
    - Configure a transient analysis: Step Size = 200n, Start Time = 0, End Time = 100m.
    - Add `.tran 200n 100m 0 200n` to the schematic text.
3. **Run the Simulation:**
    - Execute via Simulation → Run or the green "Run" icon.
4. **Plot Waveforms:**
    - Plot `out` in the waveform viewer.
    - Add `N2` below `out` using Plots → Add plot.
5. **Analyze out Waveform:**
    - Measure the peak amplitude (expected 10 V, depending on comparator threshold) and period (25 µs).
    - Explain how the peak amplitude is determined by C1 (100 µF), and the sawtooth source's peak (10 V).

1.6 Circuit #3: NE555 as Sawtooth Source

The circuit in figure uses an NE555 timer, an inbuilt QSPICE component, to generate a sawtooth waveform that is fed to a comparator instead of using the previously defined voltage source. The output of the comparator controls a buck converter's switch, replacing the external voltage source from previous circuits.

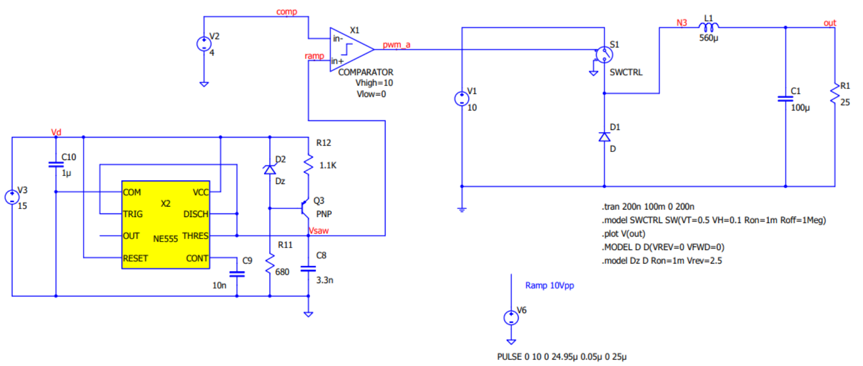
Figure 1.6.1: Buck Converter with NE555 Sawtooth Source

Components:

- Voltage source (V1): 10 V DC
- Switch (S1): Controlled by comparator output
- Inductor (L1): 560 µH
- Capacitor (C1): 100 µF
- Load resistor (R1): 25 Ω
- NE555: Configured for sawtooth generation

Configuration: The NE555 generates a sawtooth waveform with a period of 25 µs (50 kHz). The comparator output controls S1 and voltage is observed across the `out` net.

QSPICE Schematic File:



Circuit #3: A comparator which generates PWM with NE555 as a component for sawtooth input followed by a second order (LC) filter and a diode (L = 560 µH, C = 100 µF and R = 25 Ω)

1.7 Tasks

1. **Construct the Circuit in QSPICE:**
   - Open a new schematic.
   - Place V1 (10 V), S1, L1 (560 µH), C1 (100 µF), and R1 (25 Ω).
   - Add the NE555 component from the QSPICE library. Configure it for a 50 kHz sawtooth output (period = 25 µs) by setting appropriate resistor and capacitor values, referencing the NE555 model documentation in QSPICE.
   - Connect the NE555 output to the comparator input pin. Define the switch model as in Circuit 1a.
   - Set `N3` at the S1-L1 junction and `out` across R1.
2. **Simulation Parameters:**
   - Set End Time = 100 ms, Step Size = 200 ns, Start Time = 0.
   - Add `.tran 200n 100m 0 200n` to the schematic.

3. **Run the Simulation:**
   - Execute via Simulation → Run.
4. **Plot Waveforms:**
   - Plot `out` (output voltage) in the waveform viewer.
   - Add `N3` (input to filter) using Plots → Add plot.
5. **Measure and Analyze:**
   - Zoom into 4–5 cycles at the end (View → Zoom).
   - Measure the steady-state `out` value and peak-to-peak ripple (Plots → Measure).
   - For `N3`, note the period (25 µs) and assess the NE555 waveform's shape (sawtooth characteristics).
6. **Explain NE555 Impact:**
   - Discuss how the NE555 sawtooth source affects the output compared to Circuit 1a's pulse-driven switching, focusing on duty ratio and ripple.

1.8 Tasks

Complete the following table.

	Switching Frequency	**Duty Ratio**	**Peak-to-peak Input Voltage to Filter**	**Steady State Average Output Voltage**	**Peak-to-peak Output Voltage Ripple**
Circuit #1					
Circuit #3					

- Explain the differences seen in the peak-to-peak ripple voltage values between Circuit #3 and Circuit #1. Are they in line with your expectations? Why?

QSPICE Usage Notes:

- Ensure all components are connected at nodes, as QSPICE requires precise node-based connections.
- Make sure to save the schematics for reference.
- Use the waveform viewer's zoom and measurement tools extensively for accurate analysis.