# Exercise 2b PV Panel Maximum Power Point Tracking

#### M. K. Ranjram

#### 1 Introduction

Power electronic converters are fundamental to photovoltaic (PV) systems. In this exercise, we will explore the use of the dc/dc "boost" converter that was introduced in the lectures to implement maximum power point tracking (MPPT) of the PV panel we studied in Exercise 2a.

The larger PV system we are interested in studying in this and future exercises is shown in Fig. 1. This system shows a string of PV panels<sup>1</sup> connected to a 480Vac (line-line, rms), 60Hz grid connection through a dc/dc boost converter and another power electronic block, called an "inverter", that converts the dc at its input to an ac voltage. The peak line-neutral value of the grid voltage is 391.9V, and we must provide a dc voltage larger than this value at the input of the inverter. We'll see in the next exercise that providing 800V to this "dc bus" is sufficient.

Note that each of the panels we considered in Exercise 1 has a nominal open-circuit voltage  $(V_{oc,nom})$  of 40.9V and MPP voltage  $(V_{mpp,nom})$  of 33.7V. This means we'd need to increase the voltage of a single panel by approximately 12 times to achieve the desired dc bus voltage. This is a relatively large "step-up ratio" for our boost converter, so we'll instead use a string of 8 panels connected to the boost converter. This increases the effective open-circuit voltage to 327.2V and the effective nominal MPP voltage to 269.6V, significantly reducing the conversion burden of our boost converter.

Note that this system is intentionally simplified to assist with our learning. In practice, the system of Fig. 1 includes *isolation*, implemented by using a dc/dc converter that employs a transformer (called an "isolated" dc/dc converter), or by interfacing the inverter to the grid via a step-up transformer. In either case, the step-up capability of our system is improved and we do not necessarily have to rely on connecting panels in series to achieve higher voltages. It is also common to "split up" a single long string into smaller strings which are then connected in series through their own converter stages. This can improve tracking of the global maximum power point in cases where the solar panels will experience very different shading conditions, such as on a residential rooftop. On the other hand, in solar PV power plants it is more common to employ the longest PV strings that can be handled by the power converters since these plants can be intentionally designed to minimize shading differences on the panels in a given string. The skills and knowledge developed in this exercise provide the foundation to study these systems.

<sup>&</sup>lt;sup>1</sup>A "string" of panels means that the panels are connected in series.



Figure 1: PV system studied in Exercises 2 and 3. The boost converter achieves MPPT of the PV strings, while the inverter (also called a "voltage-sourced converter" (VSC) or "voltage source inverter" (VSI)) delivers this power to the grid and regulates the dc bus. This exercise focuses on the boost converter and its MPPT operation.

In this exercise, we'll focus on understanding the boost converter's key operational characteristics, exploring the difference between the switching and averaged models discussed in the lecture, and then implementing this this conversion block to achieve MPPT.

This exercise also serves as an introduction to LTspice [1], an excellent free circuit simulator which works well for simulating switching power electronics, and is widely used for this  $purpose^2$ .

## 2 Implementing a Switching Boost Converter in LTspice

For your convenience and reference, an example boost converter implemented in LTspice accompanies this exercise. Review this file and answer the following questions:

- 1. The simulation employs an ideal switch model. Confirm the ideal nature of this model by writing down the on-resistance, off-resistance, and switch threshold voltage used in the simulation. For your reference, try to determine how these parameters can be changed, and also determine how the switch's ".model" statement is connected to S3 in the model. *Hint: Control+right-click on the switch*.
- 2. The diode is also ideal. Confirm the ideal nature of this model by writing down the on-resistance, off-resistance, and forward voltage drop used in the simulation. For your reference, try to determine how these parameters can be changed, and also determine how the diode's ".model" statement is connected to D1 in the model. *Hint:* Control+right-click on the diode.
- 3. The pulse-width-modulation command applied to the switch is specified by the "PULSE" voltage source. What are the on-voltage, duty cycle, and period of this source?

 $<sup>^{2}</sup>$ A disadvantage of LTspice is that it lacks the ability to specify algorithms in "code" and this can make it complicated to implement circuit control – it is certainly possible, but it generally requires using circuit elements to implement the desired behavior. We'll see in future exercises that other software tools are more effective if our interest is in studying system-level behavior.

- 4. What are the switching frequency, input voltage, output voltage, and output power specified in the simulation file? How can you change these parameters? *Hint: Right-click on the .param statement.*
- 5. What is the origin of the values for the input voltage and output power in the simulation? *Hint: What is the input to our PV system?*
- 6. How are the duty cycle, switching period, and output resistance specified in the simulation? What is special about how these computations are specified?
- 7. The line that begins with ".ic" specifies an initial condition for the capacitor C1. Why might we set the simulation up this way? What happens if you change the initial condition to 0? Note: you can run the simulation by clicking Simulation  $\rightarrow Run$ .
- 8. When designing a boost converter, choosing the values of the passive components L1 and C1 is critical. These components are typically among the largest and least efficient in a real converter, and can dominate the overall volume and loss. The detailed design of this converter is outside the scope of our exercise, but it is useful to understand what their impact is. Run the simulation (use the default values of the file you can run by clicking Simulation→Run). Plot the current through L1 (by clicking on L1) and zoom in to the end of the waveform so that you can see about 10 cycles. What does the current look like?

As you decrease the inductance value, you'll see that this "ripple" gets larger. Ctrl+Right-click on ".param Lboost = 150u" and replace the this with ".step param Lboost 100u 200u 50u". Run the simulation. It will run the simulation 3 times, first with Lboost = 100uH, then 150uH, and finally 200uH. How much larger is the peak-to-peak amplitude of this current when Lboost is 100uH instead of 150uH?

If the value of Lboost is too small, the operation of the converter is fundamentally changed. This is associated with the current in the inductor trying to become negative (which the diode D1 does not allow) and is called "discontinuous conduction mode" operation. Ctrl+Right-click on ".step param Lboost 100u 200u 50u" and replace the phase with ".param Lboost = 25u" to examine this kind of operation. Run the simulation. Probe the output voltage by touching Vo. Is the output voltage still 800V?

### 3 Implementing an Averaged Boost Converter in LT-SPICE

We have seen that the details of the switching waveforms are critical for understanding the boost converter's operation. However, in system-level studies, we often do not need to simulate a converter at this level of detail. One challenge with simulating the switching waveforms is that it can be slow if we need to study long timescales in our system.

Fortunately, if the converter is well-designed and we understand its operating limits, then at the system-level it is often sufficient to evaluate how the *averages* of the currents and voltages of the converter evolve over time. For example, in the system of Fig. 1, it is most important that the output voltage is dc at 800V, and less important that there is some small voltage ripple at the switching frequency. We'll explore this idea in this section. For your convenience, a simulation file of both the switched and averaged model of the boost converter accompany this exercise. This simulation includes steps in the output current, replicating steps between half-power and full power, so that we can explore the dynamic behaviour of these models.

- 1. Run this simulation file and plot both the switching output voltage and the averagemodel output voltage. How do these waveforms compare?
- 2. Now plot the current in L1 (switching model) and L2 (averaged model). How do these waveforms compare?
- 3. If we are most interested in how the currents and voltages of the converter evolve over time, is the averaged model a reasonable way to explore this?

## 4 Implementing the PV panel in LTSPICE

We can now connect our averaged boost converter model to our PV circuit model. For your convenience, a simulation file of this system in LTSPICE accompanies this exercise. Some notes:

- 1. The PV circuit model parameters are directly obtained from the modeling we did in part 1 when the temperature is fixed at 25 degrees. Studying just a single temperature makes the implementation in LTSPICE much simpler, since the diode circuit model is defined at a specific temperature.
- 2. We account for the number of series cells simply by setting the parameter 'N' of the diode to be the ideality constant we computed for our PV panel multiplied by the number of series cells.
- 3. We account for our 8-panel string by simply repeating the circuit model 8 times and connecting them in series.
- 4. There is a protection diode in series with the string which ensures that current is never fed *into* the PV panels.
- 5. The simulation includes a realistic 24 hour irradiance profile. In the simulation, 1 second corresponds to 24 hours. Note that, in practice, the array is also warming up as the irradiance goes up, and to be precise we might also want to include the temperature variation in a full study. As mentioned previously, this is difficult to do in LTSPICE. However, the simulation can still give us a good idea on how our system generally behaves.

To achieve maximum power point tracking, the current and voltage of the array is sampled every minute and the boost converter adjusts its duty cycle to try and push the system to its maximum power point. This kind of control is called "perturb and observe". Every sample, the boost converter's controller checks to see if the previous action it did caused the power produced by the panel to increase or decrease. It then decides if it should repeat the last action it took, or if it should do the opposite in order to move to maximum power. Let's learn more about this MPPT algorithm:

- 1. The simulation file has a parameter 'MPPTON' which determines whether or not the boost converter control is enabled. Set this value to 0 this causes the boost converter to operate with a fixed duty ratio (selected to be 0.75). What is the voltage imposed on the PV string? What is the energy extracted from the solar panel? Since we are only after a relative understanding, you can just extract the energy reported directly from the simulation, even though the units are incorrect (hours instead of seconds). *Hint: Alt+click on B1 to measure the power it processes, and then Ctrl+left click on the signal name to see the average of the waveform.*
- 2. Now enable the control by setting 'MPPTON' equal to one. Has the energy extracted from the PV panel changed? By how much?
- 3. Explore the implementation of the MPPT algorithm in LTSPICE and write out the control algorithm using a control diagram or a flow chart.
- 4. Continue to explore this system try changing the irradiance seen by one of the modules, or changing what the default duty cycle is when MPPT is off.

Note that the boost converter is actually switching at 100kHz while the irradiation pattern varies over 24 hours. If we attempted to study this system in simulation using the switched model of the boost, the simulation time would be extremely long owing to these very different time periods. By reducing the boost to its averaged model, we maintain its representative operation and greatly simplify the study of this system.

#### References

[1] Analog Devices Inc., "Ltspice," 2022. [Online]. Available: https://www.analog.com/en/ design-center/design-tools-and-calculators/ltspice-simulator.html