SIMULATION WITH THE SEPIC TOPOLOGY
(SEPIC-Single Ended Primary Inductance Converter)
ECE562: Power Electronics I
COLORADO STATE UNIVERSITY

Modified in Fall 2011
PURPOSE: The purpose of this lab is to simulate the SEPIC converter using NL5 to better familiarize the student with some of its operating characteristics. This lab will explore some of the following aspects of the SEPIC converter:

- Time it takes for the converter to reach steady state
- Effects of varying duty cycle
- Conversion ratio of converter
- Differential current across the inductor
- Continuous versus Discontinuous conduction mode
- Effects of changing load resistance
- Output ripple voltage
- Equivalent Series Resistance (ESR) of the output capacitor
- Efficiency of the converter

NOTE: The simulations that follow are intended to be completed with NL5. It is assumed that the student has a fundamental understanding of the operation of NL5. NL5 provides tutorials for users that are not experienced with its functions.


Figure 1 below shows the SEPIC converter implemented with ideal switches. Unlike the Cuk converter which inverts the input to output polarity, the SEPIC converter maintains the same polarity from input to output. Just a couple years ago, the patent covering the SEPIC converter expired, so its popularity for use in designs is on the rise. The drawback to this converter is the additional pole created with additional LC network which makes the control loop more complicated than a simple buck, boost or buck/boost topology. [See additional reading at end of this lab]
The converter shown below is the ‘Inverse’ SEPIC converter. As also is the case for the SEPIC converter, this design also does not invert the input to output polarity. It also has the same (buck/boost) \( M(D) = \frac{D}{1-D} \) conversion ratio as the SEPIC converter. Notice that if you visually rotate the SEPIC converter around the Y axis, it looks almost exactly like the ‘Inverse’ SEPIC converter below (with the addition/subtraction of an input/output capacitor). This interesting feature allows power to be transferred in one direction when run as a SEPIC converter and then allows power to flow in the opposite direction when run as an ‘inverse’ SEPIC converter.

![Figure 2 - 'Inverse' SEPIC converter shown with ideal switches](image)

Below is a plot of the conversion ratio \( M(D) = \frac{D}{1-D} \) for the SEPIC converter.

![Figure 3 - \( M(D) = \frac{D}{1-D} \)](image)

The schematic below shows the SEPIC converter with actual components substituted for the ideal switches and the resistance of non-ideal inductors added.
Now implement the schematic shown above in Figure 4 in NL5. It should look like Figure 5 below when complete.

**\( V_g \)** is a DC voltage source (VDC) from the source library. It needs to be set for 120 volts.

**\( L_1 \)** is an ideal inductor from the library. Set to 500 µH.

**\( R_{L1} \)** is an ideal resistor from the library simulating the resistance of \( L_1 \). Set to 100 mΩ.

**\( C_1 \)** is an ideal capacitor from the library. Set to 47 µF.

**\( L_2 \)** is an ideal inductor from the library. Set to 100 µH.

**\( R_{L2} \)** is an ideal resistor from the library simulating the resistance of \( L_1 \). Set to 20 mΩ.

**\( D_1 \)** is an ideal diode from the library. Set to 700 mV (diode drop).

**\( C_2 \)** is an ideal capacitor from the library. Set to 200 µF.

**\( O_1 \)** is an ideal comparator used to turn the switch \( S_1 \) on and off. By varying the width of \( V_3 \) below, its output will act as a Pulse Width Modulator.

**\( S_1 \)** is a voltage controlled switch, a standard component in the library.

**\( V_2 \)** is 0.5 volt reference for the Schmitt trigger comparator \( O_1 \). Set \( V_2 \) to 500 mV.

**\( V_3 \)** is Pulsing source. Set to values listed below using the components editing window. This sets it to a switching frequency of 100 kHz with a 50% duty cycle.

<table>
<thead>
<tr>
<th>( V_3 )</th>
<th>Pulse</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \text{V} )</td>
<td>1</td>
</tr>
<tr>
<td>( \text{V0} )</td>
<td>0</td>
</tr>
<tr>
<td>Period</td>
<td>10e-6</td>
</tr>
<tr>
<td>Width</td>
<td>50e-6</td>
</tr>
<tr>
<td>Slope</td>
<td>Linear</td>
</tr>
<tr>
<td>Rise</td>
<td>10e-9</td>
</tr>
<tr>
<td>Fall</td>
<td>10e-9</td>
</tr>
<tr>
<td>Delay</td>
<td>0</td>
</tr>
</tbody>
</table>
**HINT:** Press F3 (Schematics->components pulldown) and adjust the names and values as needed. To adjust a name, select the component name and then select the upper left [a|b] symbol. To select a value, select the component on the left column of components and then type a new value in the right side component value field.

![Component Adjustment Window](image)

**Figure 5 - Window (Press F3) to adjust name and values of components**

**HINT:** To rotate left or right, mirror or flip components use shortcuts below.

To change orientation of a schematic element, use Rotate, Mirror, and Flip commands:

- Rotate right (Ctrl-R). Rotate right (clockwise).
- Rotate left (Ctrl-L). Rotate left (counterclockwise).
- Mirror (Ctrl-M). Mirror (flip around vertical axis).
- Flip (Ctrl-F). Flip (flip around horizontal axis).

Set the transient simulation parameters (Transient->Settings pulldown) to those shown below:
The schematic below shows the NL5 schematic for the SEPIC converter.

![Schematic of SEPIC NL5 converter](image)

**Figure 6- SEPIC NL5 schematic**
Now select pull down menu item Transient-> Start (Hotkey F6) to start the simulation.
The output should look like the display below.

![Figure 7 - Simulation result from SEPIC converter](image)

**QUESTION 1:** How long does it take the converter to reach steady state? What is the output voltage of the converter at steady state? Verify your results mathematically.

First select (highlight) V3 on the schematic view to allow it’s the V3 parameters to be swept. Now, select from the pull down menu item Tool->Script and set up to run the pulse width from 0.5uS to 9.5uS to see the effects of varying duty cycle on the converter. For future sweeps of a components values, remember to always to first select the component on the schematic and then its name will show up on the sweep list.

![Figure 8 - Script settings to change duty cycle](image)

Now press the blue arrow on the Tools-Script screen to start the simulations. The results should look like the window below.
**QUESTION 2**: How does the output voltage of the converter vary with increasing switch duty cycle? Does it follow $M(D) = D/(1-D)$? How does increasing the duty cycle affect the settling time of the converter?

Now clear all the data traces from the Transient Display window.
**Hint:** To clear all the traces, select Transient->Data pull down menu, select the ‘storage’ tab and click on the black middle left ‘X’ to delete all data traces from the display.

![Clearing data traces](image)

Figure 10 – Select Transient ->Data then Storage tab to get screen to clear the data

Now look at the current waveform of L1 by applying a current marker.

![Current waveform of L1](image)

Figure 11 - Current measurement of L1

**QUESTION 3:** What is the peak operating current now? What is the operating mode of the converter (remember that you can observe this by zooming in and determining if the sign of the current reverses)? Also, verify the mode mathematically.
Now look at the current waveform of L2 by applying a current marker.

Figure 12 - Current waveform of L2

**QUESTION 4:** Does the current waveform of L2 show a continuous or discontinuous mode of operation?

Now place a current marker on both L1 and L2 and swept the Rload from 10 to 100 ohms using the script feature.

Figure 13 - L1 and L2 currents with Rload changes from 10 to 100 ohms in 10 ohm steps.
**QUESTION 5:** Does the converter change modes of operation when the load resistance changes? If so, what is the approximate load resistance where the converter changes from one mode to the other (continuous versus discontinuous) mode?

Now clear all the data and set $R_{load}$ back to 50 ohms. Place a voltage marker on the output and run the transient simulation.

![Figure 14 - Measurement of voltage ripple](image)

**QUESTION 6:** What is the output ripple voltage of the converter?

Now insert a resistor in series with $C_2$ to simulate the capacitor’s Equivalent Series Resistance. Set the ESR resistor’s initial value to 1 ohm.

![Figure 15 - SEPIC converter modified to simulate the output capacitor's ESR effects](image)
QUESTION 7: What is the effect of increasing output capacitor ESR on output voltage? Does the output ripple voltage also increase as the capacitor’s ESR goes up?

Now set the C2_ESR to 1 milliohms to minimize the ESR of the C2 capacitor and add two wattmeters to measure the input power from the source and the power delivered into the load resistor. (Hint: Wattmeter is under the ‘W’ component list)

The watt meter can measure both current and voltage. The power can be derived using by multiply current times voltage as a trace function. (IE: V(W1) * I(W1))
The schematic below shows what the NL5 schematic should look like with the addition of two wattmeters and the ESR of C2 set to a minimum value for filtering.

Now add the ‘Function’ traces to measure the power of W1 and W2
Now run the simulation.

![Figure 21 – NL5 Simulation showing the input and output power of converter](image)

Right click on the trace screen and ‘Zoom In’ vertically and horizontally where the converter has reached steady state so that the ‘rms’ values of the power measurements can be obtained to calculate the converter’s efficiency.
QUESTION 8: What is the converter’s efficiency at this operating point?

QUESTION 9: What happens to the converter’s efficiency if the duty cycle is decreased to only 10% on time?

QUESTION 10: What happens to the input source when the duty is reduced?

QUESTION 11: If the duty cycle is 50% and the load decreases to 10 ohms what happens to the converter’s efficiency?

Additional Reading