

**CECS 211 - LAB 7**  
**LTSpice Intro and Inductor in DC**

NAME:

POSSIBLE POINTS: 10

STUDENT ID:

COURSE SECTION:

DIRECTIONS:

Using LTSpice, Simulate an Inductor with:

- 10V DC Voltage Source (We will pulse it on and off to see the effect on current)
- 10ohm Resistor in series with the inductor
- 1H Inductor

For the DC Voltage Source use a voltage source and change the function to PULSE. We want 1 second before the voltage goes high (10v) and then wait 1 second before the voltage goes back to low (0v). When adding the Inductor, make sure to right-click on the component and change the Series Resistance to 0. This gives us an Ideal Inductor with no unwanted resistance. Set up the simulation command to stop exactly after 3 seconds.

On the Waveform, show:

- Show the voltage applied to the circuit
- Show the current through the inductor
- Once you have a printout of the waveform, draw in an approximation of the induced voltage (Back-EMF)

Calculate the following:

Max Current through the inductor after fully storing energy in the magnetic field:

L/R Time Constant:

How many Time constants until the Max I is reached:

Time from when the pulse goes high until the Max I is reached:

Energy stored in the Inductor at Max I:

L/R Time Constant	% of Max I	Current at each L/R	Measured Current at each L/R
1			
2			
3			
4			
5			