Software Report #3
Ayhan Okuyan
21601531
EEE202-03

Software Report 3

Introduction

In this laboratory assignment we are supposed to design and obtain high voltage spikes by using 10Vp-p square wave voltage as source and 50Ω of source resistance. The frequency is expected to be less than 5MHz. Throughout this report I will be showing the necessary design steps and calculations. Then we will put our design theory to test and simulate it by using LTSpice and discuss the outcomes of the simulations. The desired output voltage conditions are given below.

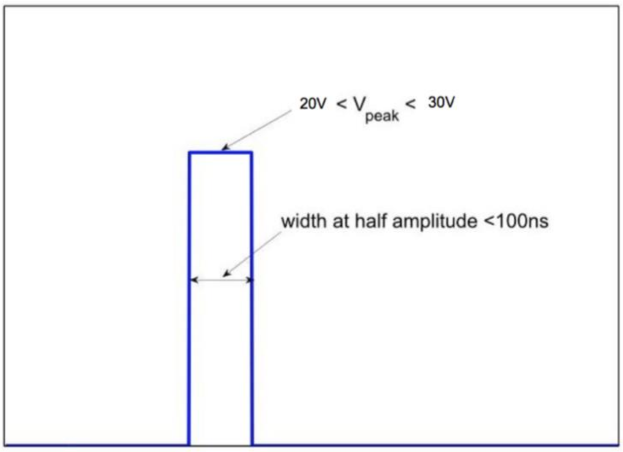

Figure 1 – The Desired Output

Easily observed from the graph, what we need is to be able to obtain a spike voltage that is between 20V and 30V. To add, the width at half amplitude must be smaller than 100ns. One important matter given in the prerequisites, all of the circuit elements must be passive devices, which means that we can only use resistances, inductors or capacitors and independent voltage sources.

In order to remain in these boundaries, my design consists of a voltage source, two inductors, two resistors and a capacitor. The inductors will be used to create a transformer that will enhance the given 10V to the voltage range that we want it to.

Considering the losses that will happen, we will calculate the windings so that the voltage is tripled from 10V to 30V. In order to minimize the width that should be smaller than 100ns, we will be using a load resistor combined with a capacitor in order to make an RC circuit.

Design

In order to design a valid transformer, we need to understand how one functions. Below, there is a transformer schematic that exemplifies how it is built.

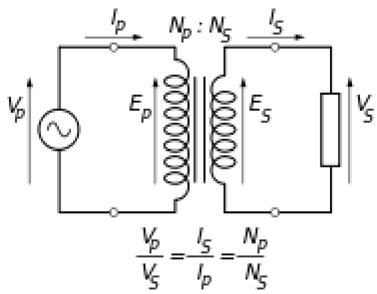

Figure 2 – Transformer

Transformers are devices that use electromagnetic induction to transfer electrical energy. The convenient model of a transformer uses two inductors in parallel with different winding values. Primary inductor is connected to the source while the secondary inductor is connected to the load. The voltage ratio as seen from the Figure 2 is equal to the proportion of the primary winding to the secondary winding. The ratios that will be used, which is based on Fraday's Law of Induction can be found below.

$$\frac{V_P}{V_S} = \frac{I_S}{I_P} = \frac{N_P}{N_S} = K$$

Combining this with the relation between the inductance and winding values

$$L = A_L \, x \, n^2$$

we can easily say that

$$\frac{L_P}{L_S} = \left(\frac{N_P}{N_S}\right)^2 = K^2$$

Hence, the final equation becomes

$$\frac{L_P}{L_S} = \left(\frac{V_P}{V_S}\right)^2 = K^2$$

As we have obtained the necessary equation that we will be needing to create the desired transformer, we can begin with the calculations. The input voltage given by the instructor is 10V which is equal to V_P. The desired output voltage is chosen as 30V. Then the $\frac{V_P}{V_S}$ value becomes 1/3. Which means that $\frac{L_P}{L_S}$ becomes 1/9. This means that we can choose the inductance values as we want, however, we need the ratio to be 1/9 at all times. Since, the core that will be used for primary and secondary inductors is the same, the A_L value does not change. For this reason I have concluded that the primary inductor to be 0.1µH and respectively the secondary inductor to be 0.9µH. The LTSpice drawing of the transformer that I have chosen to use is given below combined with the source and the given source resistor.

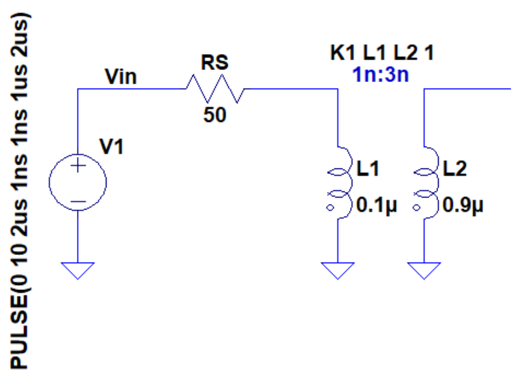

Figure 3 – The Transformer Chosen to be Used

Then, the next step is to configure the RC circuit that will operate on the secondary side. The secondary part will be used in order to minimize the width as much as possible (definitely smaller than 100ns) and ensure that the voltage is between 20V and 30V. Furthermore, as can be seen from the graph, I have deicded to use 500kHz frequency in the design.

The next thing to do is to calculate the load resistance value through the relationship between the source and load resistances and the inductance values. The equation can be found below.

$$\left(\frac{L_S}{L_P}\right)^2 = \frac{R_L}{R_S}$$

Since the left-hand side of the equation is equal to 9^2=81 and the R_S is 50Ω, we multiply these in order to find the load resistance.

$$R_L = 81 * 50 = 4050\Omega \approx 3.9k\Omega$$

We will be using 3.9kΩ for the load resistance as it is close to the theoretical 4050Ω and is also a standard value that will easily be used in the hardware segment of the lab assignment. As we do know the appropriate resistance value, we can calculate the capacitance value that will be needed easily through the RC constant.

To find the time constant, we first need to understand the voltage equation for an RC circuit which is given below.

$$V(t) = V_{in} \,.\, e^{-t/RC} = V_{in} \,.\, e^{-t/\tau}$$

For these types of exponential equations, it is assumed that when t=5τ, V(t=5τ) is considered to be so small that it equals zero entirely. Therefore, there is this time relation between the equation and the 100ns that is mentioned.

$$5RC \leq 100 \,.\, 10^{-9}s$$

$$RC \ \leq 20\,.\,10^{-9}s$$

As we now know the range of the time constant, and the resistance value, we can find the capacitance value easily. Furthermore, as I have chosen the inductance values as small as I could, there is no way that the secondary inductance gets in the way of our RC calculations. At the end, if we divide the RC_{max} to the known 3.9kΩ, the capacitor turns out to be

$$C \ \leq 5.128 * 10^{-9}F \ \approx 5pF$$

As we have reached each and every value that needs to be get, we can finalize the circuit drawing and simulate our design on LTSpice and measure the voltage and time difference values. The final circuit design is given below.

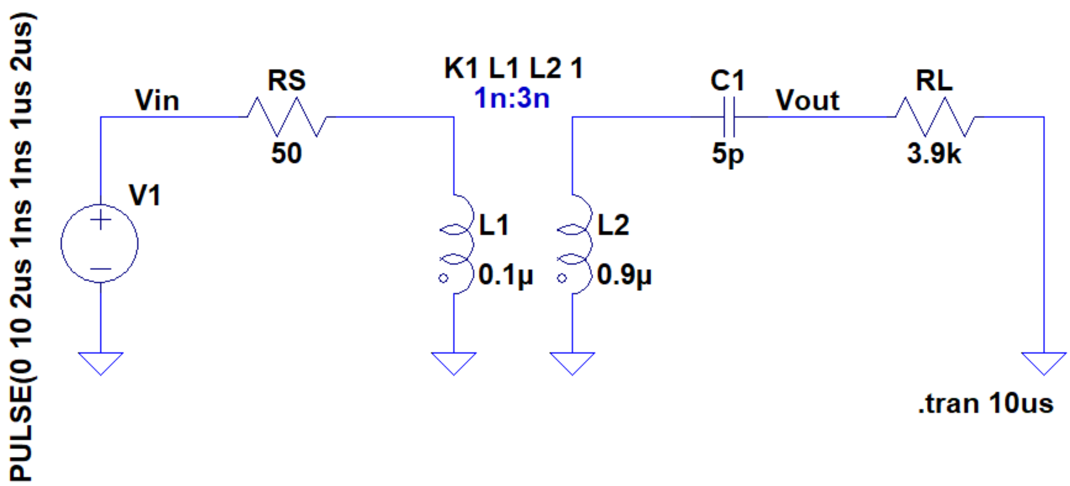

Figure 4 – Schematic of the Designed Circuit

The final circuit can be seen below. In order to create the transformer, the SPICE directive "K1 L1 L2 1" is used. K1 is the constant that we used in the equations and L1, L2 are the names of the inductors, and finally, "1" is the mutually coupling coefficient. We have chosen it to be 1 as we do not wish to see leakage throughout the simulation. The PULSE signal at the left shows the given desired input, which is a

10Vp-p square wave with 2ms of delay which has a frequency of 500kHz. The graph of the input is described below.

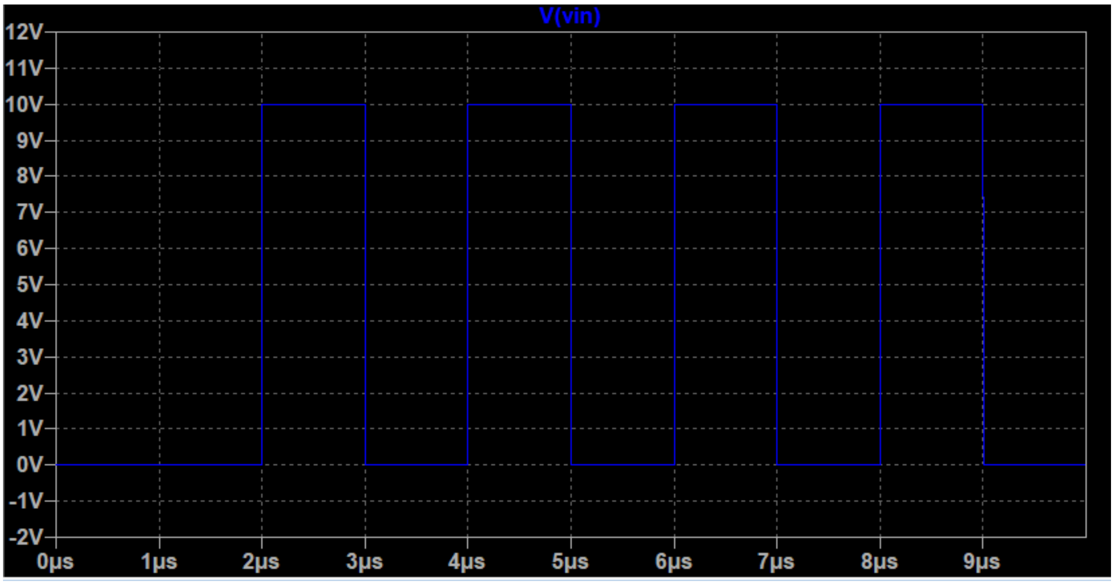

Figure 5 – The Input Voltage

The input and output on the same graph can be observed below.

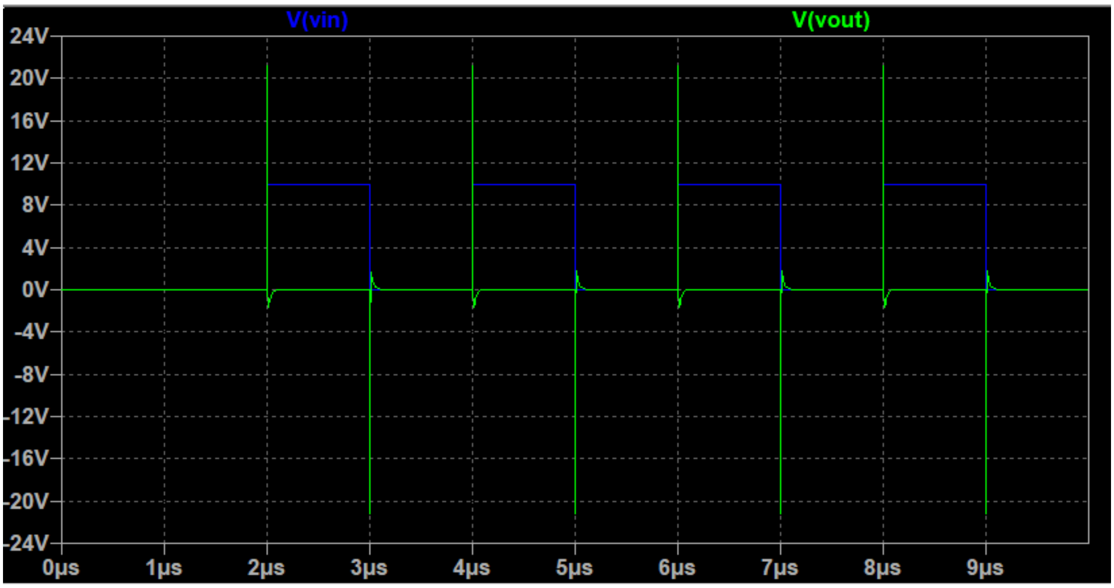

Figure 6 – Input and Output Voltages

We can easily see the spike is in the range of 20V and 24V, which is acceptable according to the requirements. There is a symmetry in terms of voltages as it goes down exactly as it goes up, however, there isn't any limitation that prohibits this, so that is also acceptable. The next thing to do is to check on one of the spikes carefully and find the exact voltage and width.

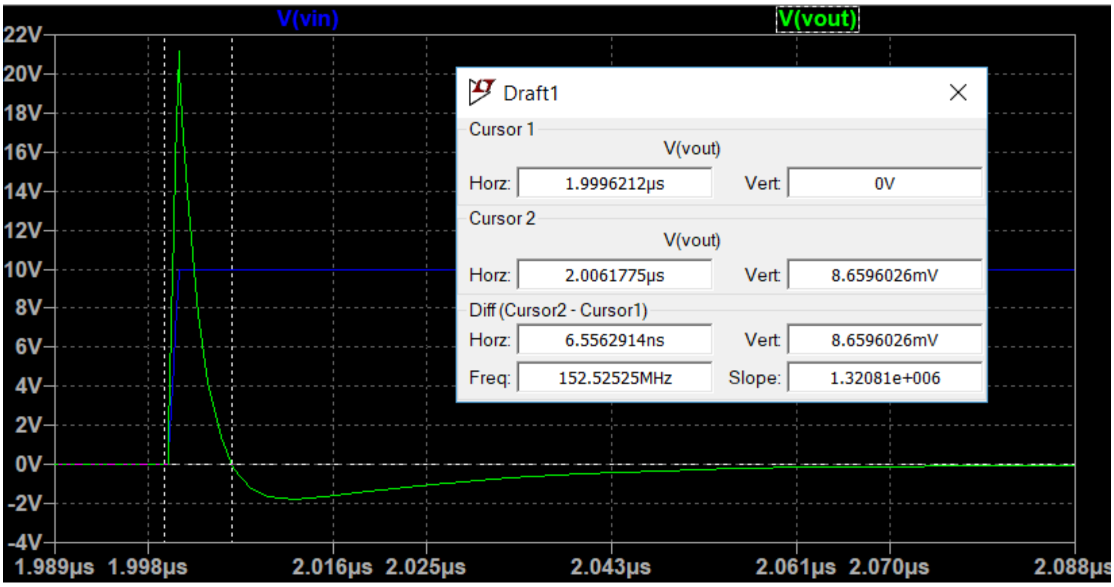

Figure 7 – The Close-up Look of One of the Spikes and the Width

As we can easily observe from the graph, the time difference is equal to 6ns, which is way lower than the expected result 100ns. The reason is that the circuit actually acts like an RLC circuit on the right side of the inductor, however as we designed both the inductance value and the capacitor value accordingly, it is normal that we would get a smaller value and this is actually a positive result as 6ns is 16.667 times smaller than the limit. Now we need to observe the voltage of the spike if it is between the desired 20-30V range. To do that, we again zoom into the spike and try to find the maximum point with the use of the cursor. The graph of the spike and correctly positioned cursor is given below.

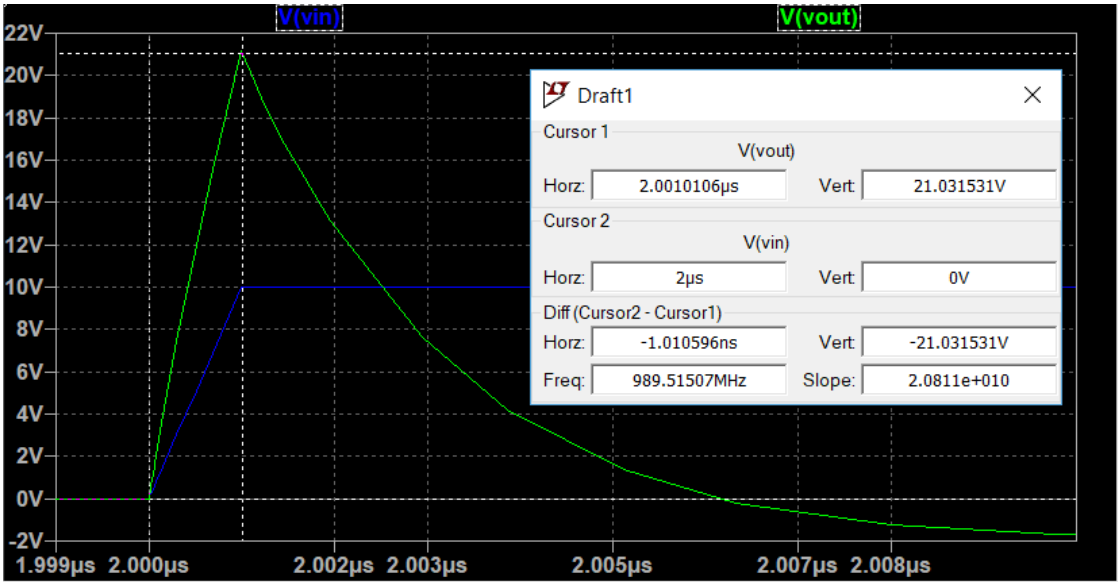

Figure 8 – The Close-up Look of One of the Spikes and the Maximum Voltage

It is easily observable from the graph that the maximum voltage is equal to 21.031531V, which is perfectly acceptable within our range. This means that the spike that we have created has all the attributes that it needs to have.

Conclusion

To conclude, we have successfully obtained a voltage spike between 20-30V and with under 100ns of width at half amplitude in a square wave with frequency below 5MHz. The results show that we have obtained a 21V peak voltage with 6ns of width. We could have increased this voltage by changing the capacitor value or changing the winding ratio of the transformers.

Furthermore, we have used a simple transformer circuit in order to convert a square wave into a voltage spike. While the transformer helped the voltage to increase, the RLC Circuit made it sure that the half-width is smaller then 100ns. There were notable errors when compared with the calculations, however, they were negligible as we still have obtained acceptable results despite them. As we tried to use standard values for the resistors and neglected the inductance of the RLC circuit after choosing it small enough, it is normal that there are errors occurring.

This was a very useful lab exercise in terms of understanding the way how transformers work and how to make voltage spikes with square waves. We learned more about RLC and RC circuits, while trying to create a more accurate design. Furthermore, our skills in terms of LTSpice has improved notably since, we learned how to create transformers with inductances and SPICE directives. Overall it was a beneficial laboratory exercise.