Pulse Width Modulation Generator (PWM Generator)

Student : Pop Emanuel.

Group : 2021.

Coordinator professor: Fizeșan Raul Traian.

Project Requirements :

$$\begin{cases} Fn_1 = 60\% \\ Fn_2 = 80\% \\ APWM_1 = 1 \\ APWM_2 = 6 \\ f = 600Hz \end{cases}$$

Contents

Section 1: Introduction

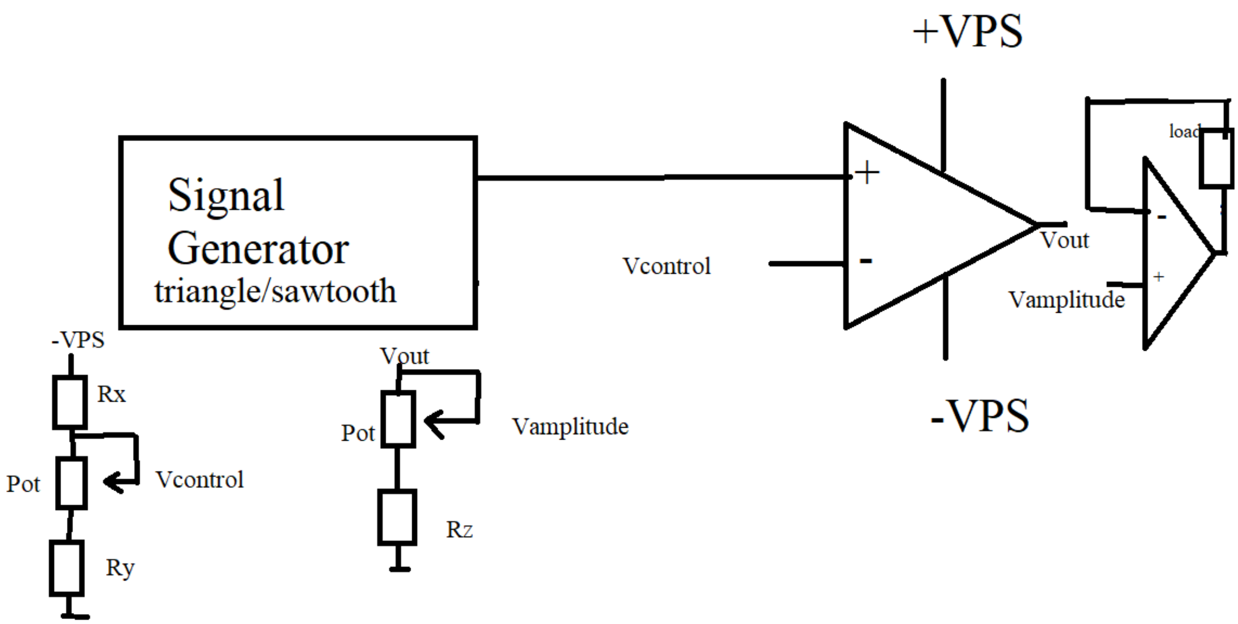
Pulse width modulation (PWM) is a method of changing the duration of pulse with respect to any analog input. The duty cycle of a square wave is modulated to encode a specific analog signal level. The PWM signal is digital because, at any given instant of time, the full DC supply is either fully on or fully off.

PWM method is commonly used for speed controlling of motors, fans, lights in varying intensities etc. (pulse width modulation controller). PWM signals may also be used to approximate time-varying analog signals. PWM is employed in a wide variety of applications :

- Measurements;
- Communications;
- Power control;

For example, the PWM is commonly used to control the speed of electric motors, the brightness of lights, in ultrasonic cleaning applications, and many more.

Section 2: Block Diagram



Section 3 : Circuit's Functioning

Considering the block diagram above :

- The op amp will act as a comparator.
- It will compare the input signals connected to the op amp : the DC signal gave by the voltage source (input signal) and the triangular / sawtooth signal gave by the signal generator.
- When the signal generated by the signal generator intersects the input signal, the op amp will do a switch.
- The DC signal has the role of controlling the duration of a pulse between the high / low levels set by the VPS power supplies.
- The signal generator is required in the circuit in order to deliver a signal that will help the comparator to do the switch.
- The frequency of the output signal is the same as the frequency of the signal generator's signal.

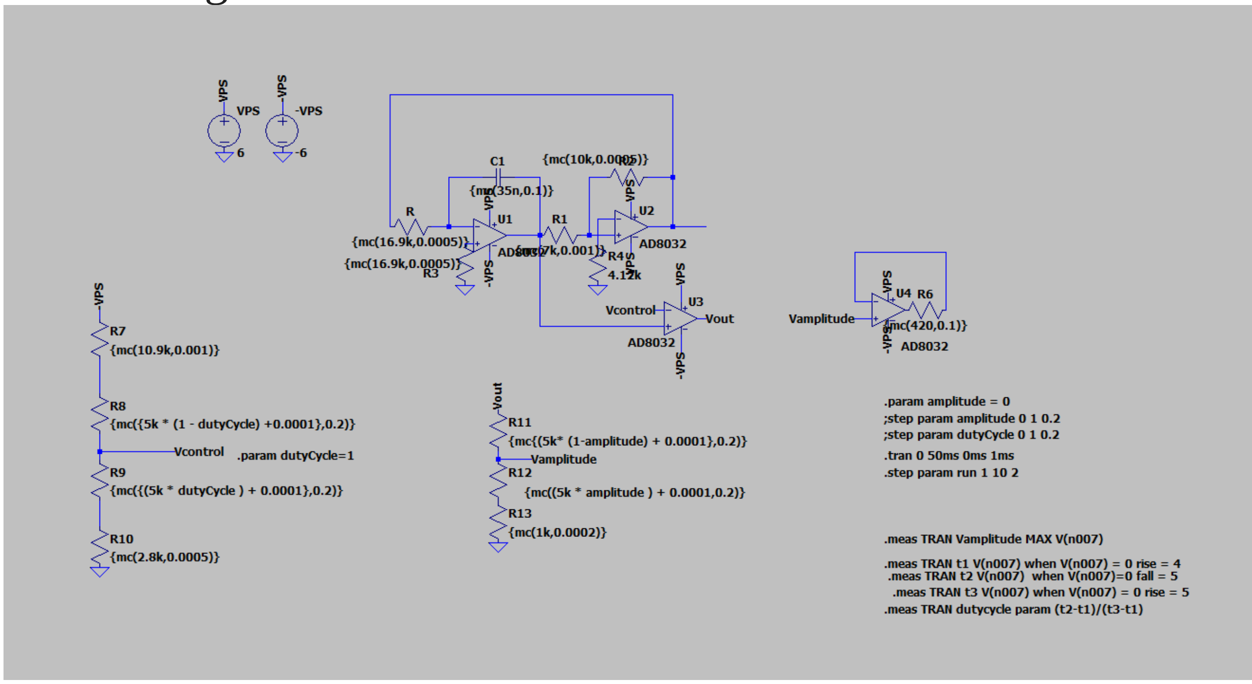
Important parameters :

Period (T) -> How long each complete pulse cycle takes.

Frequency (F) -> How often the pulses are generated. This value is specified in Hz (cycles per second).

Duty Cycle -> Refers to the ratio of time in the period that the pulse is active or high to the total time of a cycle. The duty cycle is typically specified as a percentage of the full period.

Signal Generator Electrical Schematic



In order to achieve the frequency = 600 Hz we need to apply the following formula :

$$\boxed{T = 2RC\frac{V_{ThH} - V_{ThL}}{V_{OH}} = 4RC\frac{R_1}{R_2}}$$

Initially chosen :

- C = 35 nF;
- R_1 = 7 kΩ;
- R_2 = 10 kΩ

  R_2 has to be slightly bigger than R_1 in order for the signal generator to function.

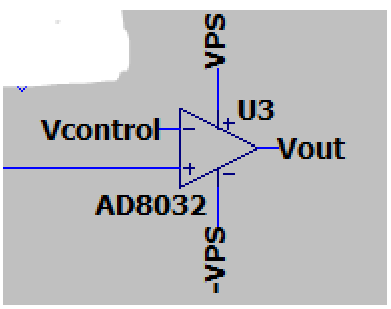
f = $\frac{1}{4*RC} * \frac{R_2}{R_1} = \frac{1}{2400*35*10^{-9}*R} * \frac{10}{7} \Rightarrow R = \frac{10^9}{35*2400}$ * 1.428 = 17 kΩ, but the tolerance available for this component is 1%, so we'll choose the value of

16.9 kΩ instead, since it has more tolerances available, resulting a frequency of : 603.5 Hz => period of 1.657 ms.

$$\text{The amplitude of the triangular signal} = \frac{R_1}{R_2} * (V_{PS}) = 4.2$$ V

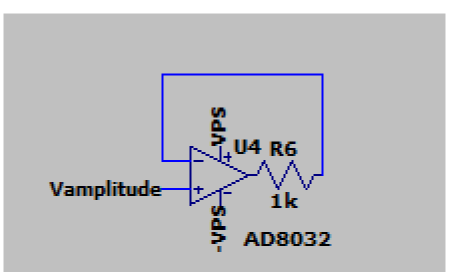
R_3 and R_4 are placed at the inputs of the op amp in order to “balance it” so there is no big pressure applied on a single input. R_4 is the parallel equivalence of R_1 and R_2

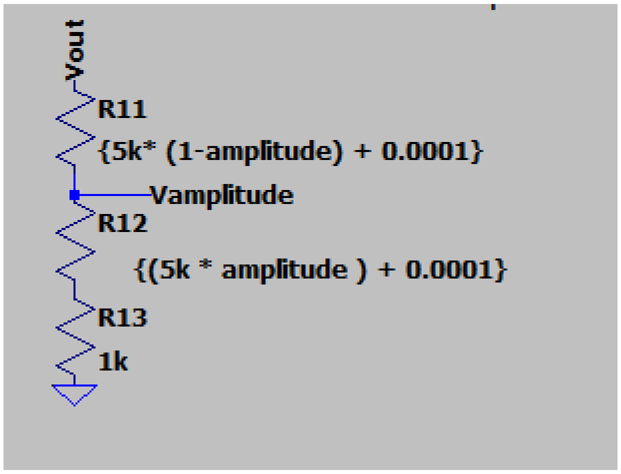
Op amp comparator electric schematic, with the DC voltage as input :



The output of this block will go into another opAmp in order to be able to vary the amplitude of the output signal. The required amplitude is between 1V and 6V.

Adjustable output





The resistors R_{11} and R_{12} represent a potentiometer (one of the possible ways of implementing one in LTspice). We will use Vamplitude later, in order to connect it to the positive input of the buffer, which will be a voltage follower.

For the potentiometer we will choose the standard value of : 5 kΩ. Leaving R_{13} the only component that has to be computed

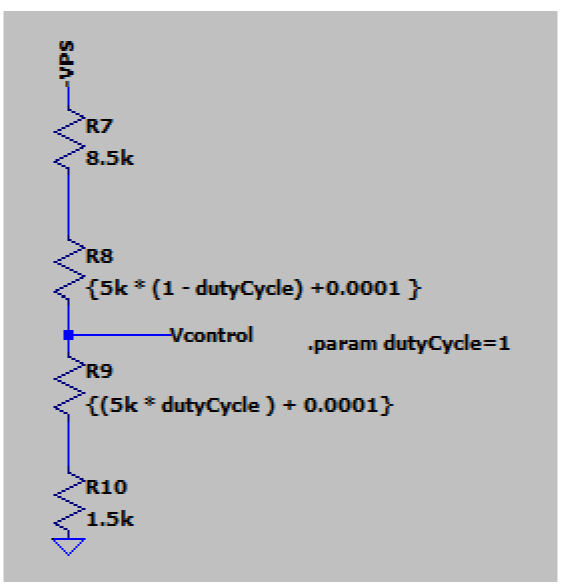
$$\frac{R_{13}}{R_{13} + P} * V_{PS} = 1$$

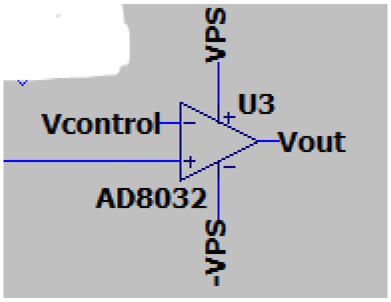
⇨ $6R_{13} = R_{13} + 5\ k\Omega$
⇨ $R_{13} = 1k\Omega$.

We can observe inside the parameter brackets of the potentiometer's implementation that there is +0.0001 value. This is to avoid the risk that the potentiometer would have the value 0 which would affect the circuit's functioning (in terms of the software program).

Adjustable Duty Factor

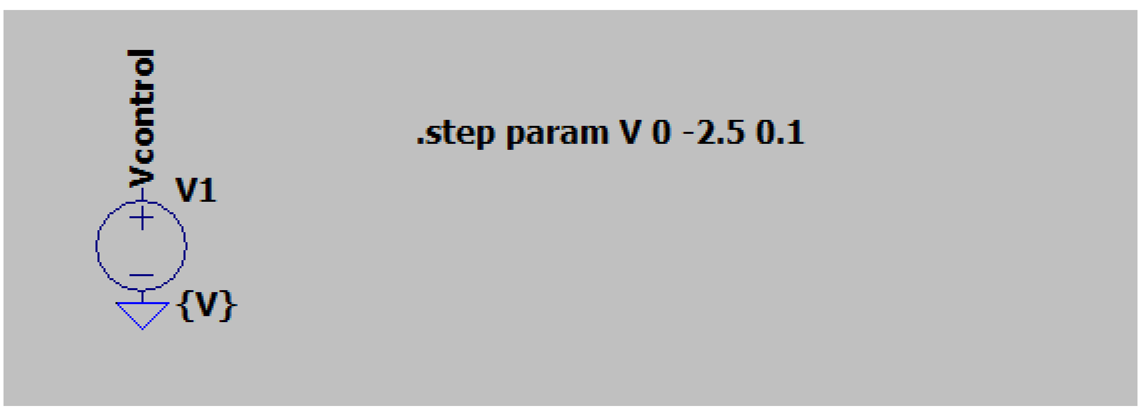
$$\delta = \frac{T_{on}}{T}$$

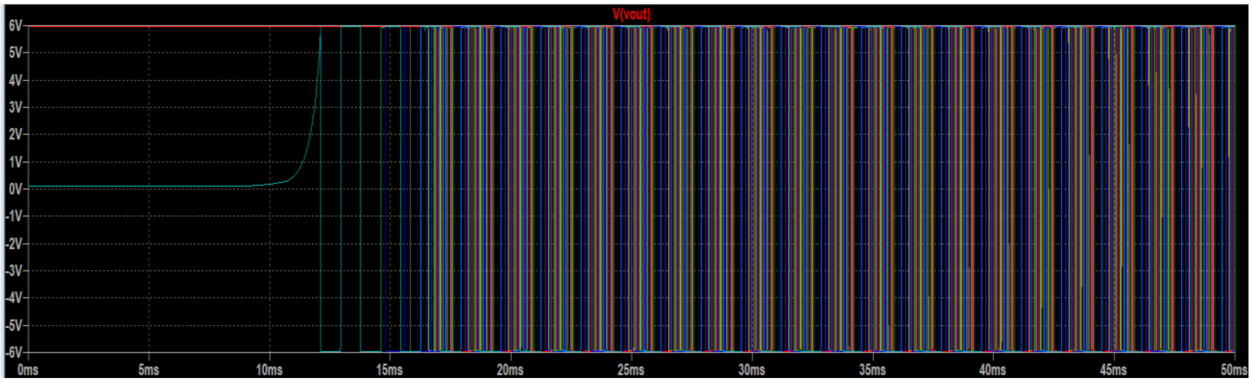


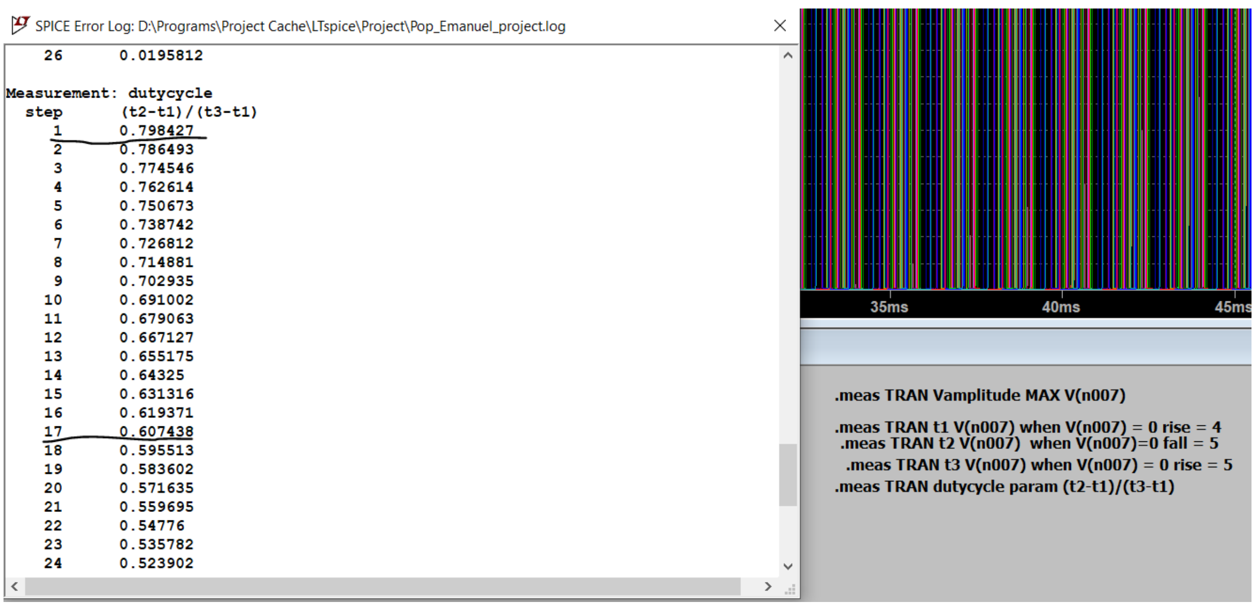


For the variable duty factor (required between 60% and 80 %), we will use again a potentiometer in order to vary the voltage, in this case R_8 *and* R_9 representing the implementation of it, and it will have the same value as the previous used potentiometer (5 kohm's) in order to avoid using to many different components. We will have to compute R_7 *and* R_{10}.

In order to find the optimal voltage for the varying duty factor, we will vary a DC voltage source and choose certain steps to find the values.







SPICE Error Log: D:\Programs\Project Cache\LTspice\Project\Pop_Emanuel_project.log

```
   26      0.0195812

Measurement: dutycycle
  step       (t2-t1)/(t3-t1)
     1       0.798427
     2       0.786493
     3       0.774546
     4       0.762614
     5       0.750673
     6       0.738742
     7       0.726812
     8       0.714881
     9       0.702935
    10       0.691002
    11       0.679063
    12       0.667127
    13       0.655175
    14       0.64325
    15       0.631316
    16       0.619371
    17       0.607438
    18       0.595513
    19       0.583602
    20       0.571635
    21       0.559695
    22       0.54776
    23       0.535782
    24       0.523902
```

```
.step v=-2.5
.step v=-2.4
.step v=-2.3
.step v=-2.2
.step v=-2.1
.step v=-2
.step v=-1.9
.step v=-1.8
.step v=-1.7
.step v=-1.6
.step v=-1.5
.step v=-1.4
.step v=-1.3
.step v=-1.2
.step v=-1.1
.step v=-1
.step v=-0.9
.step v=-0.8
.step v=-0.7
.step v=-0.6
.step v=-0.5
.step v=-0.4
.step v=-0.3
.step v=-0.2
.step v=-0.1
.step v=0
```

The equations will be the following (we can simply multiply with a -1, since the voltage divider and the result after the ‘=’ sign will be negative) :

We choose the same potentiometer used previously (5 $k\Omega$).

$$\begin{cases} \dfrac{R_{10} + 5 * 10^3}{R_{10} + 5 * 10^3 + R_7} * 6 = 2.5 \\ \dfrac{R_{10}}{R_{10} + 5 * 10^3 + R_7} * 6 = 0.9 \end{cases}$$

$$\begin{cases} 6R_{10} + 30 * 10^3 = 2.5R_{10} + 12.5 * 10^3 + 2.5R_7 \\ 6R_{10} = 0.9R_{10} + 4.5 * 10^3 + 0.9R7 \end{cases}$$

⇨ $R_{10} = \frac{0.9R_7 + 4.5 * 10^3}{5.1}$

⇨ $3.5 * \frac{0.9R_7 + 4.5 * 10^3}{5.1} + 30 * 10^3 = 2.5R_7 + 12.5 * 10^3$

⇨ $R_{10} = 2.8\ k\Omega$

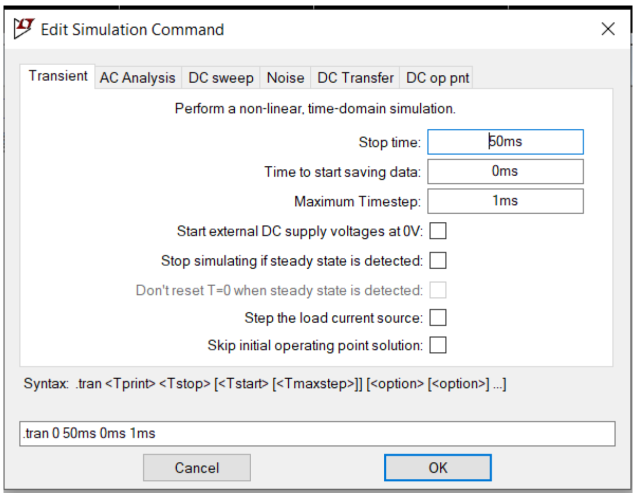
⇨ $R_7 = 10.9k\Omega$

Section 4 : Simulations

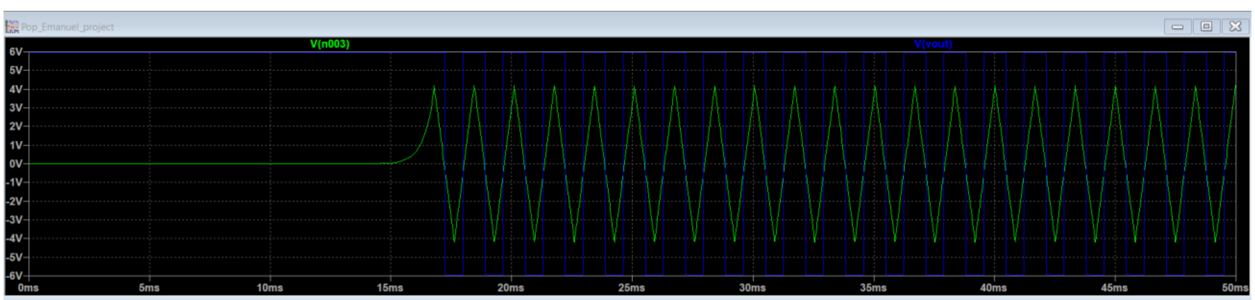
I. Transient

➔ for the transient simulation we will want at least 50 point / period, so we'll respect the following formula : $\frac{StopTime - StartTime}{maxStepSize} = 50$. We will choose the maximum step size = 1 ms, => StopTime = 50ms.

➔ in order to study the signal we will use a transient simulation.

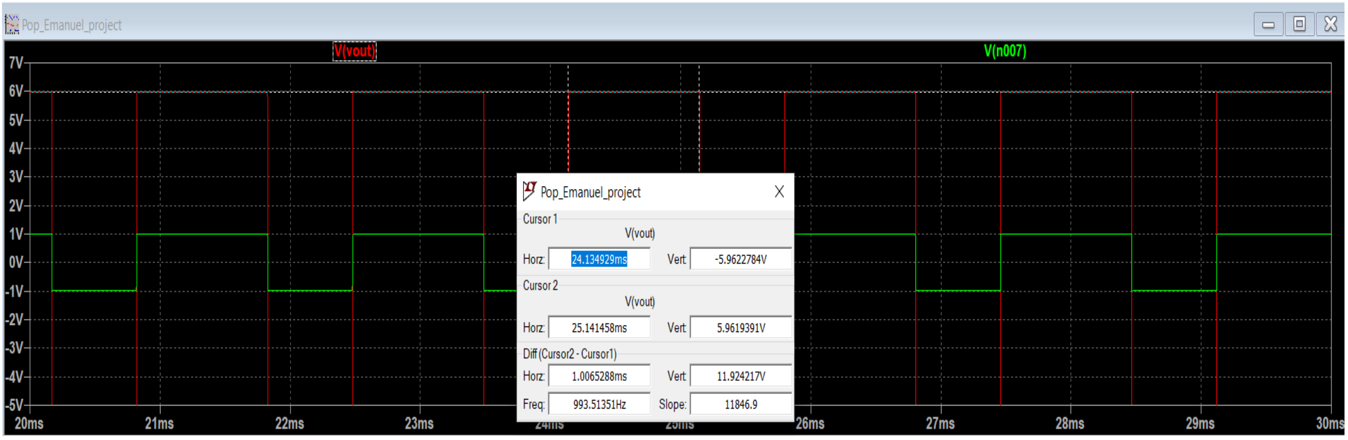


The graph is displayed below :



Let's take a look at it's functionality :

➔ the duty factor can vary between 60% and 80% as required.

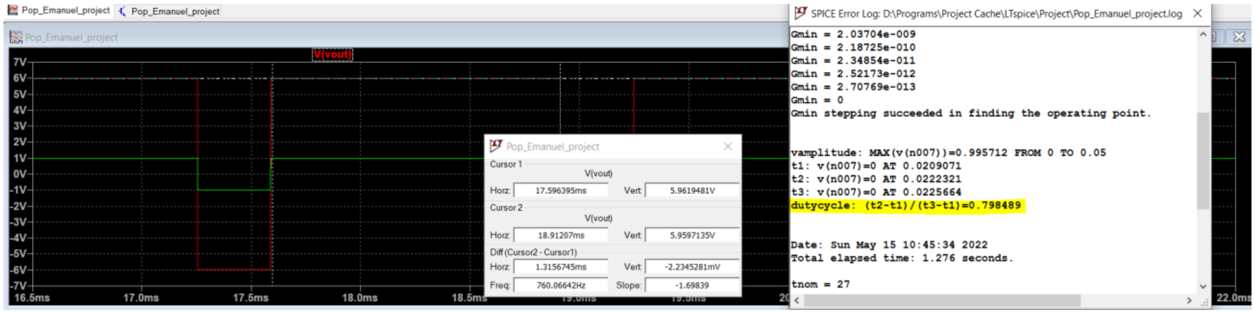


SPICE Error Log: D:\Programs\Project Cache\LTspice\Project\Pop_Emanuel_project.log

```
Gmin = 0.00132923
Gmin = 0.000142725
Gmin = 1.5325e-005
Gmin = 1.6455e-006
Gmin = 1.76685e-007
Gmin = 1.89714e-008
Gmin = 2.03704e-009
Gmin = 2.18725e-010
Gmin = 2.34854e-011
Gmin = 2.52173e-012
Gmin = 2.70769e-013
Gmin = 0
Gmin stepping succeeded in finding the operating point.


vamplitude: MAX(v(n007))=0.995369 FROM 0 TO 0.05
t1: v(n007)=0 AT 0.0208211
t2: v(n007)=0 AT 0.0218285
t3: v(n007)=0 AT 0.0224804
dutycycle: (t2-t1)/(t3-t1)=0.607119


Date: Sun May 15 10:32:53 2022
Total elapsed time: 1.182 seconds.

tnom = 27
temp = 27
method = modified trap
```

If we set the dutyCycle parameter to 0, we will obtain the lower duty factor, as computed in the previous equations.

In the same manner we will compute the higher duty factor, by setting the potentiometer' parameter to 1.



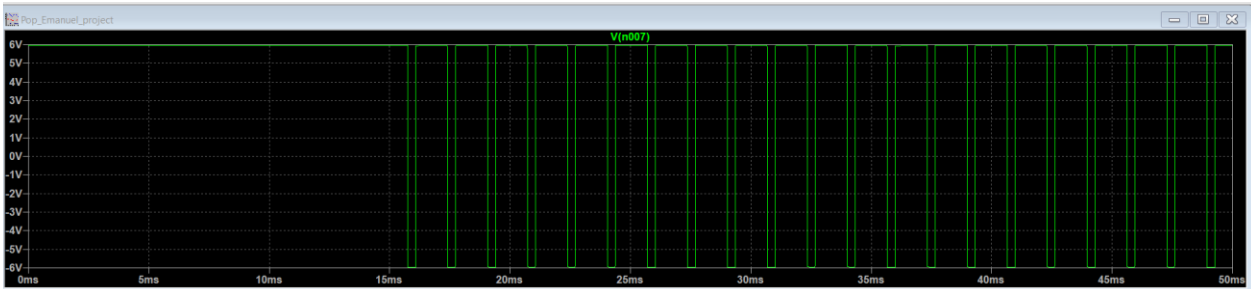
And we can see that the duty factor will be equal to 79.

We control the values of the parameter by the command .param paramName = value in the SPICE Directive.

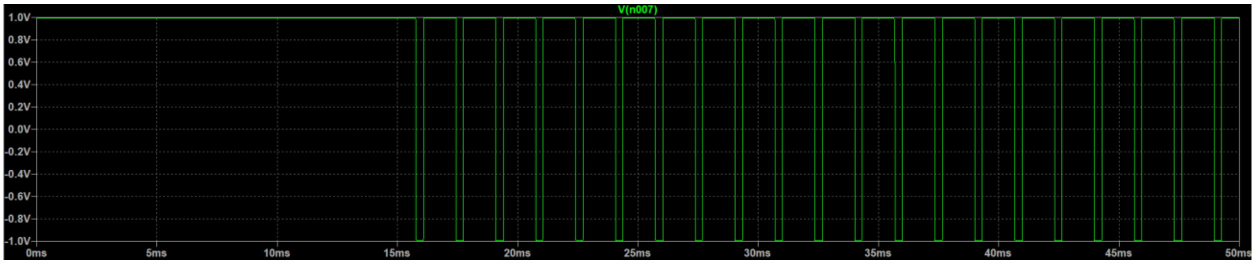
Now, let’s take a look at the varying amplitude :

- ➔ the requirement for the amplitude is to be between 1V and 6V.
- ➔ by using a voltage follower, we will be able to separate the impedances and with the help of a potentiometer we will vary the voltage so we can get the desired amplitude.

We can observe that if we set the potentiometer parameter (named amplitude for this case) we will obtain the maximum of 6V.



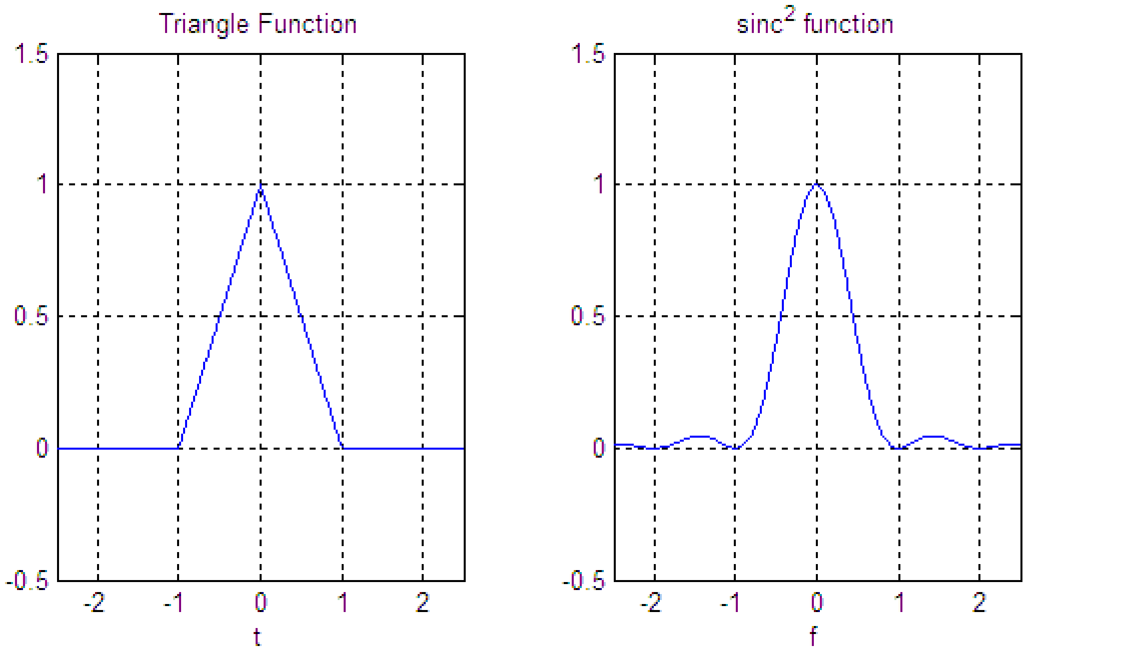
For the minimum value of 1V, all we have to do is to switch the ‘amplitude’ parameter to 0.

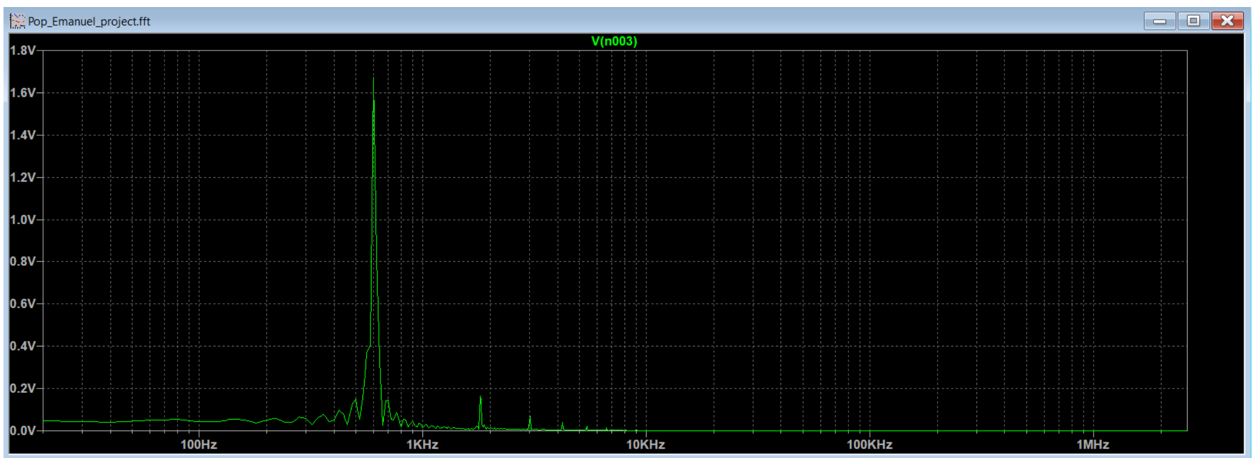


The potentiometer parameter can be controlled in the same manner as explained above.

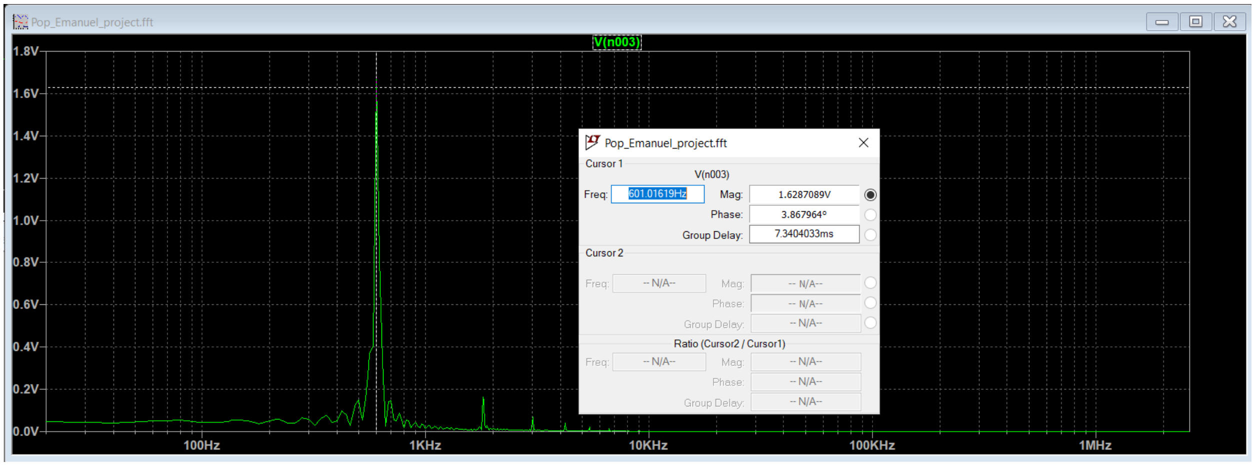
II. Fast Fourier Transform

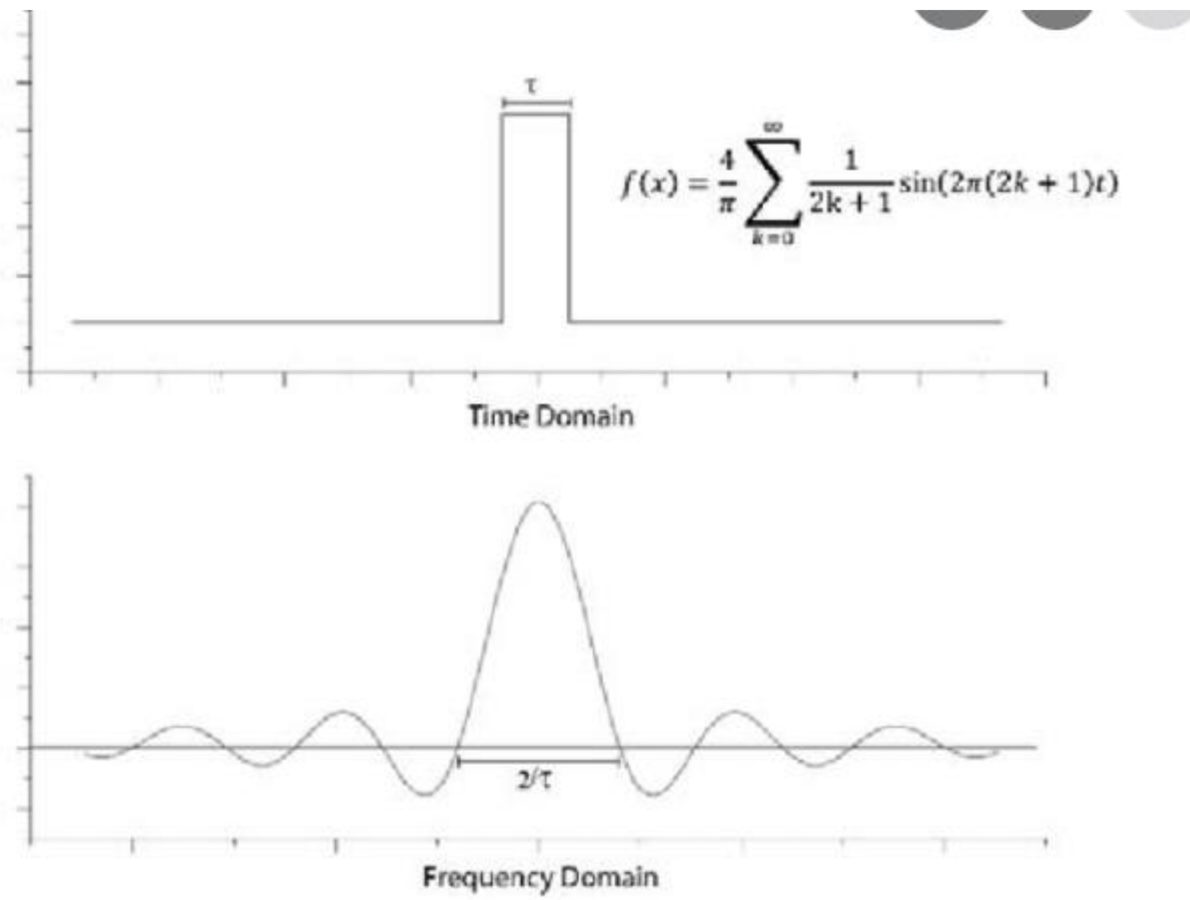
A fast Fourier transform (FFT) is **an algorithm that computes the discrete Fourier transform (DFT) of a sequence, or its inverse (IDFT)**. Fourier analysis converts a signal from its original domain (often time or space) to a representation in the frequency domain and vice versa.

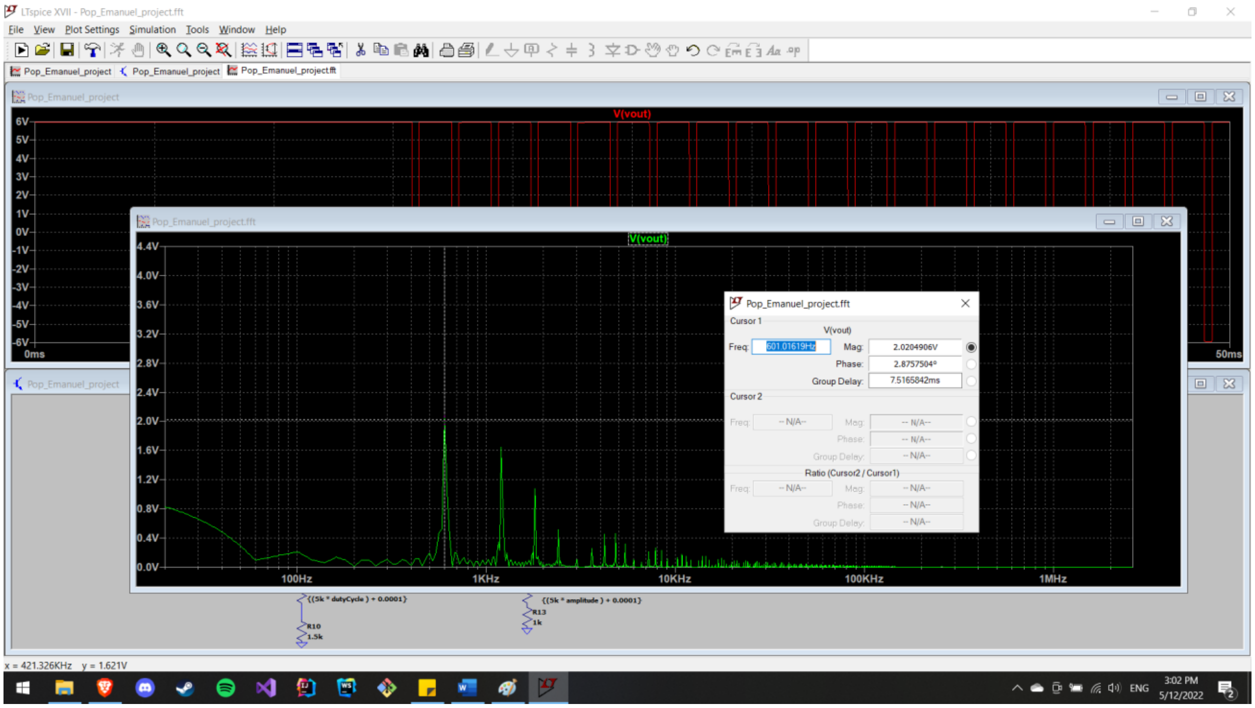




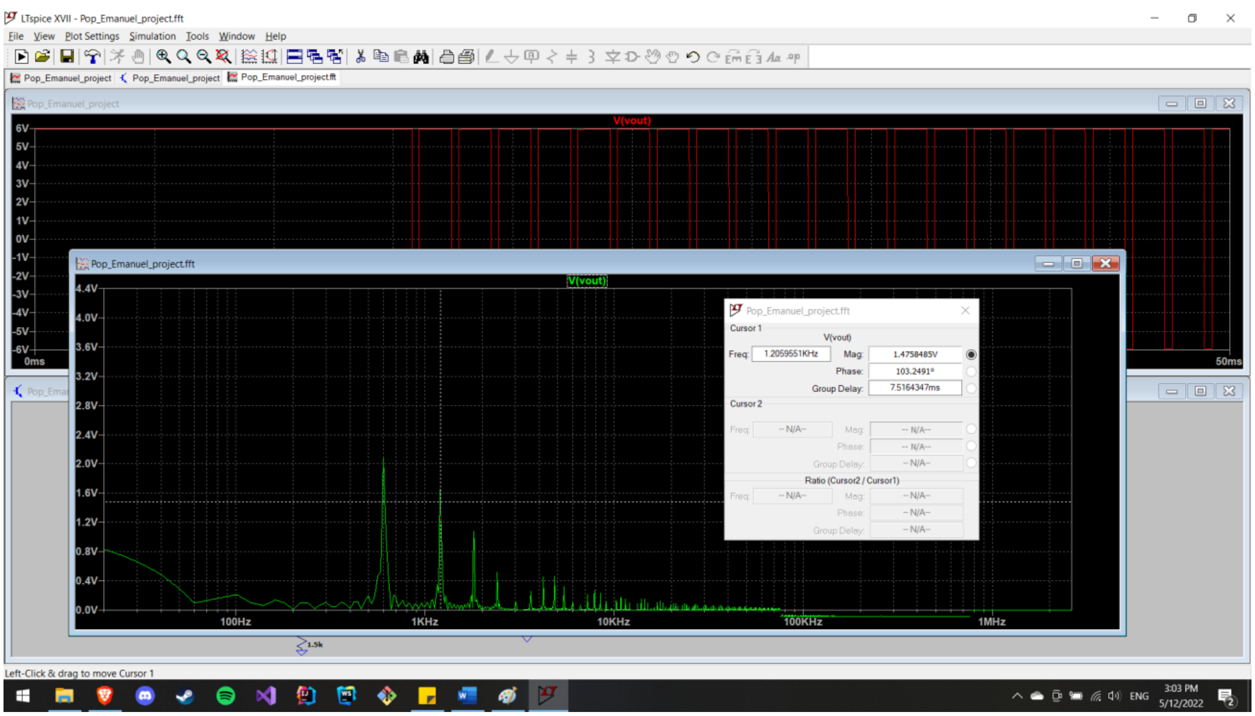
If we set the cursor to the frequency of our circuit, we can notice that at the frequency of 600 Hz, there is a harmonic.

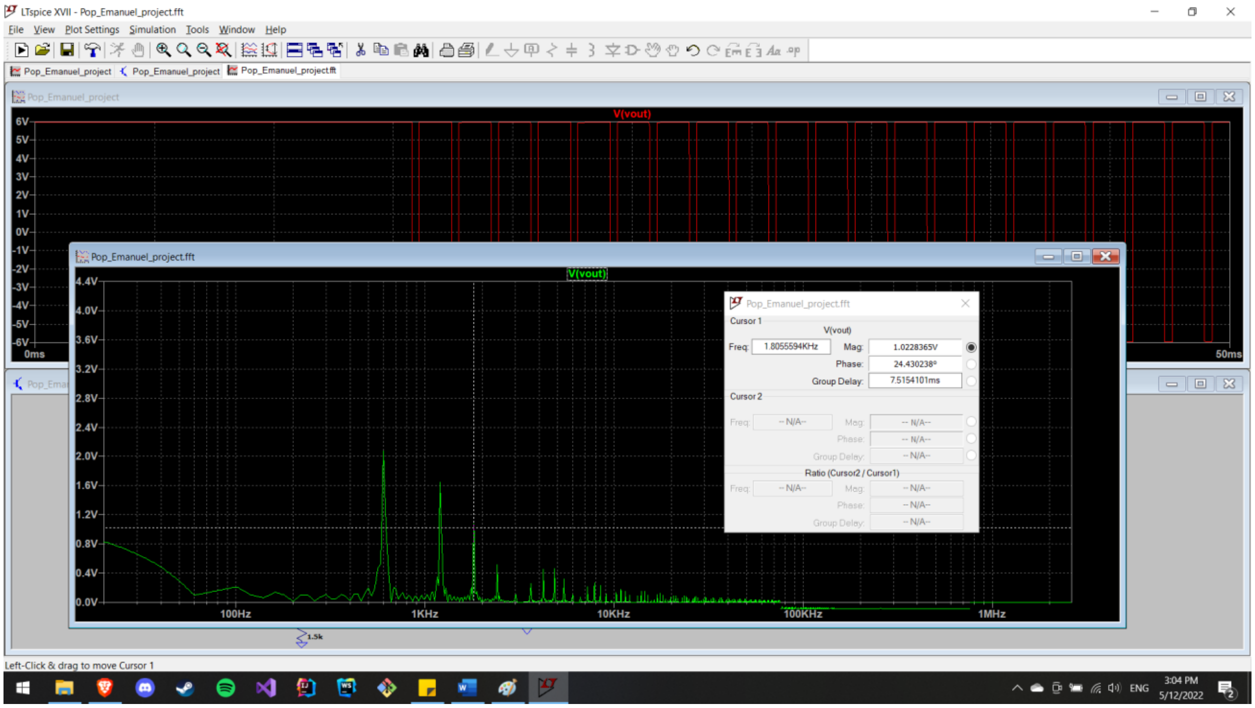

Pop_Emanuel_project.fft
V(n003)
1.8V
1.6V
1.4V
1.2V
1.0V
0.8V
0.6V
0.4V
0.2V
0.0V
100Hz
1KHz
10KHz
100KHz
1MHz
Pop_Emanuel_project.fft
Cursor 1
V(n003)
Freq: 601.01619Hz
Mag: 1.6287089V
Phase: 3.867964°
Group Delay: 7.3404033ms
Cursor 2
Ratio (Cursor2 / Cursor1)


τ
$f(x) = \frac{4}{\pi}\sum_{k=0}^{\infty}\frac{1}{2k+1}\sin(2\pi(2k+1)t)$
Time Domain
2/τ
Frequency Domain



At the frequency of 600 Hz, we can see a harmonic. They are also noticed at multiples of our frequency :

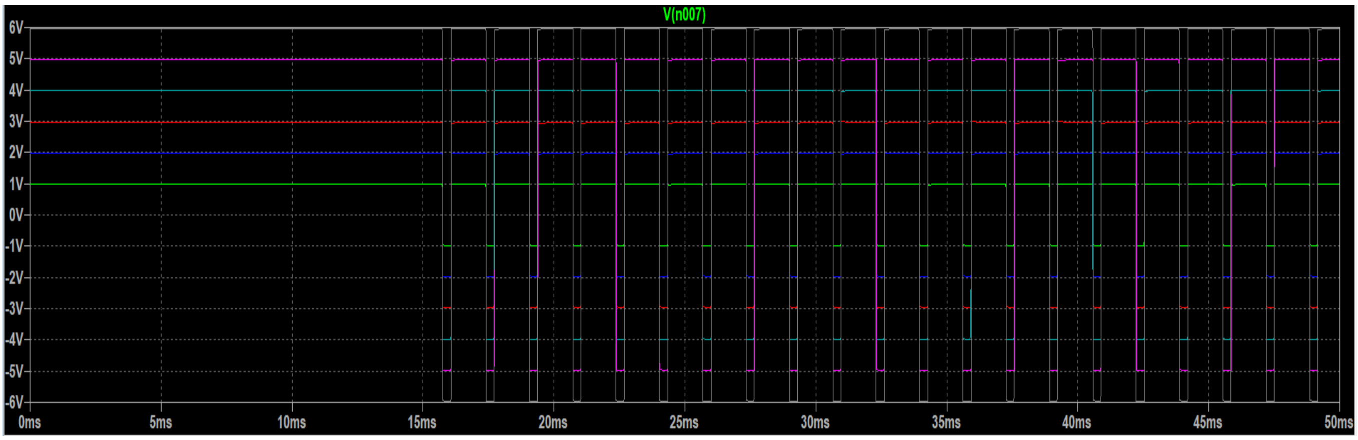



LTspice XVII - Pop_Emanuel_project.fft
File View Plot Settings Simulation Tools Window Help
V(vout)
Pop_Emanuel_project.fft
Cursor 1
V(vout)
Freq: 1.8055594KHz
Mag: 1.0228365V
Phase: 24.430238°
Group Delay: 7.5154101ms
Cursor 2
Ratio (Cursor2 / Cursor1)
100Hz
1KHz
10KHz
100KHz
1MHz
Left-Click & drag to move Cursor 1

III. Step Param

➔ in this scenario we will vary the potentiometer parameter in order to see that we get the desired results.

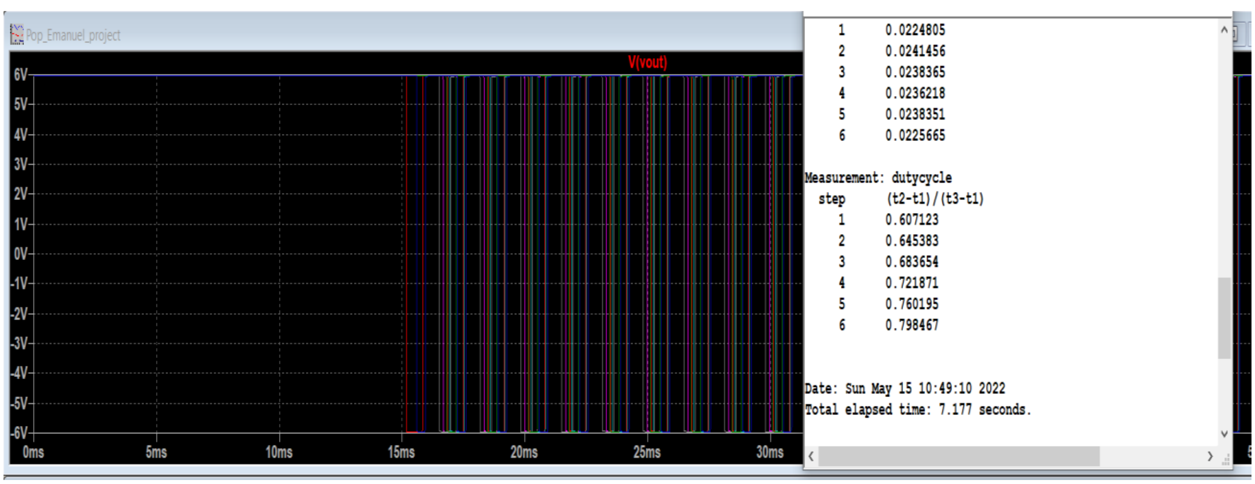
.step param amplitude 0 1 0.2



We can notice that the output amplitude doesn't go over 6V, neither under 1V. So, from this point of view, the circuit works between the desired parameters.

Let's take a look at circuit part responsible for varying the duty factor :

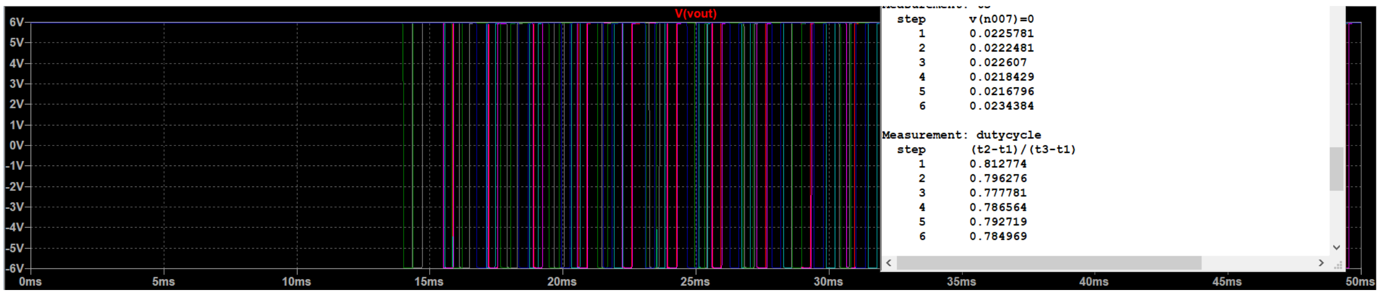
.step param dutyCycle 0 1 0.2



We can observe here as well, that the circuit is functioning within the intervals we measured and computed earlier.

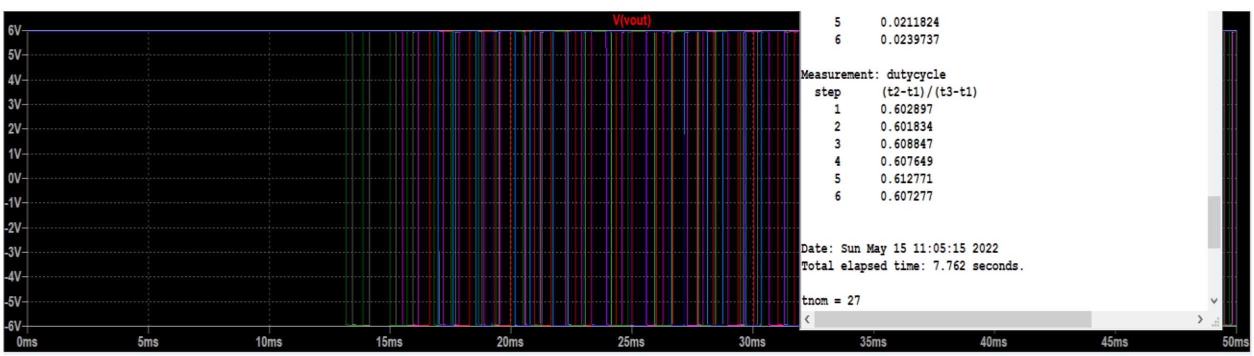
IV. Monte Carlo

➔ for this type of analysis we want to see how do the tolerances of our circuit components affect the functioning of it.



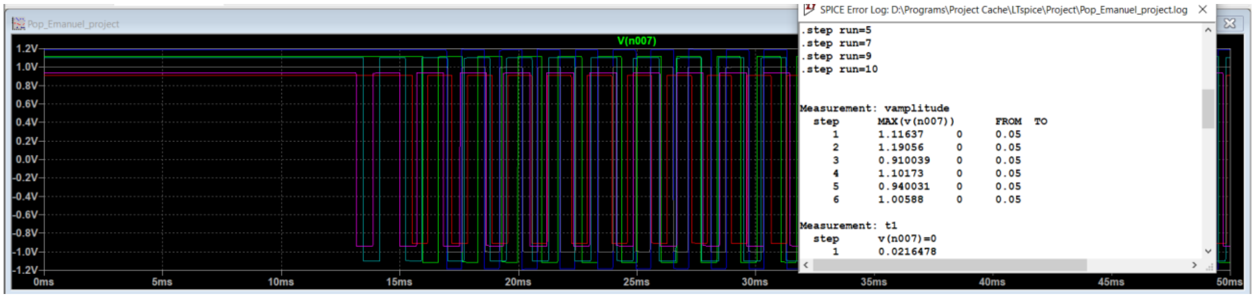
Our duty cycle potentiometer' parameter was set on 1. This means we are expecting to see a duty cycle around 79%, which we do, if we take a look at the SPICE Error Log. For the chosen tolerances, the circuit is varying it's top duty cycle between 78 % and 81%.

Let's set the dutyCycle param to 0 and analyze the results.

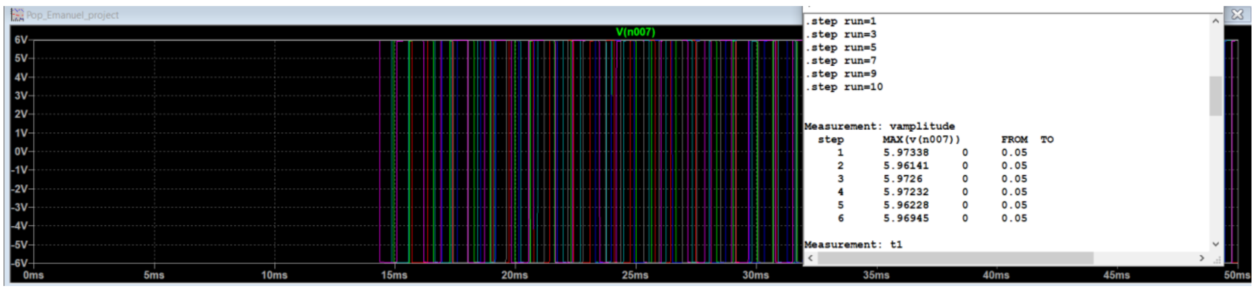


We can observe that the duty cycle varies around the value we want, which is 60%.

Let’s take a look at the amplitude now.



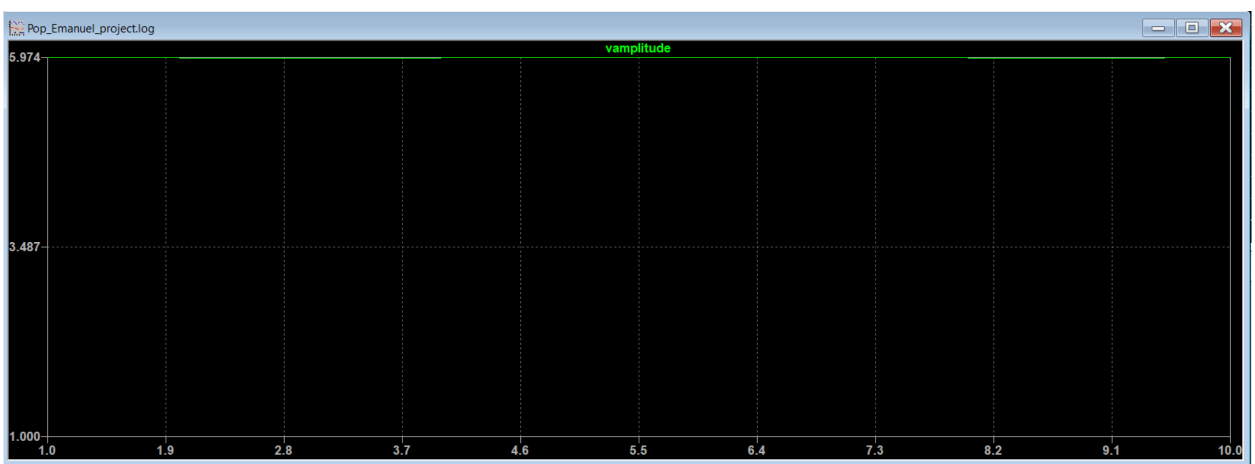
The “amplitude” parameter was set on 0. Which means we are expecting a voltage of amplitude around 1. And if we take a look at the graph and at the SPICE Error Log, we can see that the circuit works as we’d expect.



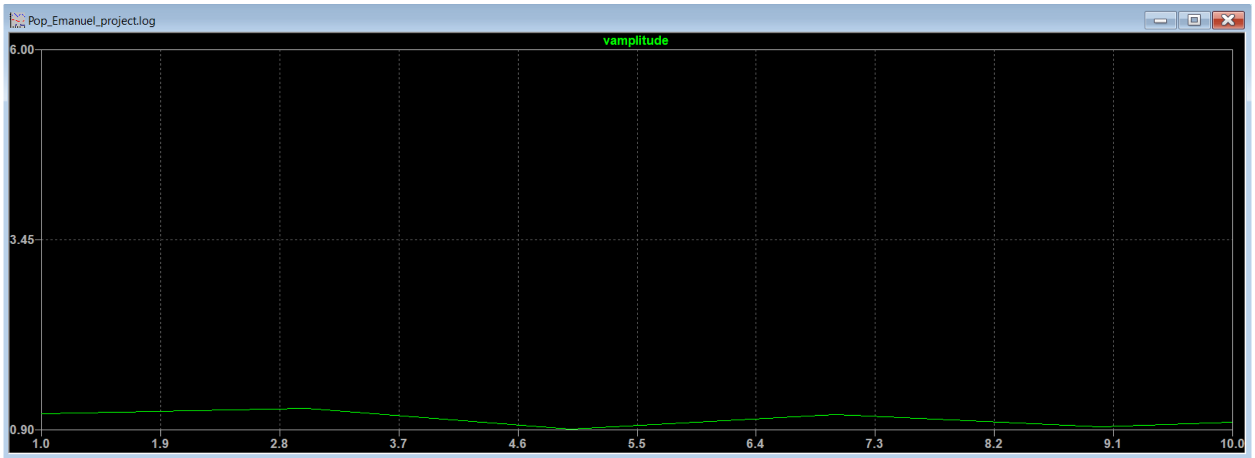
The same principle applies for our upper limit voltage.

V. Performance Analysis

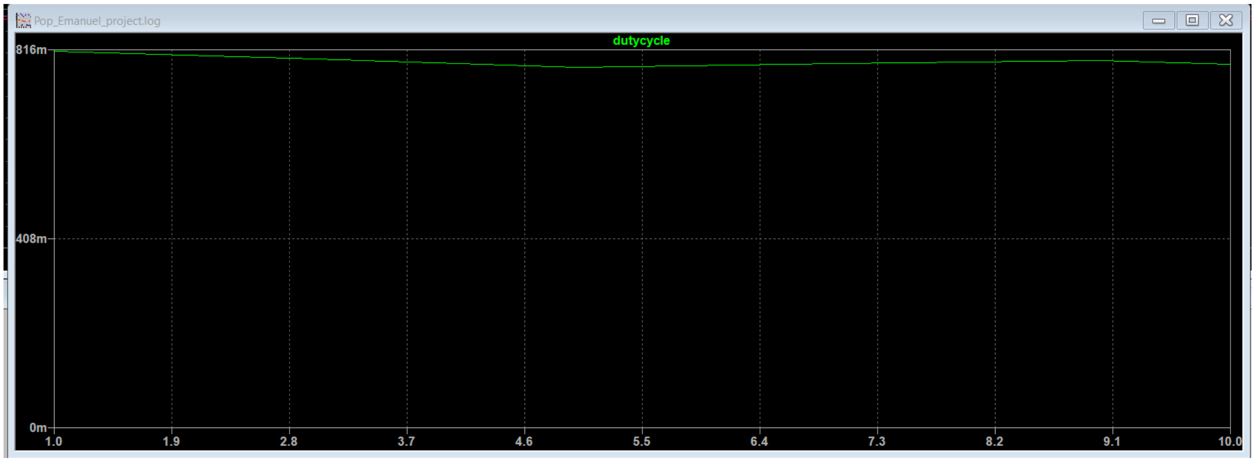
➔ with this analysis we will see the variation of the circuit, plotted.



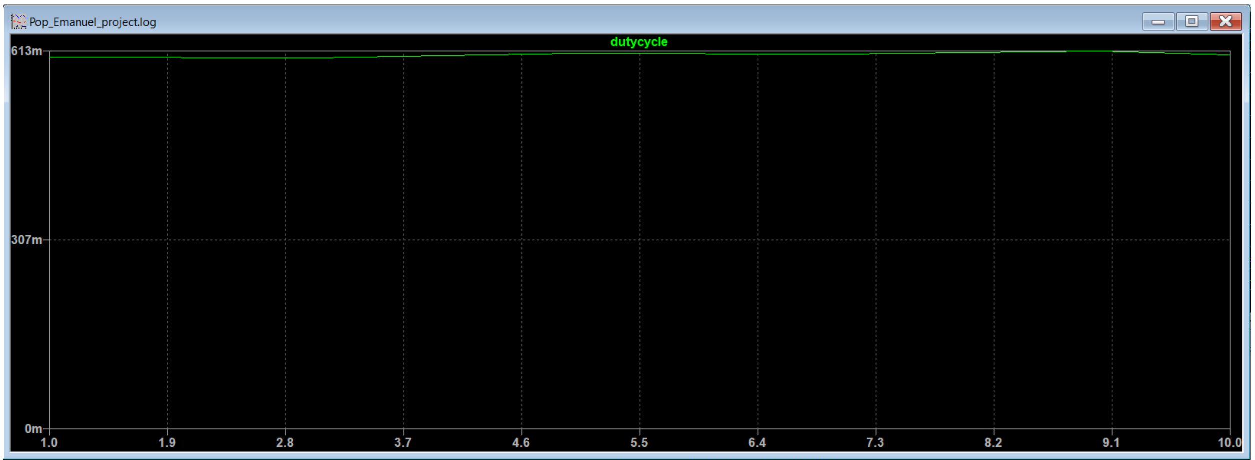
In the case of the top limit amplitude, the circuit does not vary at all, the op amp being supplied by a 6V source also helps to achive this stability.



While at the lower amplitude limit we see a few variations, at certain points of time, but the variation is around 200mV, from the lowest value to the highest, which is acceptable.



We can state the same idea for the duty cycle.


Pop_Emanuel_project.log
dutycycle
613m
307m
0m
1.0
1.9
2.8
3.7
4.6
5.5
6.4
7.3
8.2
9.1
10.0

Section 5 : Components Used

- 2x Thin Film Resistors - SMD 1/10W 16.9K Ohm .05% 0603 25ppm | ***Mfr. Part No.*** RG1608P-1692-W-T5 ***Mouser Part No***754-RG1608P-1692-WT5 = 1.54 RON
-

Metal Film Resistors - Through Hole 1/8watt 7Kohms .1% 10ppm = 8.22 RON | ***Mfr. Part No.*** PTF567K0000BYEK

- Thin Film Resistors - SMD 10KOhm 0.01% 0805 5 PPM | ***Mfr. Part No.*** APC0805T10K0Z = 7.57 RON
- Thin Film Resistors - SMD 1/16W 4.12Kohm 0.5% 25ppm | ***Mfr. Part No.***RR0816P-4121-D-60H =0.47 RON
- Thin Film Resistors - SMD 0805 10K9 Ohms 0.1% | ***Mfr. Part No.***RN732ATTD1092B50 = 2.84 RON
- Thin Film Resistors - SMD Thin Film Chip Resistors 0603 size, 1/16W, 2.80 Kohm, 0.05%, 10ppm | ***Mfr. Part No.*** RG1608P-2801-W-T1 = 3.39 RON

- 2x Precision Potentiometers rotary potentiometer single channel horiz 25mm D-cut detent linear taper 5K ohm | ***Mfr. Part No.*** PT01-D225D-B502 = 7.6 RON
- Thin Film Resistors - SMD 1/10W 1K Ohms 0.02% 0603 25ppm | ***Mfr. Part No.*** RG1608P-102-P-T1 = 3.91 RON
- Metal Film Resistors - Through Hole | ***Mfr. Part No.*** MFR-25FTF52-420R = 0.52 RON
- Safety Capacitors 800V 0.035uF 10% LS=10mm | ***Mfr. Part No.*** F871AO353K330A = 3.34 RON
- 3x Operational Amplifiers - Op Amps MINI-SO DUAL LOW POWER OP AMP | ***Mfr. Part No.***AD8032ARMZ-REEL7 = 80.19 RON

Total Price : 119.59 RON

Why the components above have been chosen ?

For the resistors we want to get a value as close, or exactly (ideal cases), to the ones we compute. And for that reason we either look for that specific value, or we look for the closest standard value close to the value we computed with a small tolerance.

In general, we want small tolerances in order for our circuit to function with our components having their nominal value (because we use them for computing).

We also are looking for cheap components, but for this project that wasn't the main reason. The priority was getting the desired value and tolerance and then the best price.

Most of the values above have been explained earlier in this documentation, but the 420 ohm resistance represents an exception and for that, we want to highlight few things about the AD8032 amplifier.

Why that one ?

It has a decent slew rate to begin with; 30V/microsecond, from the research that was made, if the slew rate is above 10V/microsecond, maybe even 6V/microsecond, the slew rate is alright.

Secondly, it can take up to 6V supply for dual channel, 12V for the single one. For our circuit we had to vary the amplitude from 1V to 6V. Having the power supply as an upper limit was a solid way to assure it's functionality from that point of view.

Now, going back to that resistance's value, the output current of the AD8032 is 15 mA, and the maximum voltage is 6. By computing the minimum resistance in order to assure our opAmp's safety, we get a value of 0.4 k Ohms. However, for safety, I considered that it's safer to pick the 0.42 k Ohms one, to take in consideration the tolerances of the resistor.

Conclusion

The PWM generator is a complex circuit, that is essential for many applications. Given certain parameters for it's functioning definitely made it more difficult to build, but not impossible. By analyzing it several times, we can observe that it's fundamental functioning is as we would expect, meaning that we are able to generate a triangular signal and with the help of a DC voltage and a comparator we're able to receive a rectangular one, and finally it works within the parameters that were given, with small variations, but small enough in order to not affect it's performance or functioning.

Bibliography

http://www.bel.utcluj.ro/dce/didactic/fec/20_nonsinusoidal_oscillators.pdf

http://www.bel.utcluj.ro/dce/didactic/fec/22_power_amplifier_class_D.pdf

https://www.circuitsgallery.com/pwm-generator-circuit-using-741-op-amp/

https://ro.mouser.com