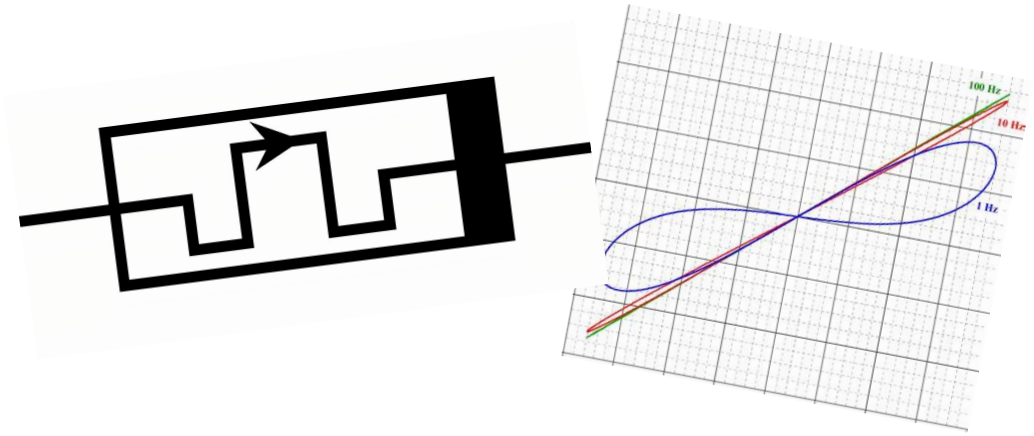
Simulation of Memristor Based Digital Logic Design



Memristor Simulation Model:

I studied several resources & articles to find an authentic model of memristor for simulation. I found this documentation [1] Memristor found: HP Labs proves fourth integrated circuit element. And more from this article [2] https://www.researchgate.net/publication/330087355_SPICE_Simulation_of_Memristor_Series_and_Parallel

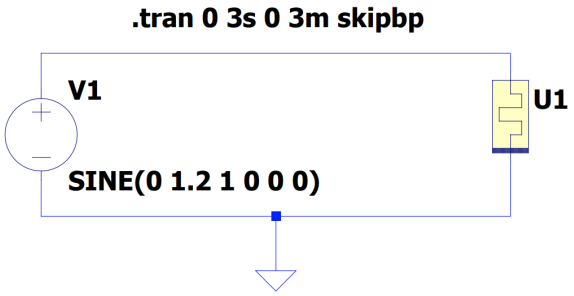
I used this information to create a subcircuit for the LTSpice model.

```
**********************************************
* LTSPICE Memristor Model
* Sourced from HP's Memristor Model Specification
* Ron, Roff - Resistance in ON / OFF States
*
* Rinit - Resistance at T=0
*
* D - Width of the thin film
*
* uv - Migration coefficient
*
* p - Parameter of the WINDOW-function for
```

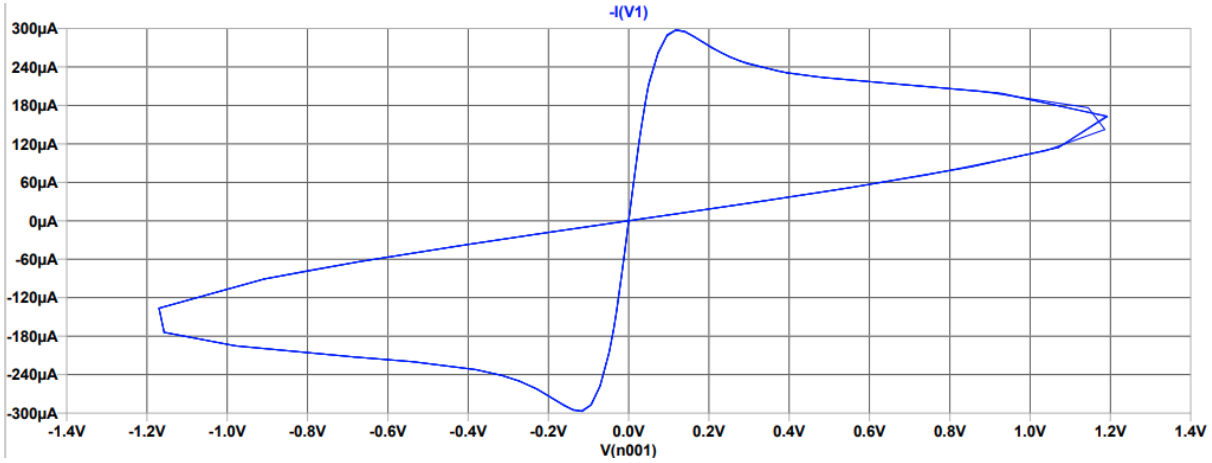
```
*     modeling nonlinear boundary conditions
*
* x - W/D Ratio, W is the actual width
*     of the doped area (from 0 to D)
*
**********************************************
.SUBCKT memristor plus minus PARAMS:
+ Ron=100 Roff=16K Rinit=11K D=10N uv=10F p=10

**********************************************
* DIFFERENTIAL EQUATION MODELING *
**********************************************
Gx 0 x value={I(Emem)*uv*Ron/D**2*f(V(x),p)}
Cx x 0 1 IC={(Roff-Rinit)/(Roff-Ron)}
Raux x 0 1T
Emem plus aux value={-I(Emem)*V(x)*(Roff-Ron)}
Roff aux minus {Roff}

**********************************************
* FLUX COMPUTATION *
**********************************************
Eflux flux 0 value={SDT(V(plus,minus))}

**********************************************
* CHARGE COMPUTATION *
**********************************************
Echarge charge 0 value={SDT(I(Emem))}

**********************************************
* WINDOW FUNCTIONS
* FOR NONLINEAR DRIFT MODELING *
**********************************************
* window function, according to Joglekar
.func f(x,p)={1-(2*x-1)**(2*p)}

* proposed window function
;.func f(x,i,p)={1-(x-sttp(-i))**(2*p)}

.ENDS memristor
```

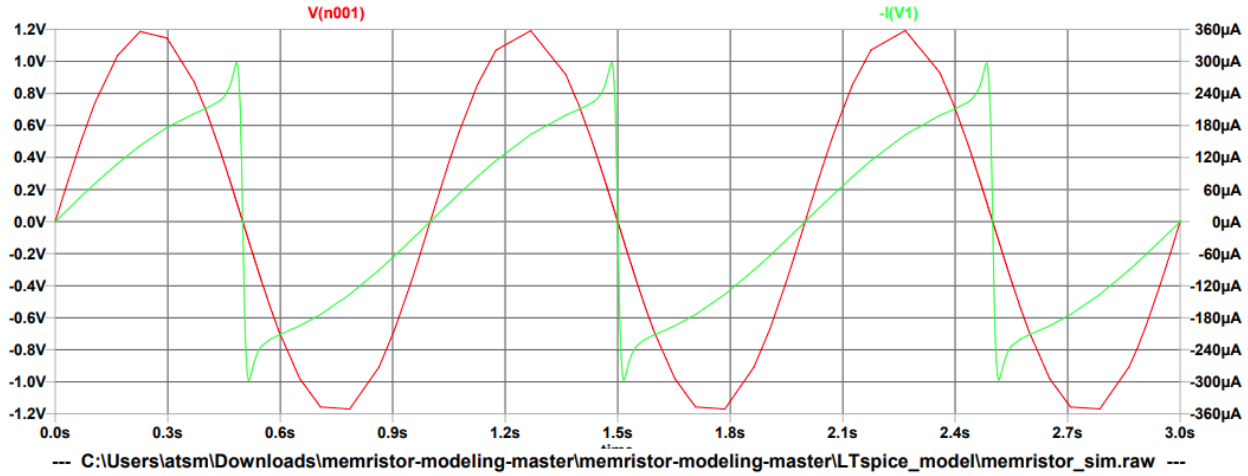
Verification of Simulation model:

I created a simple Series circuit for memristor with an AC source to obtain the hysteresis curve.



Below are the I-V and hysteresis curve for memristor model.





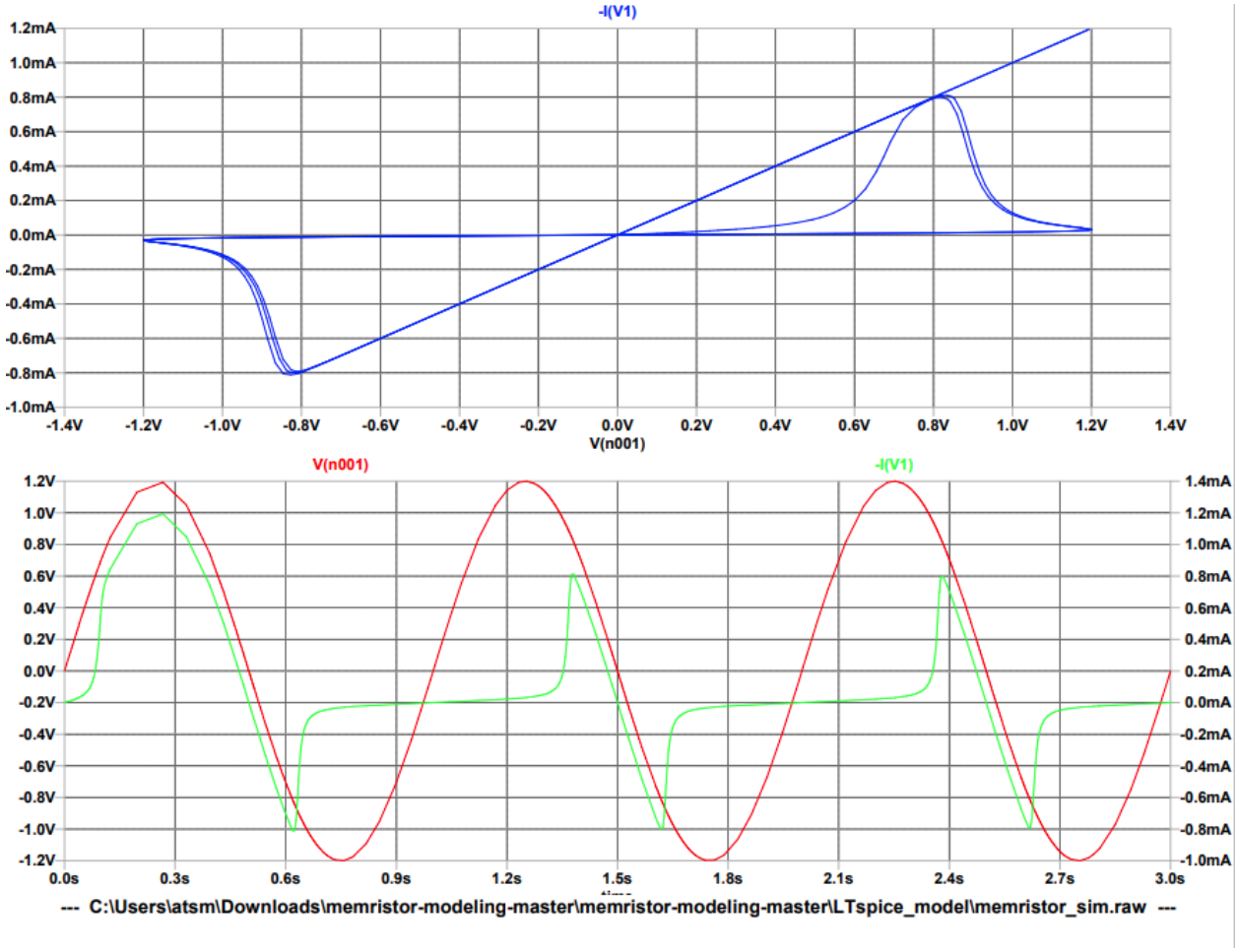
The parameters are set to Ron = 100 ohm, Roff = 16k ohm obtained from [2].

This waveform matches the results found in [2] which ensures the model is accurate & acceptable.

Memristor modeling for Digital Logic:

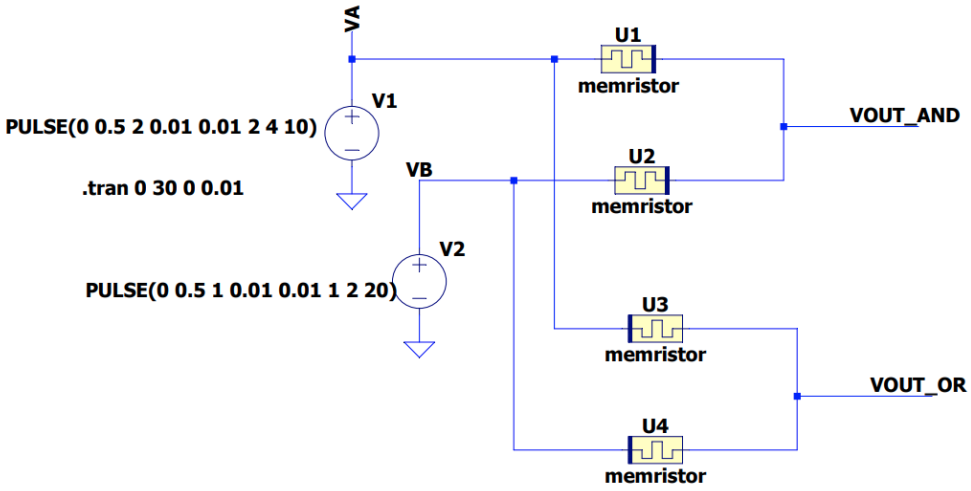
I studied more resources and found these two articles very helpful.
[3] IRJET-V8I7286.pdf
[4]https://www.researchgate.net/publication/335164336_Memristor_Based_Full_Adder_Circuit_for_Better_Performance

To simulate the digital logic, I had to change my model to a different Nonlinear Dopant Drift variant defined in [5] which is recommended by [3] and [4].
[5] ResearchGate
I reverified the model for Nonlinear Dopant Drift variant and waveform looked like this:



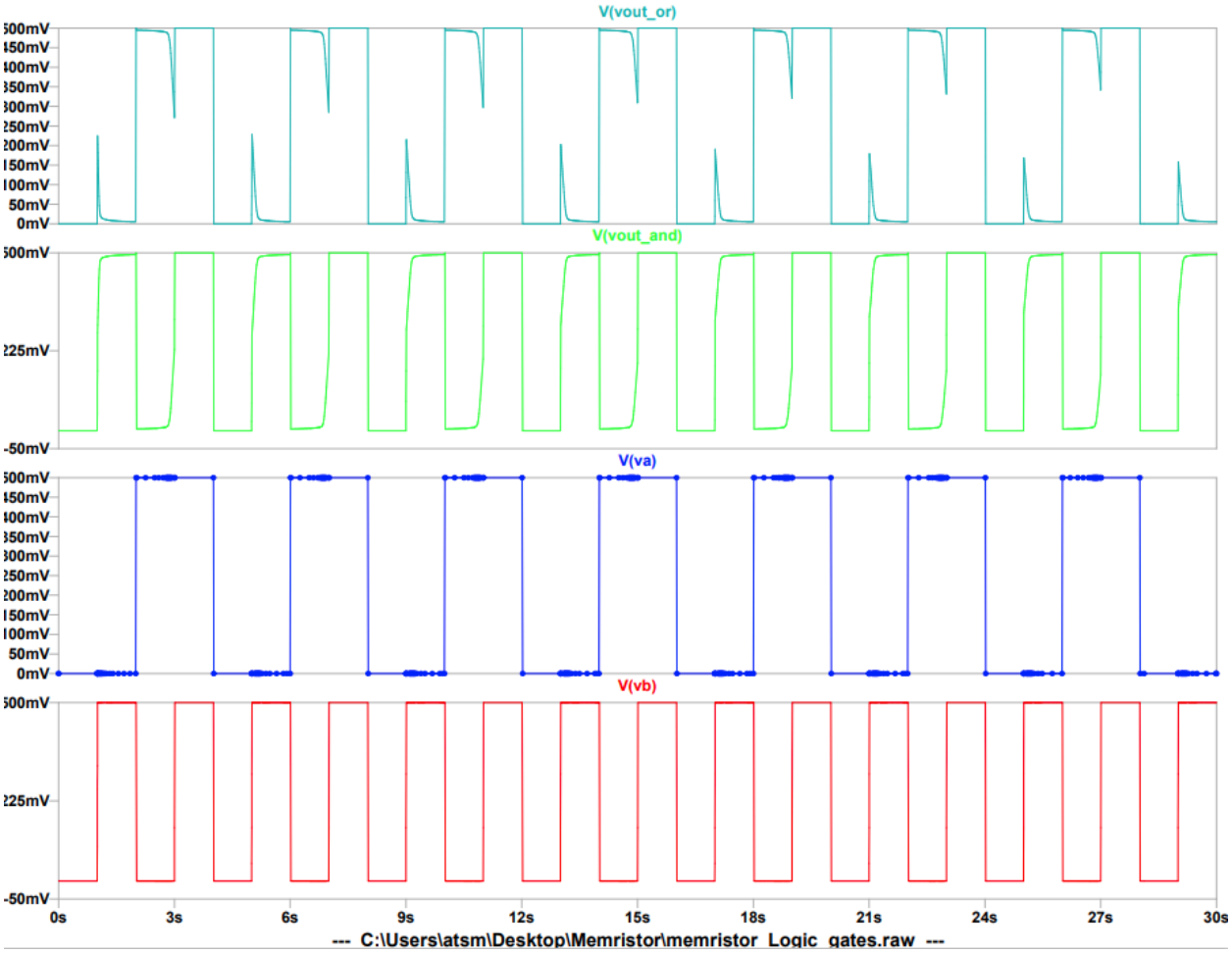
Memristor Model for AND & OR Gate:

I designed a simple AND and OR gate circuit as defined by [3] and [4] and simulated the results as below.



The waveform looks like this

Input Voltage: 0.5V; Parameters: Ron=1k Roff=100K Rinit=11K D=10N uv=10F p=10



The output for AND & OR gates is not completely accurate.

1. **A = 0, B = 0, AND = 0, OR = 0,** **INACCURATE**
2. **A = 0, B = 1, AND = 1, OR = 0 (small spike),** **INACCURATE**
3. **A = 1, B = 0, AND = 0, OR = 1,** **ACCURATE**
4. **A = 1, B = 1, AND = 1, OR = 0,** **ACCURATE**

I tried the following measures but couldn't obtain desired results:

- Changed the voltages from 0.3V to 20V. The values are very indeterminate above 5V and below 0.4V.
- Tried coupling the output with CMOS logic as defined in some of the articles (hybrid memristor architecture) but that doesn't return accurate results either.
- Tried several Ron/Roff combinations but Ron = 1k & Roff = 100k appears to be the only reasonable choice but digital logic.
- Tried to provide input with an AC signal where input crust and troughs represent 0 and 1 but no reasonable output.
- Tried a few different models I found Online from a source knowm.com. One of them is similar to mine but the rest respond differently.
- Implemented larger circuits like XNOR but still not accurate results.

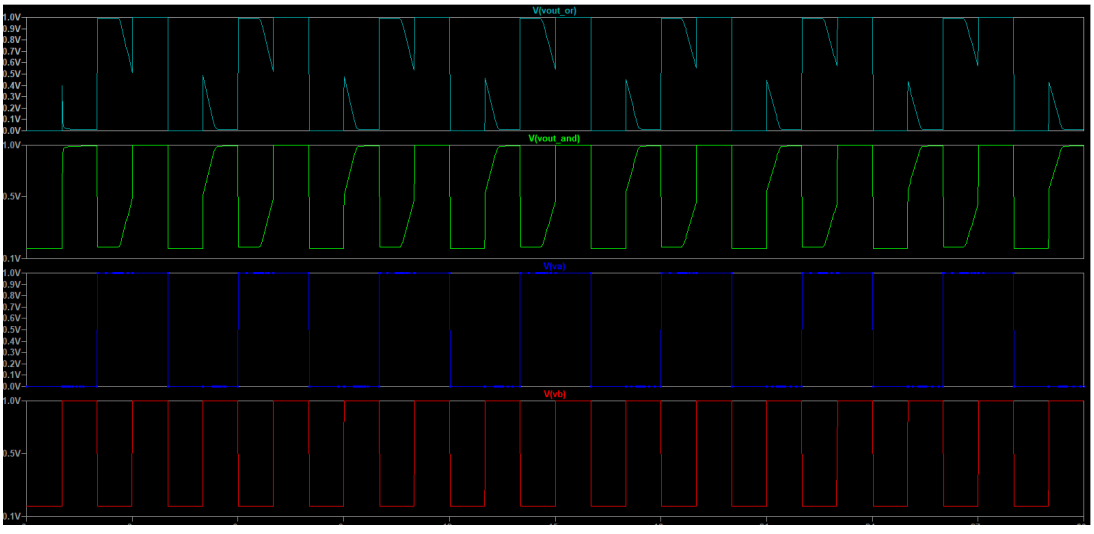
I thought of other SPICE environments but I am highly convinced that it is possible in LTSPICE and I am well familiar with it in terms of creating custom models and its simulation parameters are highly configurable.

I think the issues could be:

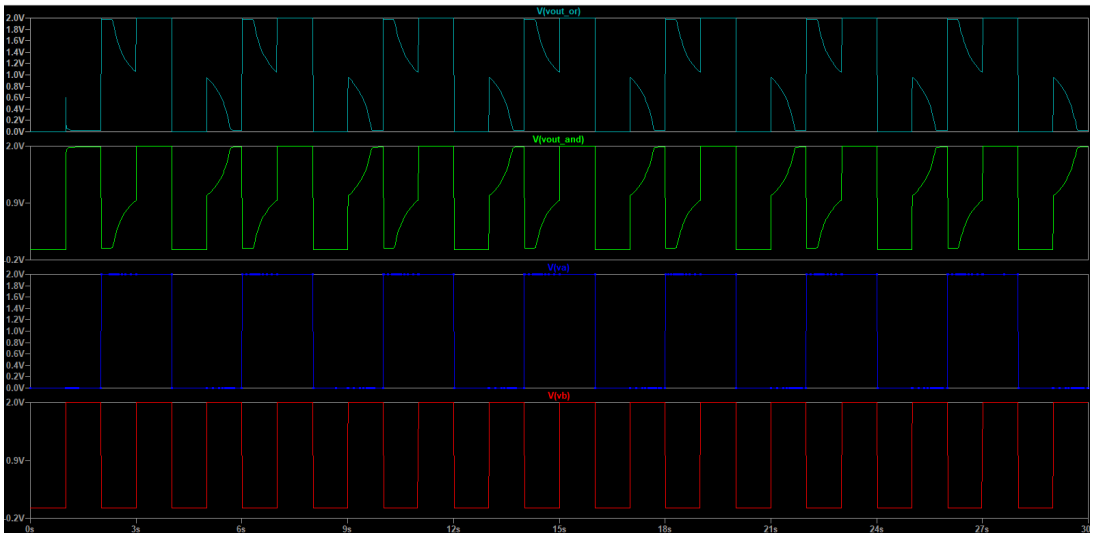
1. Differential Equation of the model is inaccurate.
2. Operating voltage or Parameters of the Memristors are not suitable for Digital Logic
3. May require some additional circuit, CMOS to accurately simulate the digital Logic.

Below are some experimental waveforms obtained with different parameters.
Red: Changed from default:

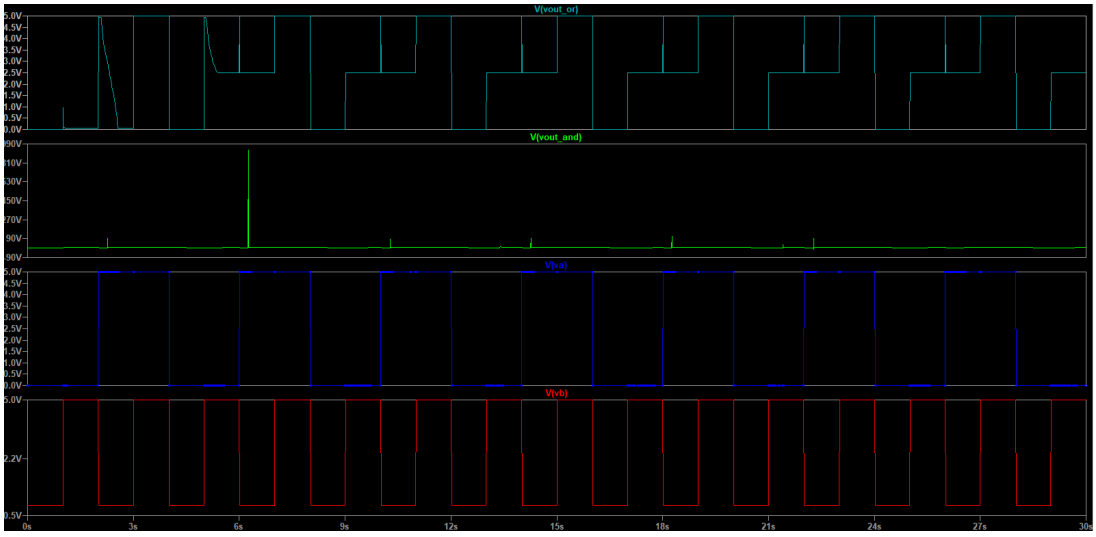
Input Voltage:1V; Parameters: Ron=1k Roff=100K Rinit=11K D=10N uv=10F p=10



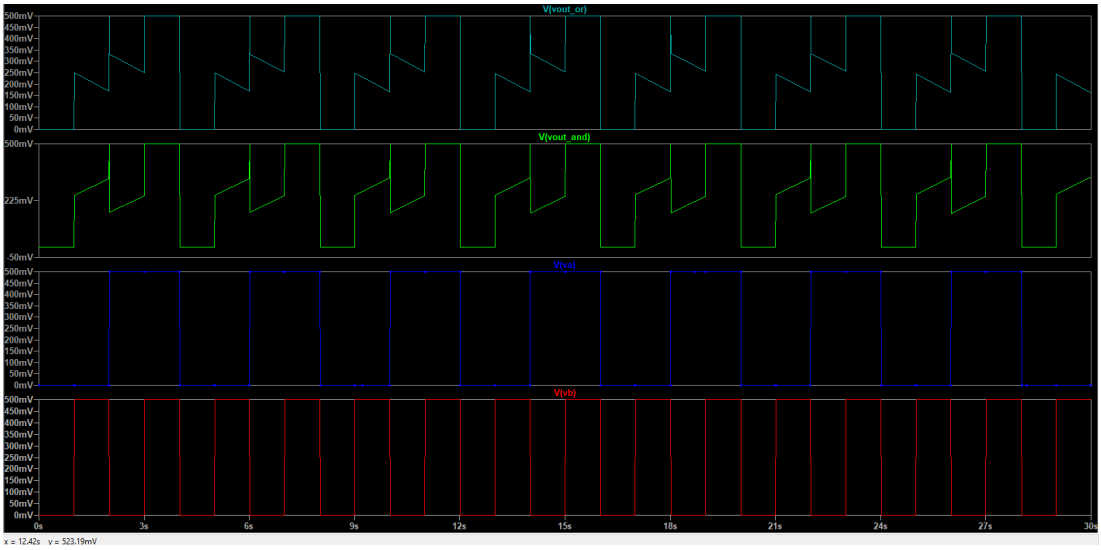
Input Voltage:2V; Parameters: Ron=1k Roff=100K Rinit=11K D=10N uv=10F p=10



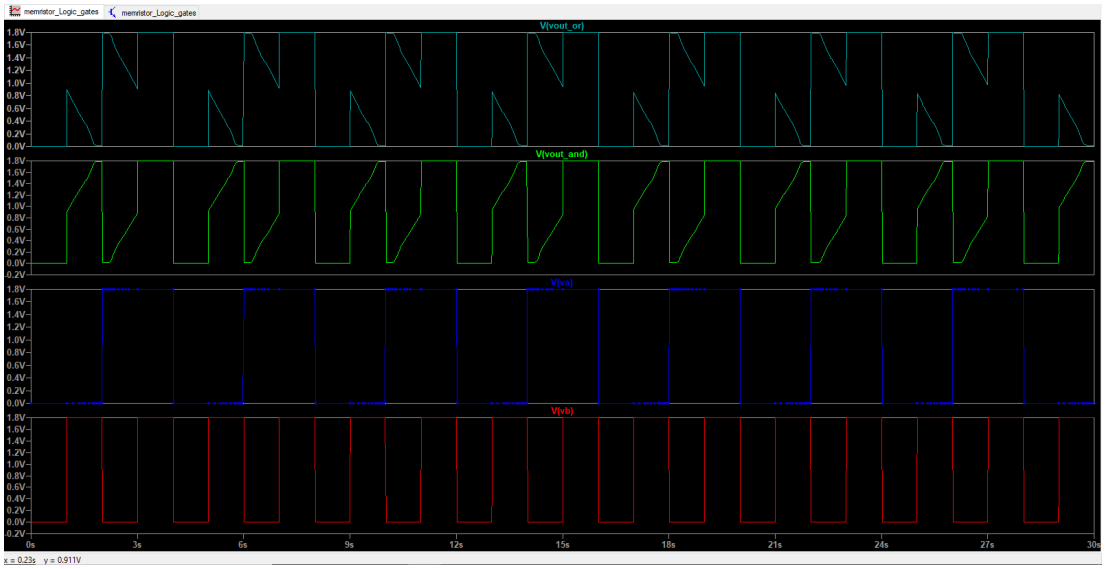
Input Voltage:5V; Parameters: Ron=1k Roff=100K Rinit=11K D=10N uv=10F p=10



Input Voltage:0.5V; Parameters: Ron=100 Roff=16K Rinit=11K D=10N uv=10F p=10



Input Voltage:1.8V; Parameters: Ron=100 Roff=16K Rinit=11K D=10N uv=10F p=10



What's strange is that, although both memristors have very similar values,
@ A = 0, B = 1, AND = 1
but
@ A = 1, B = 0, AND = 0

[4] Have successfully implemented the adder and pasted SPICE screenshots. They do not specify what SPICE tool they've used. I searched for the waveform Display windows of many common SPICE tools and found the appearance of their tool matches a lot with COSMOSCOPE which is a graphing tool used with HSPICE which is a scripting based SPICE environment. But I suspect, It would return similar results if the current model of Memristor is used.