Work Experience - Lucas Zhou-August, 2024

Project Objective:

To develop a SPICE model for a new varactor (voltage controlled capacitor) device so the company's designers can use it in their circuit simulations.

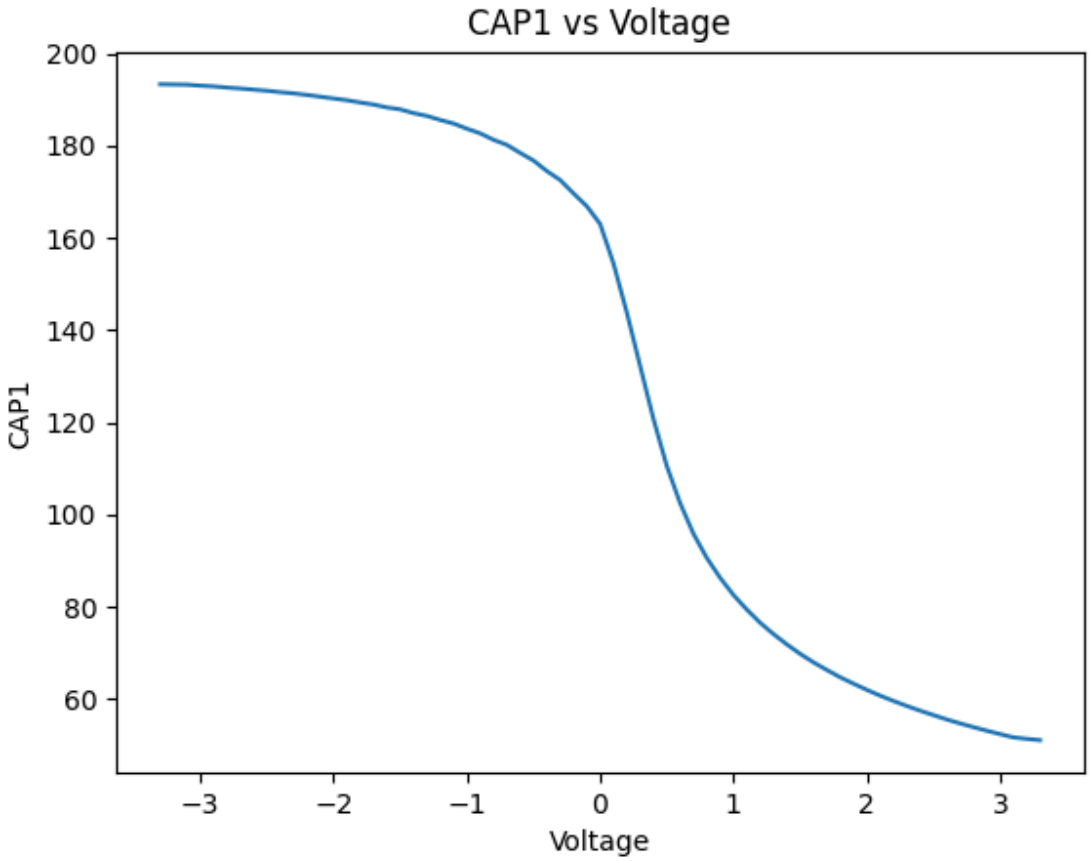
Background:

The semiconductor company Accusilicon, where I interned for, developed a new varactor device. The test engineers have done initial measurements on the device and collected data in Microsoft Excel format. I was given a job to develop a SPICE model based on the measured capacitance over voltage data. In order to create the SPICE model, I need to perform curve fitting on these data using polynomial functions, and write the fitted function in SPICE format for other designers to use.

How I did it:

First, I wrote Python code to read the measurements and plot graphs of capacitance over voltage. Next, I used a key function PolyFit to perform polynomial curve fitting using the least mean square method. After the fitting is completed, I verified the result using the PolyCalc function to evaluate the polynomials and drew the generated voltage-capacitance graph. Then, I placed it alongside the measurement graph to compare the results. After that, I output the polynomial coefficients in a SPICE subckt format as the SPICE model, and put the model in circuit simulation to verify the result against the measurement data.

Figure 1: Measured Data Graph



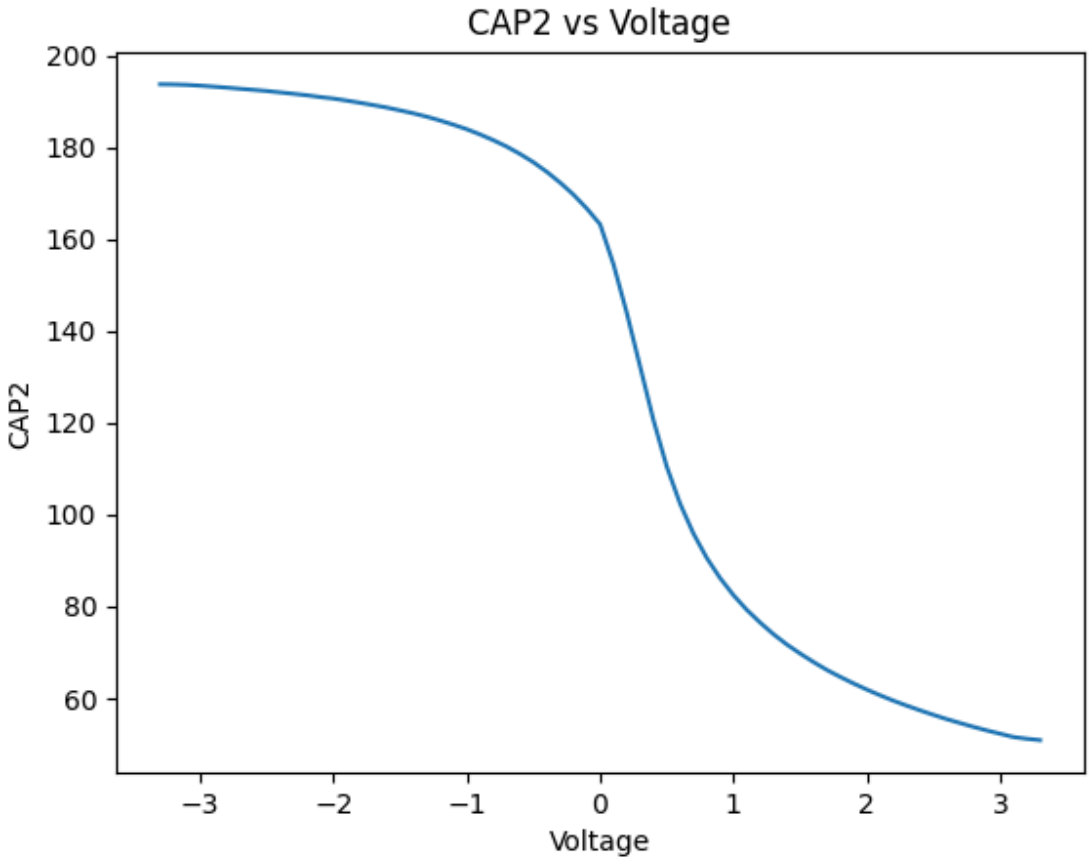


Figure 2: Measured Data Graph vs PolyCalc Graph

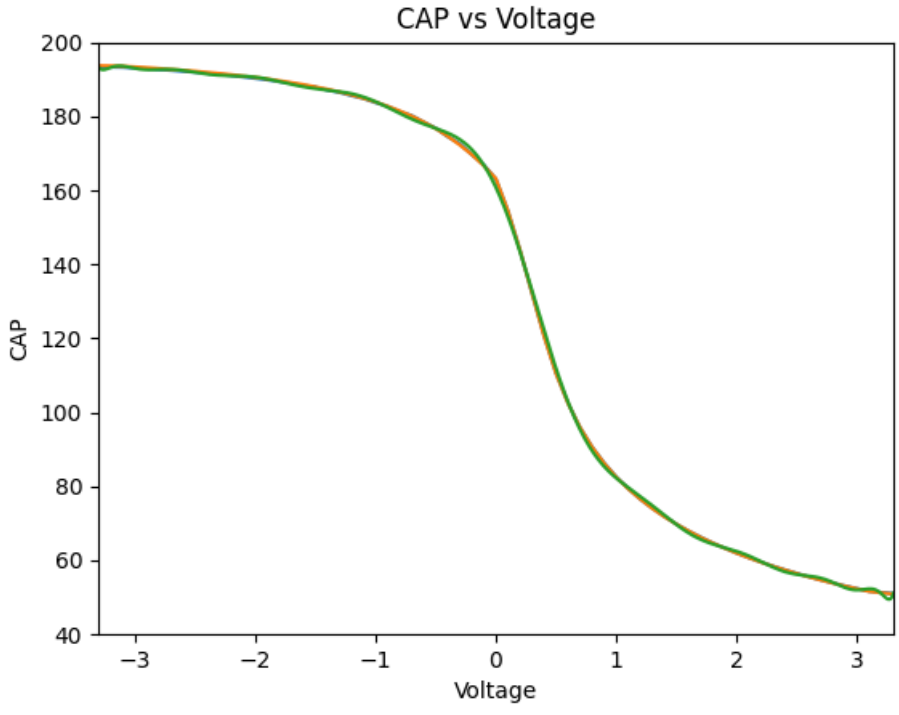
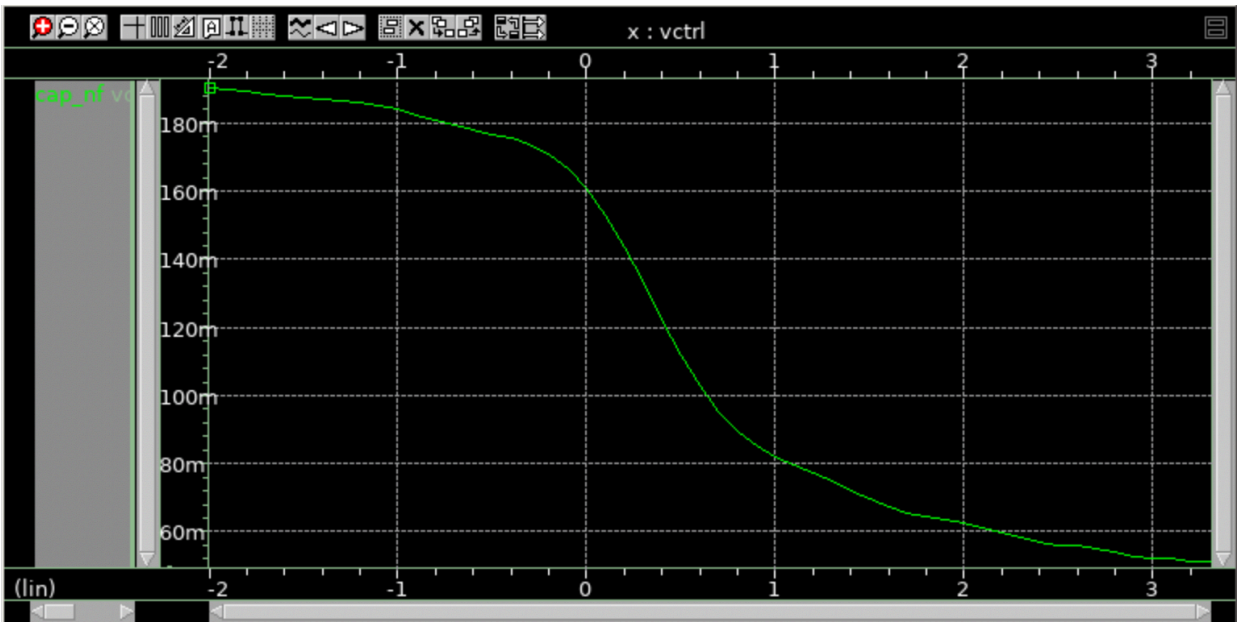


Figure 3: Generated SPICE Model (I am only allowed to publish low-order models)

```
.subckt tsmcbcd_c77_5th p n
C1 p n
+ '1.0*(146.48912766259588e-12) +
+ (v(p,n)^1.0)*(-55.064620590182666e-12) +
+ (v(p,n)^2.0)*(-7.354850241709864e-12) +
+ (v(p,n)^3.0)*(6.796721038969536e-12) +
+ (v(p,n)^4.0)*(0.5147871619410462e-12)'
.ends
```

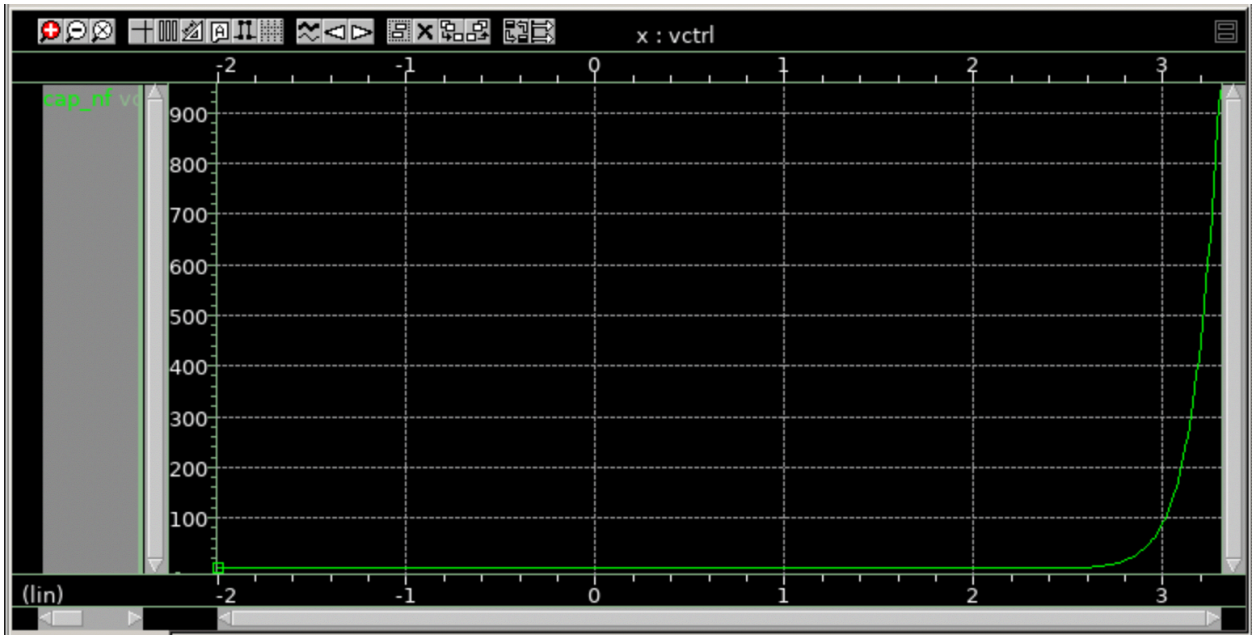
Figure 4: SPICE Simulated Graph



Challenges and difficulties and how I overcame them:

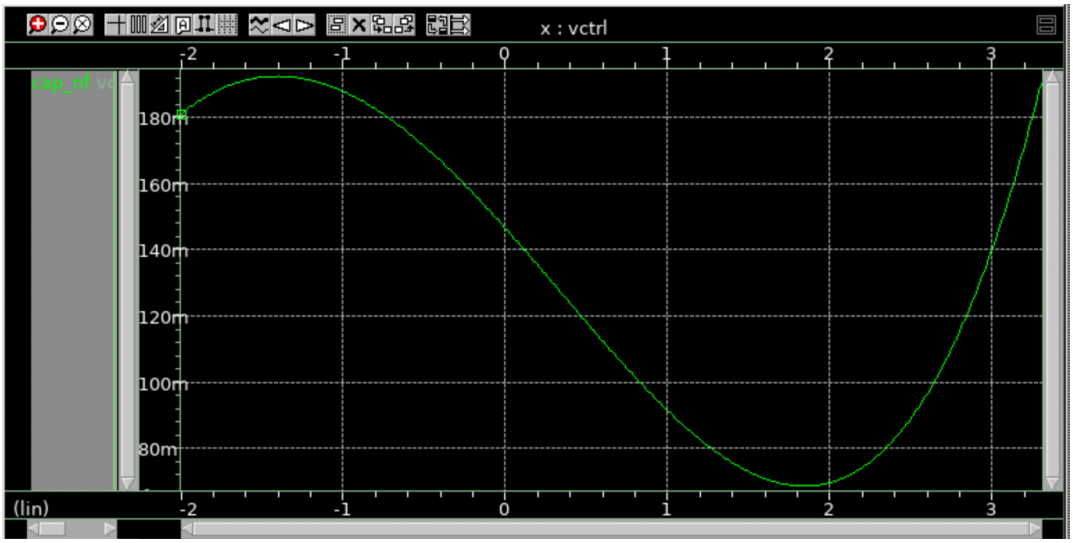
At the beginning, the curve fitting was right, but as soon as I output the polynomials into SPICE format, the circuit simulation result was wrong.

Figure 5: Wrong Circuit Simulation with 25th Order Polynomial



I tried to lower the polynomial order down to 5th order, which made the generated SPICE model perform better, although not as good as the measurement data.

Figure 6: SPICE Simulated in 5th Order



I guess the error could come from the quantization error of double precision floating point data. When the polynomial order goes very high, these errors get amplified by high exponentials and cause errors in circuit simulation.

To fix this issue, I slowly increased the curve fitting polynomial orders, put generated SPICE models into simulations, and found a sweet spot. Using 21st order polynomial functions, the generated SPICE model fit the measurement well and performed flawlessly in circuit simulation.

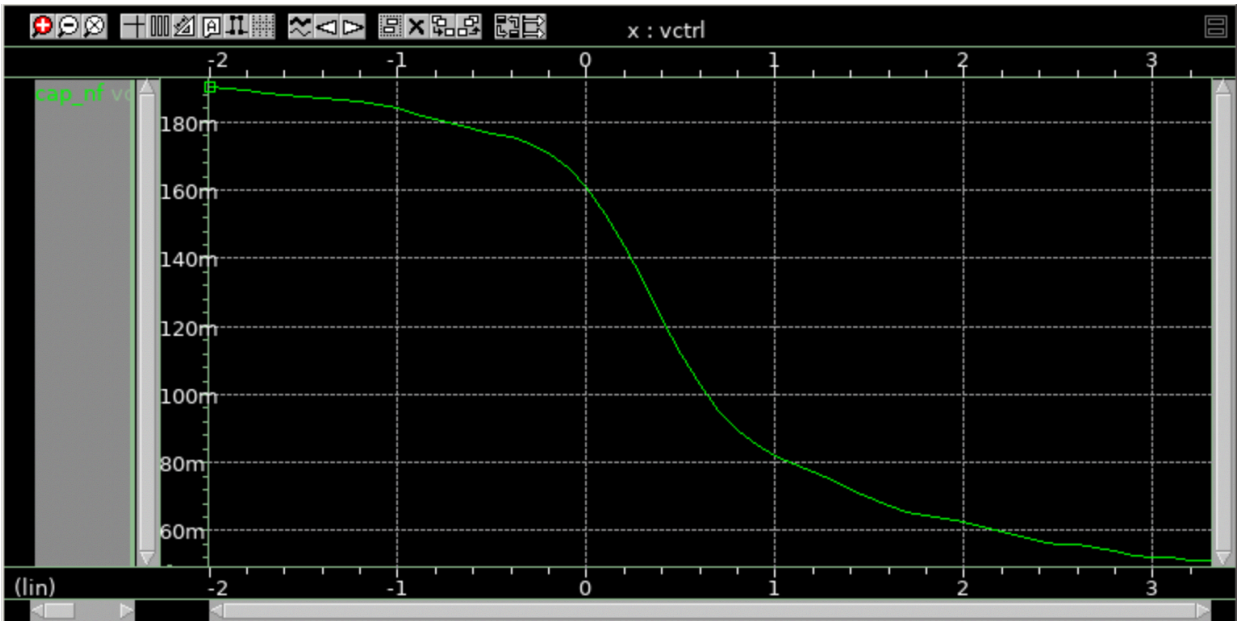



Figure 7: 21st order polynomial SPICE model Circuit Simulation Graph