

Analog and Digital Simulation

Nov. 18, 1999

The simulations will be done using the commercial software, “Electronics Workbench”, which is installed on the laboratory computers. With some preparation before the lab, you should be able to do both an analog and a digital simulation. The simulations run the circuits in software and all voltages, currents, and waveforms can be examined and printed out.

The circuits are entered by schematic capture, i.e. you draw the circuit on the computer screen by placing the parts and connecting them with wires. Once the circuit is entered, an analysis is run. There are several kinds of analyses including DC conditions, small signal transfer functions, and digital logic states for the case of digital circuits.

Part 1– Analog Simulation

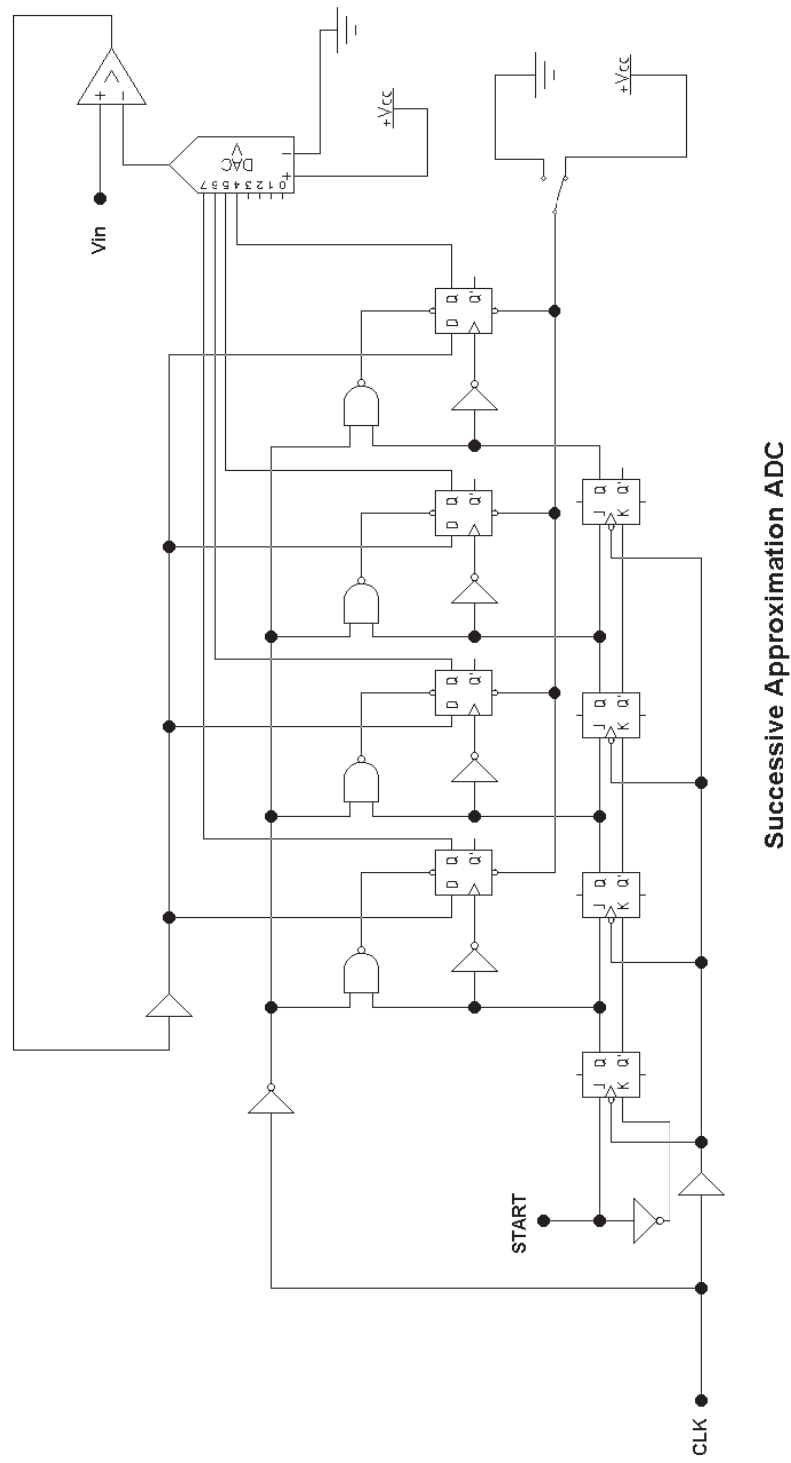
The analog simulation circuit will be the difference amplifier from the difference amplifier lab. Before the lab prepare a table of of the measured and calculated DC node voltages, the sum and difference gains, CMRR, and the amplifier high and low roll-off frequencies. You should have almost all this information in your laboratory notebook. The simulated circuit should have the same component values used in the lab circuit. The results of the simulation should then be entered as a separate column in your table.

1. Simulate the difference amplifier circuit using the circuit you built for the Difference Amplifier lab. The component values should be identical to those used in the lab. We use the Electronics Workbench application.
2. The first step is to make a schematic diagram.
3. Perform a DC analysis. Check the results against the expected values.
4. Perform a Transient analysis for 5 cycles and compute the difference and common mode gains. Use the same frequency as you used in the lab. Use the Analysis Graphs window to print out the desired waveforms.
5. Perform a frequency analysis and determine the low and high rolloff frequencies. Print out the Bode plot using the Analysis Graphs window.
6. Present the results as a three column table: theory, measurement, and simulation for the requested quantities. Most people did not measure the frequency dependence in lab, but you should be able to calculate the roll-off frequencies.
7. The final results will include the table, a copy of the circuit diagram, waveforms, and your explanations as to why the simulation might be giving different results from theory and measurement.

Part 2– Digital Simulation

You will simulate a 4 bit Successive Approximation ADC made up of a DAC, comparator, D flip flops, J-K flip flops, and NAND gates. You will enter a DC input voltage, run the simulation, and read out the digital state of the circuit under the control of CLK pulses. The circuit also includes START and RESET lines.

1. Enter the circuit via schematic capture.
2. The initial conditions for the various lines are entered using the Word Generator tool, which allows you to enter the CLK, START and RESET pulses and run them as a sequence either step by step or as an automatic sequence at a given frequency.
3. Use the Logic Analyzer tool to display the circuit output. This allows you to display the waveform on up to 16 channels simultaneously.
4. Make a plot of the DAC output as a function of successive clock pulses. You should see the DAC output converge to the analog input voltage with an accuracy consistent with the 4 bits of the circuit, i.e. one part in fifteen. The digital result resides in the four D flip flops. Try several different DC input voltages.



Successive Approximation ADC